5-Axis Function Application Manual.

匯出日期：2025-05-17
修改日期：2025-01-15

1 1. Introduction of 5-Axis Machine

The features of 5-axis machine, machine types, definition of rotary axis and related parameters of the controller will be introduced in this chapter.

1.1 1.1 5-Axis Machine Features

- A 5-axis machine includes 3 linear axis and 2 rotary axis to increase the degrees of freedom while machining.
- It's able to process the machining at the interfered area of the mechanism or on complicated surfaces, thus it has a higher acceptance on workpiece appearance (Fig.1).
- In addition, 5-axis machine also provides three advantages below:

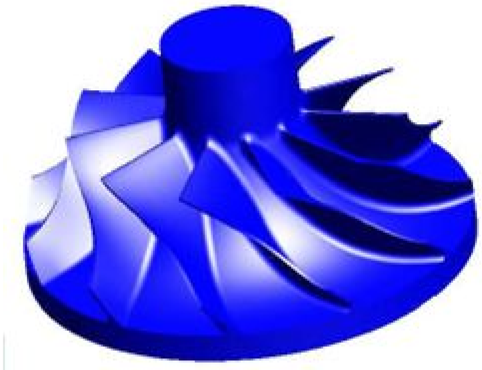


Fig.1

1.1.1 **High** Efficiency

- Normally, ball end cutters are applied when machining on curved or tilted surfaces, but the machining efficiency might drop due to poor cutting ability of the center of the cutters. For 5-axis machines, the tool angle can be adjusted according to the machining surfaces and process the machining by the part with better cutting ability. It's able to protect the tool and also improves both the efficiency and quality.

1.1.2 **High Precision**

- For workpieces with unique appearance, such as negative angles, it requires to turn over the workpiece if machined with traditional 3-axis machines, which increases the reload and reorientation time and also affects the precision. With 5-axis machines, it's able to finish the complete process without reload, which saves time and keep the precision.

1.1.3 Enhancement of Tool Rigidity

- When machining a deeper feature with 3-axis machines, it requires to elongate the tool to avoid the collision between tool holder and workpiece, thus the holding part of the tool will be reduced and the tool rigidity will drop (Fig.2). With 5-axis machines, it's able to adjust the tool angle when facing such situations, the holding part can be remained longer and the tool rigidity will be better, the precision will also be higher (Fig.3).

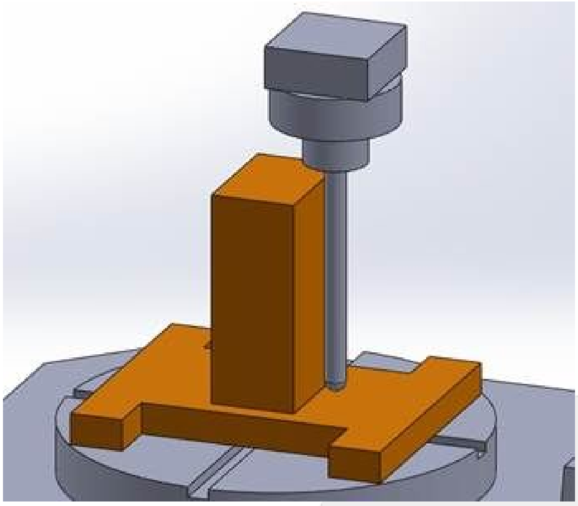


Fig.2

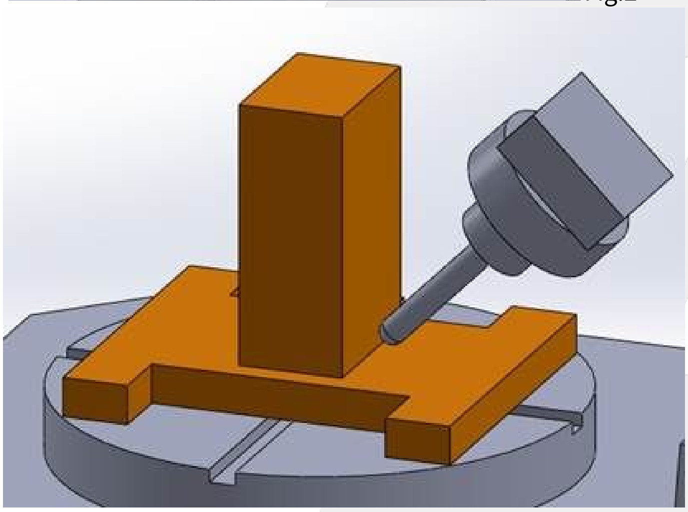


Fig.3

1.2 1.2 Machine Type

5-axis machines can be sorted into 3 types with different arrangements of rotary axis：

1. Spindle Type
2. Table Type
3. Mix Type

Shown as Fig.4

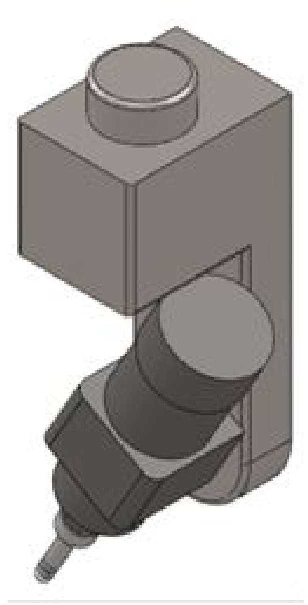
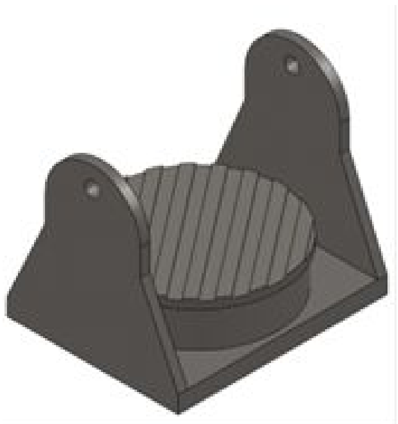
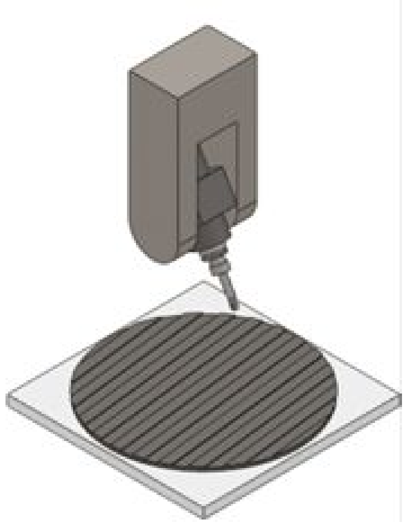
		
Spindle Type	Table Type	Mix Type

Fig.4

1.2.1 Spindle Type

Both rotary axis are on the spindle for this type of 5-axis machines.

They're normally C axis with A or B axis, but A axis with B axis is also possible for special machine types.

This type is suitable for large workpieces such as the ships or airplanes.

Since both rotary axis are on the spindle, the loading capacity of working table can be increased, thus the size of the machines are usually larger.

Also, the spindle is much lighter than the entire machine, thus the machine stability during machining can be secured.

But the manufacturing precision is required since both rotary axis are on the spindle, totally 3 axis are placed together including spindle itself.

On the other hand, the rotary axis limit the loading capacity of spindle, thus this type is not suitable for high speed feeding or heavy cutting.

1.2.2 Table Type

Both rotary axis are on the working table for this type of 5-axis machines.

They're normally A or B axis with C axis, but A axis with B axis is also possible for special machine types.

Contrary to the spindle type, this type has better spindle rigidity so it's suitable for high speed feeding and heavy cutting.

But since both rotary axis are on the working table, the workpiece weight is lighter and the size of the machine is smaller than the spindle type.

On the other hand, sine the rotary axis are on the working table, it's less flexible during the machining.

1.2.3 Mix Type

The rotary axis are on the spindle and working table separately.

They're normally A or B axis for spindle and C axis for table, but A or B axis on the table is also possible for special machine types.

For properties such as flexibility, machine size, workpiece weight, manufacturing precision, this type lands between the previous 2 types.

But since it requires lower manufacturing precision, it has advantage on the cost.

4-axis machines can be sorted into 2 types with different arrangements of rotary axis：

- Single Spindle Type
- Single Table Type

Shown as Fig.5

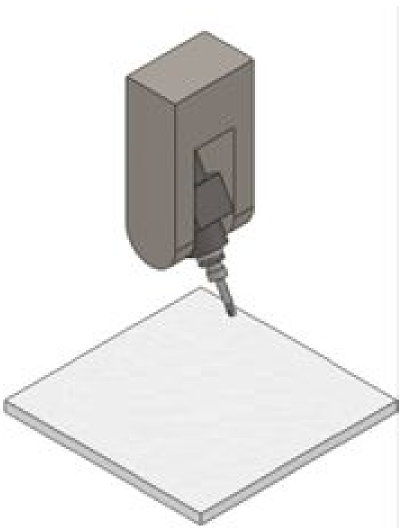
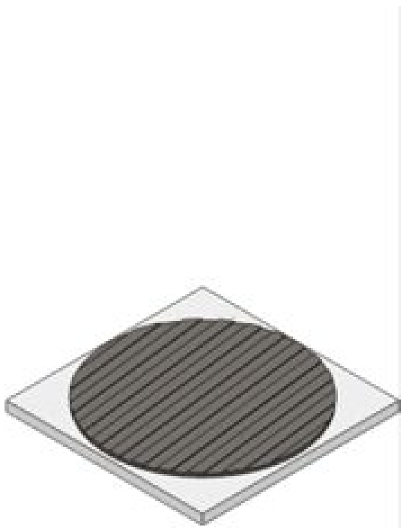
	
Single Spindle Type	Single Table Type

Fig.5

1.2.4 Single Spindle Type

This type is suitable for large workpieces, since the rotary axis is on the spindle, the loading capacity of working table can be increased.

Also, the spindle is much lighter than the entire machine, thus the machine stability during machining can be secured.

1.2.5 Single Table Type

This type has higher flexibility, better spindle rigidity and it's suitable for high speed feeding and heavy cutting.

But since the rotary axis is on the working table, the workpiece weight is lighter.

1.3 1.3 Definitions of Rotary Axis

Since the two rotary axis in 5-axis machines have different mechanism arrangements, we define the relations between rotary axis with Master & Slave for controllers to calculate precisely.

Correct definitions and settings are required or the calculation could be wrong and leads to abnormal moving paths.

Master axis is also called the fist rotary axis; Slave axis is also called the second rotary axis.

The Master-Slave relation of rotary axis in these 3 types will be introduced below:

1.3.1 Spindle Type

Shown as Fig.6, the second axis is attached to the first axis.

For actual operation, the rotation of second axis will not affect the posture of first axis; in opposite, the rotation of first axis will.

Therefore, it's able to identify the fist axis.

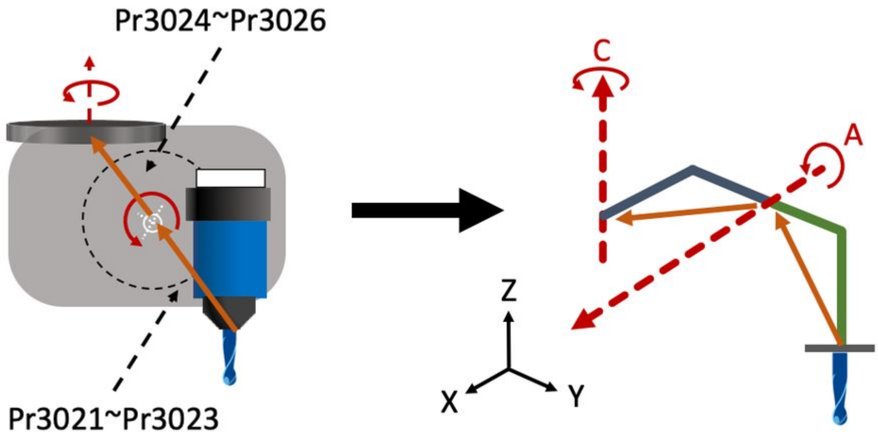


Fig.6

1.3.2 Table Type

Same as the spindle type, it's able to identify the first and second axis for table type machines, shown as Fig.7.

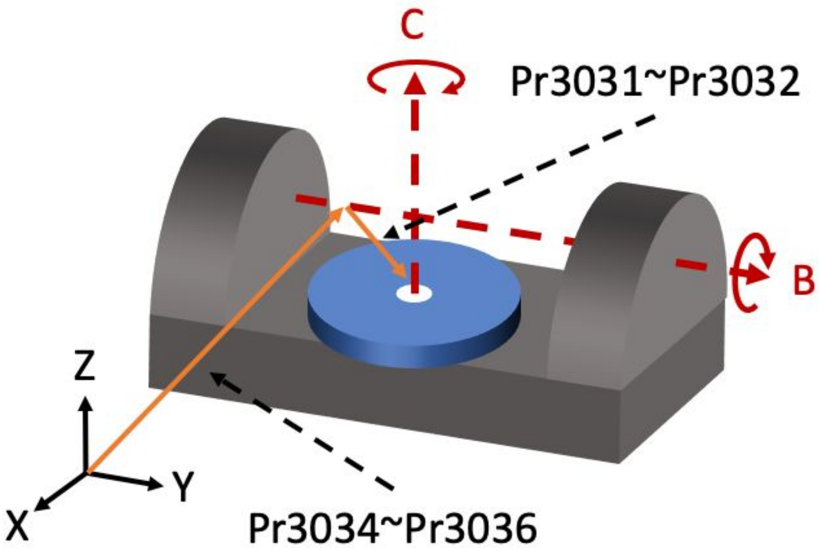


Fig.7

1.3.3 Mix Type

The Master-Slave relation of this type can't be clearly defined, it's normally defined from top to bottom.

The rotary axis on spindle is defined as first axis and the one on working table is defined as second axis, shown as Fig.8.

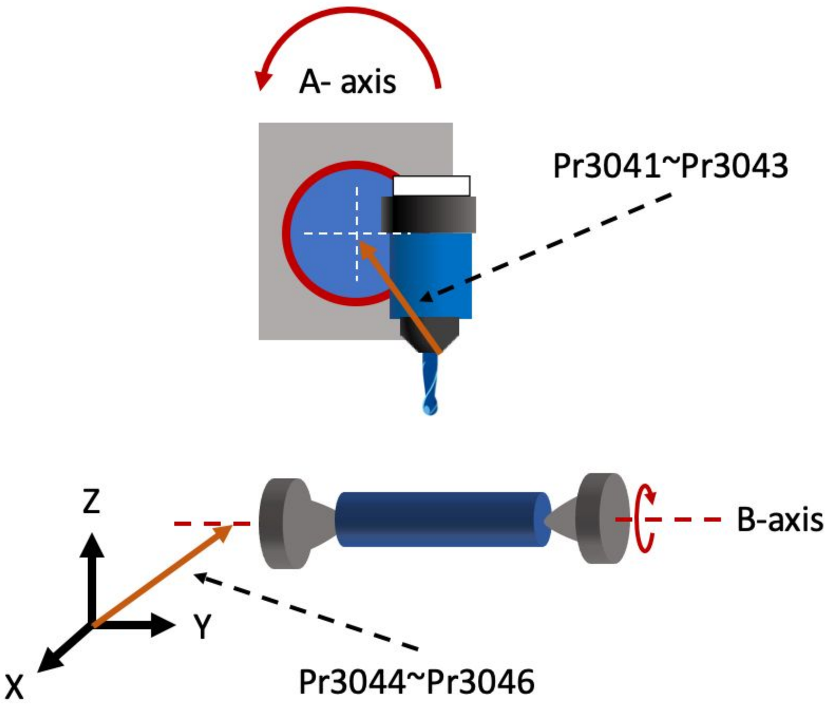


Fig.8

1.4 1.4 Parameter Descriptions

The parameters related to 5-axis function will be introduced in this section, including the definitions and effective time after modification.

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3001		*1st Organization for five axis machine	[0,5]	-	0	Restart
V12	P4001	N1~N4					

This parameter defines the mechanism arrangement of the 5-axis machine.

Definitions:

- 0: not 5-axis machine
- 1: Spindle Type
- 2: Table Type
- 3: Mix Type
- 4: Single Spindle Type
- 5: Single Table Type

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3002		1st Direction of Tool	[0,3]	-	0	Reset
V12	P4002	N1~N4					

This parameter defines the direction of tool (from tool tip to tool holder) when the angle of rotary axis is 0.

- 0： Undefine
- 1： Positive X-Axis
- 2： Positive Y-Axis
- 3： Positive Z-Axis

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3003		1st Incline Angle of Direction of Tool(RA)	[0,3600000]	BLU	0	Reset
V12	P4003	N1~N4					

Definitions:

- Tool vector: The vector from tool tip to tool holder.
- Rotation direction: The angle for this parameter is decided by right-hand rule along the axis.
- Order of RA & RB : Tool vector can be any vector in space, and this vector can be obtained by RA first then RB from default direction of tool.

Description:

This parameter is used to correct the angle error from installation of the spindle or tool.

- When Pr3002 is 1
  Pr3003 represents the positive angle between tool direction projection on XY plane and positive X axis.
- When Pr3002 is 2
  Pr3003 represents the positive angle between tool direction projection on YZ plane and positive Y axis.
- When Pr3002 is 3
  Pr3003 represents the positive angle between tool direction projection on ZX plane and positive Z axis.

Take Pr3002 = 3 as an example.

The definition of Pr3003 (RA) is shown as the figure 9.

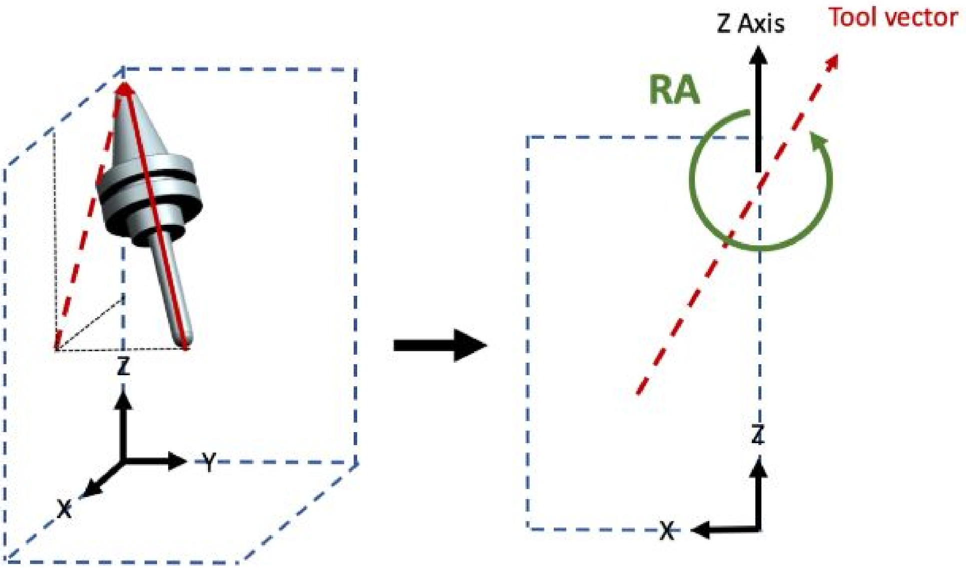


Figure 9

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3004		1st Incline Angle of Direction of Tool(RB)	[0,3600000]	BLU	0	Reset
V12	P4004	N1~N4					

Definitions:

- Tool vector: The vector from tool tip to tool holder.
- Rotation direction: The angle for this parameter is decided by right-hand rule along the axis.
- Order of RA & RB : Tool vector can be any vector in space, and this vector can be obtained by RA first then RB from default direction of tool.

Description:

After RA is defined, the result could be used to get RB and finally align to the actual tool vector.

- When Pr3002 is 1:
  Pr3004 represents the positive angle between the tool vector of RA and the tool direction projection on YZ plane.
- When Pr3002 is 2:
  Pr3004 represents the positive angle between the tool vector of RA and the tool direction projection on ZX plane.
- When Pr3002 is 3:
  Pr3004 represents the positive angle between the tool vector of RA and the tool direction projection on XY plane.

Take Pr3002 = 3 as an example.

The definition of Pr3004 (RB) is shown as the figure 10.

The vector rotated by RA is now projected onto XY plane, then the vector rotates an RB angle along positive Z direction to align with the projection of tool vector on XY plane.

Figure 10

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3005		1st first rotation axis	[0,3]	-	0	Reset
V12	P4005	N1~N4					
V10	Pr3006		1st second rotation axis	[0,3]	-	0	Reset
V12	P4006	N1~N4					

This parameter defines which coordinate axis the rotary axis is rotating around (figure 11).

Description:

- 0: Undefine
- 1: Rotate around X-Axis
- 2: Rotate around Y-Axis
- 3: Rotate around Z-Axis

Figure 11

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3007		1st rotation direction of first rotation axis	[0,2]	-	0	Reset
V12	P4007	N1~N4					
V10	Pr3008		1st rotation direction of second rotation axis	[0,2]	-	0	Reset
V12	P4008	N1~N4					

Description:

- 0: Undefine
- 1: Right-hand rule
- 2: Left-hand rule

How to determine：

- Point your thumb to positive axis direction and the other 4 fingers point out the positive rotation direction.
- Check if the rotation direction of the rotary axis matches to right-hand rule or left-hand rule.

According to ISO-230 standard, both rotary axis of spindle type must follow right-hand rule, and those of table type must follow left-hand rule (as shown in figure 12).

However, it is more accurate to make the judgement at the scene.

Figure 12

Version	No	Descriptions	Range	Unit	Default	Effective

V10	Pr3009		1st starting point of rotation angle of first rotation axis	[0,3600000]	BLU	0	Reset
V12	P4009	N1~N4					
V10	Pr3010		1st terminal point of rotation angle of first rotation axis	[0,3600000]	BLU	0	Reset
V12	P4010	N1~N4					
V10	Pr3011		1st starting point of rotation angle of second rotation axis	[0,3600000]	BLU	0	Reset
V12	P4011	N1~N4					
V10	Pr3012		1st terminal point of rotation angle of second rotation axis	[0,3600000]	BLU	0	Reset
V12	P4012	N1~N4					

Description:

The direction of this parameter is determined by Pr3007 & Pr3008.

If there is an alarm related to angle range, please check if the direction judgement is correct.

Assume that right-hand rule is used and take A axis as an example.

First, point your right thumb to positive X axis direction, and then look from positive X to negative X, as shown in figure 13;

The other 4 fingers point out the positive rotation direction.

According to the positive direction and the actual operation angle of the rotary axis, the parameters could be defined, as shown in figure 14.

While using control precision of um(Pr17=2), if Area 1 is the operation range of the rotary axis, starting point is 290 degree then Pr3011=290000; terminal point is 70 degree then Pr3012=70000.

While using control precision of um(Pr17=2), if Area 2 is the operation range of the rotary axis, starting point is 70 degree then Pr3011=70000; terminal point is 290 degree then Pr3012=290000.

Please notice that when left-hand rule is used, the positive direction will reverse.

For example, 90 degree under right-hand rule becomes 270 degree under left-hand rule; 270 degree under right-hand rule becomes 90 degree under left-hand rule.

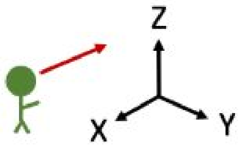


figure 13

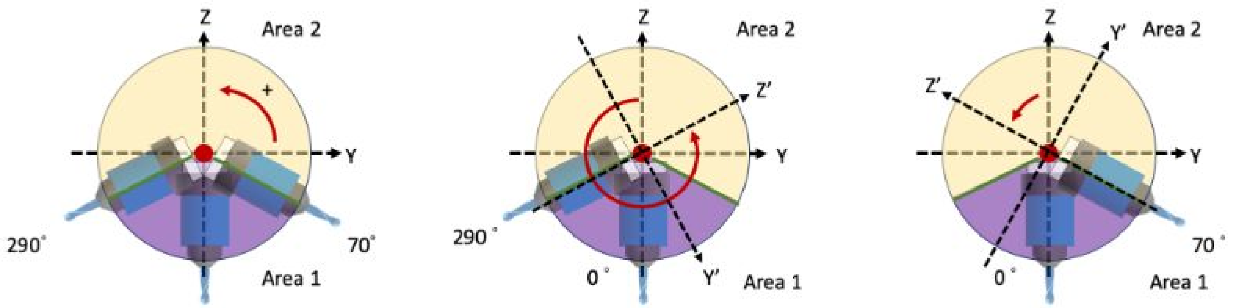


figure 14

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr 3013		1st Tool Holder Offset	[0,999999999]	BLU	0	Reset
V12	P4057	N1~N4					

Description:

This parameter is valid only when there is rotary axis on spindle side.

Tool holder offset means the distance from control point (center of rotary axis) to spindle tip (tool is not included).

It can be measured by simple process, the specific definition of tool holder offset and tool length can refer to figure 15.

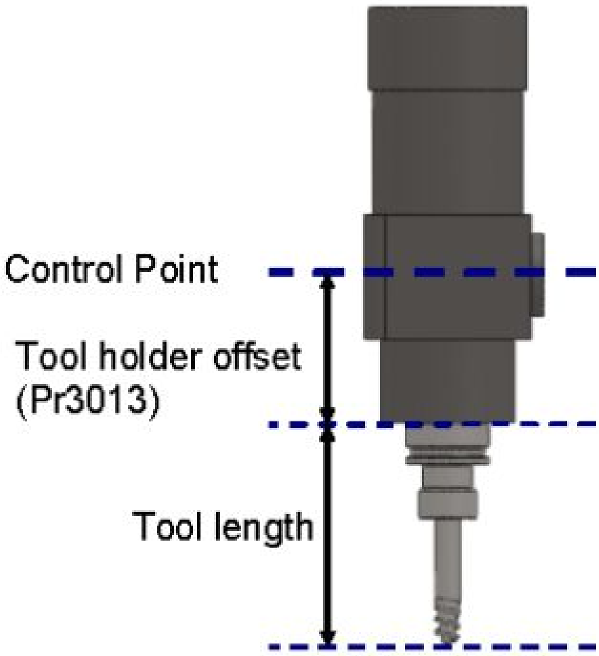


Figure 15

Version	No		Descriptions	Range	Unit	Details	Effective
V10	Pr3014		Feature coordinate persist mode	[0,2]	-	0: Do NOT preserve feature coordinate status defined by G68.2/G68.3 after reset & reboot. 1: Preserve feature coordinate status defined by G68.2/G68.3 after reset only. 2: Preserve feature coordinate status defined by G68.2/G68.3 after reset and reboot.	Reset
V12	P902	N1					

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3015		1st A-component of offset for first rotation axis	[-3600000,3600000]	BLU	0	Reset
V12	P4013	N1~N4					
V10	Pr3016		1st B-component of offset for first rotation axis	[-3600000,3600000]	BLU	0	Reset
V12	P4014	N1~N4					

V10	Pr3017		1st C-component of offset for first rotation axis	[-3600000,3600000]	BLU	0	Reset
V12	P4015	N1~N4					
V10	Pr3018		1st A-component of offset for second rotation axis	[-3600000,3600000]	BLU	0	Reset
V12	P4016	N1~N4					
V10	Pr3019		1st B-component of offset for second rotation axis	[-3600000,3600000]	BLU	0	Reset
V12	P4017	N1~N4					
V10	Pr3020		1st C-component of offset for second rotation axis	[-3600000,3600000]	BLU	0	Reset
V12	P4018	N1~N4					

Description:

Pr3015~Pr3020 are used to compensate the error when the rotary axis are not orthogonal to XYZ axis.

According to Pr3005 & Pr3006, each component of offset should be measured and entered in Pr3015~Pr3020.

For example:

If first rotary axis is C axis (Pr3005=3), then AOC, BOC, COC in figure 16 are corresponding to Pr3015~Pr3017.

Take C axis as example to explain how to define the sign of the offset.

The offset, AOC, is the angle between the projection vector of C axis on YZ plane and positive Z axis.

The sign of AOC is decided by right-hand rule along A axis, so the sign of AOC in figure 16 is negative.

The offset, BOC, is the angle between the projection vector of C axis on XZ plane and positive Z axis.

The sign of BOC is decided by right-hand rule along B axis, so the sign of BOC in figure 16 is positive.

The offset, COC, is the offset of the origin point, and the sign of COC is also decided by right-hand rule.

If this offset exists, it's recommended to reset the origin point of the rotary axis, as shown in figure 17.

Example of parameter setting

First rotary axis is C and second rotary axis is B (Pr3005=3, Pr3006=2), ∠COC=50 degree, ∠BOB=30 degree.

While using control precision of um(Pr17=2),

C component of first rotary axis is Pr3017 = 50000 (BLU)

B component of second rotary axis is Pr3019 = 30000 (BLU)

Figure 16

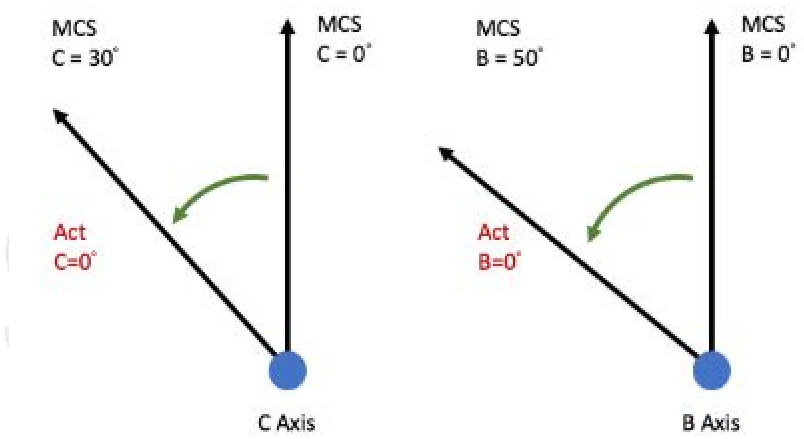


Figure 17.

*Details of AOC, BOC and COC mentioned above, please refers to chapter 5.

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3021		1st X-component of offset from tool holder to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4051	N1~N4					
V10	Pr3022		1st Y-component of offset from tool holder to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4052	N1~N4					
V10	Pr 3023		1st Z-component of offset from tool holder to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4053	N1~N4					

Description:

Pr3021~Pr3023 are used to set up the mechanism chain of the 5-axis machine for spindle type (see figure 19).

If the offset vector from tool axis to second rotary axis is (a,b,c), then
a is Pr3021, b is Pr3022, c is Pr3023.

Recommend Setting

If second rotary axis is

1. A axis, then (a,b,c) = (0, Y component from tool holder to A axis, Z component from tool holder to A axis)
2. B axis, then (a,b,c) = (X component from tool holder to B axis, 0, Z component from tool holder to B axis)

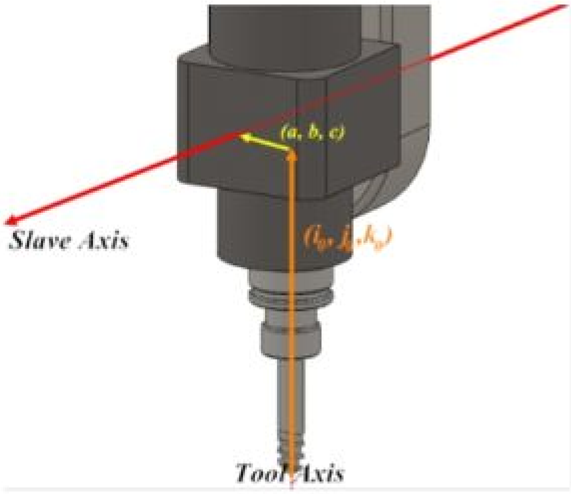


Figure 19

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3024		1st X-component of offset from second rotation axis to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4054	N1~N4					
V10	Pr3025		1st Y-component of offset from second rotation axis to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4055	N1~N4					
V10	Pr3026		1st Z-component of offset from second rotation axis to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4056	N1~N4					

Description:

Pr3024 ~ 3026 are used to set up the mechanism chain of the 5-axis machine for spindle type (see figure 20).

If the offset vector from second rotation axis to first rotation axis is (d, e, f), then
d is Pr3024, e is Pr3025, f is Pr3026.

Recommend Setting

Following the conditions of Pr3021~Pr3023, the offset vector from second rotary axis to first rotary axis is

(1) C axis + (2) A axis: (d,e,f) = (X component from A axis to C axis, Y component from A axis to C axis, 0)

(1) C axis + (2) B axis: (d,e,f) = (X component from B axis to C axis, Y component from B axis to C axis, 0)

(1) B axis + (2) A axis: (d,e,f) = (X component from A axis to B axis, 0, Z component from A axis to B axis)

(1) A axis + (2) B axis: (d,e,f) = (0, Y component from B axis to A axis, Z component from B axis to A axis)

Figure 20

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3031		1st X-component of offset from first rotation axis to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4031	N1~N4					
V10	Pr3032		1st Y-component of offset from first rotation axis to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4032	N1~N4					
V10	Pr3033		1st Z-component of offset from first rotation axis to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4033	N1~N4					

Description:

Pr3031~3033 are used to set up the mechanism chain of the 5-axis machine for table type (see figure 21).

If the offset vector from first rotary axis to second rotary axis is (a, b, c), then
a is Pr3031, b is Pr3032, c is Pr3033.

Recommend Setting

Following the conditions of Pr3034~Pr3036, the offset vector from first rotary axis to second rotary axis is

(1) A axis + (2) C axis: set (a,b,c) = (X component from A axis to C axis,Y component from A axis to C axis, 0)

(1) B axis + (2) C axis: set (a,b,c) = (X component from B axis to C axis, Y component from B axis to C axis, 0)

(1) A axis + (2) B axis: set (a,b,c) = (X component from A axis to B axis, 0, Z component from A axis to B axis)

(1) B axis + (2) A axis: set (a,b,c) = (0, Y component from B axis to A axis, Z component from B axis to A axis)

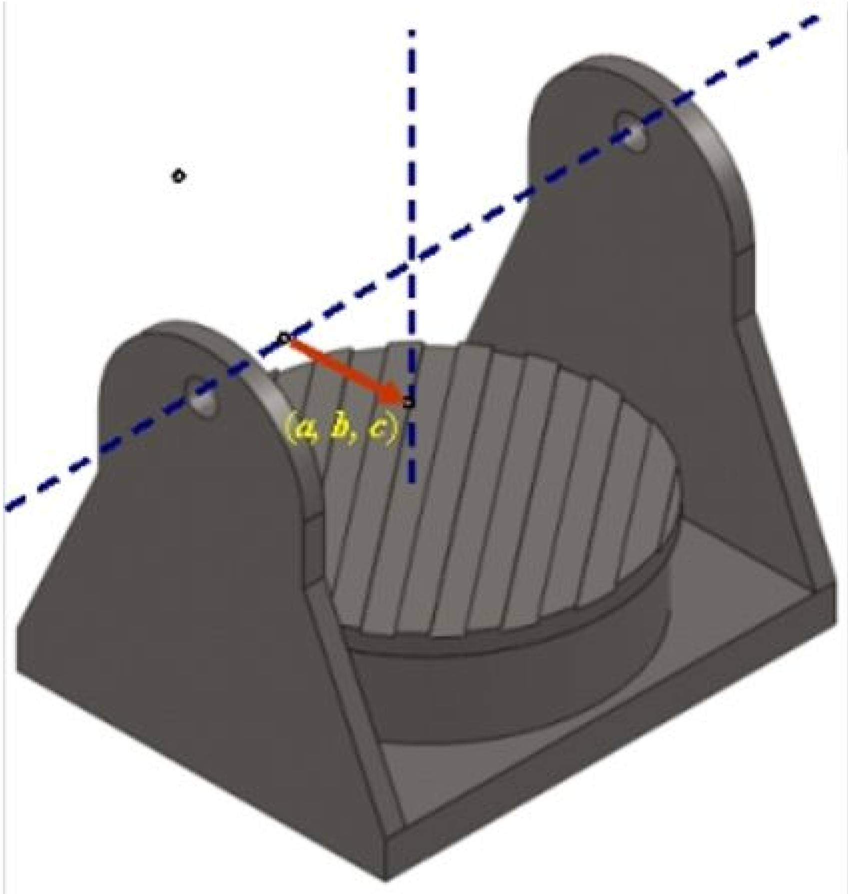


Figure 21

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3034		1st X-component of offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4034	N1~N4					
V10	Pr3035		1st Y-component of offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4035	N1~N4					
V10	Pr3036		1st Z-component of offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4036	N1~N4					

Description:

Pr3034~3036 are used to set up the mechanism chain of the 5-axis machine for table type (see figure 22)

If the offset vector from arbitrary position on the machine to first rotary axis is (d, e, f), then
d is Pr3034, e is Pr3035, f is Pr3036.

Machine origin is usually chosen as the reference point, so d, e, f are the machine coordinate of the rotary axis.

Recommend Setting

If first rotary axis is

1. A axis, then (d,e,f) = (0, Y component of the machine coordinate, Z component of the machine coordinate)
2. B axis, then (d,e,f) = (X component of the machine coordinate, 0, Z component of the machine coordinate)

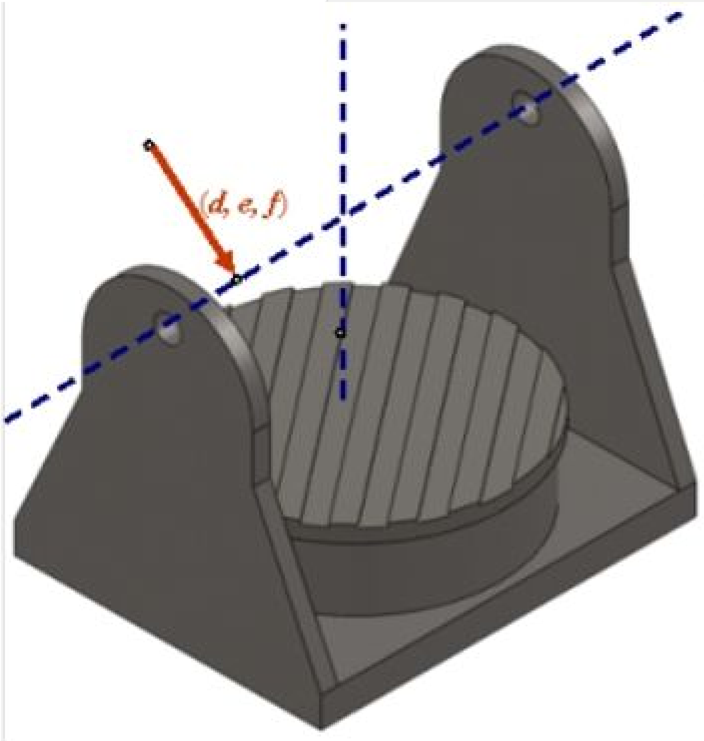


Figure 22

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3041		1st X-component of offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4041	N1~N4					

V10	Pr3042		1st Y-component of offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4042	N1~N4					
V10	Pr3043		1st Z-component of offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4043	N1~N4					

Description:

Pr3041~3043 are used to set up the mechanism chain of the 5-axis machine for mix type (see figure 23)

If the offset vector from tool holder to first rotary axis is (a, b, c), then
a is Pr3041, b is Pr3042, c is Pr3043.

Recommend Setting

If first rotary axis is

1. A axis, then (a,b,c) = (0, Y component from tool holder to A axis, Z component from tool holder to A axis)
2. B axis, then (a,b,c) = (X component from tool holder to B axis, 0, Z component from tool holder to B axis)

Figure 23

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3044		1st X-component of offset from machine to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4044	N1~N4					
V10	Pr3045		1st Y-component of offset from machine to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4045	N1~N4					
V10	Pr3046		1st Z-component of offset from machine to second rotation axis	[-999999999,999999999]	BLU	0	Reset
V12	P4046	N1~N4					

Description:

Pr3034 ~ Pr3036 are used to set up the mechanism chain of the 5-axis machine for mix type (see figure 24)

If the offset vector from machine origin to second rotary axis is (d, e, f), then
d is Pr3034, e is Pr3035, f is Pr3036.

Recommend Setting

If second rotary axis is

1. A axis, then (d,e,f) = (0, Y component of the machine coordinate, Z component of the machine coordinate)
2. B axis, then (d,e,f) = (X component of the machine coordinate, 0, Z component of the machine coordinate)
3. C axis, then (d,e,f) = (X component of the machine coordinate, Y component of the machine coordinate, 0)

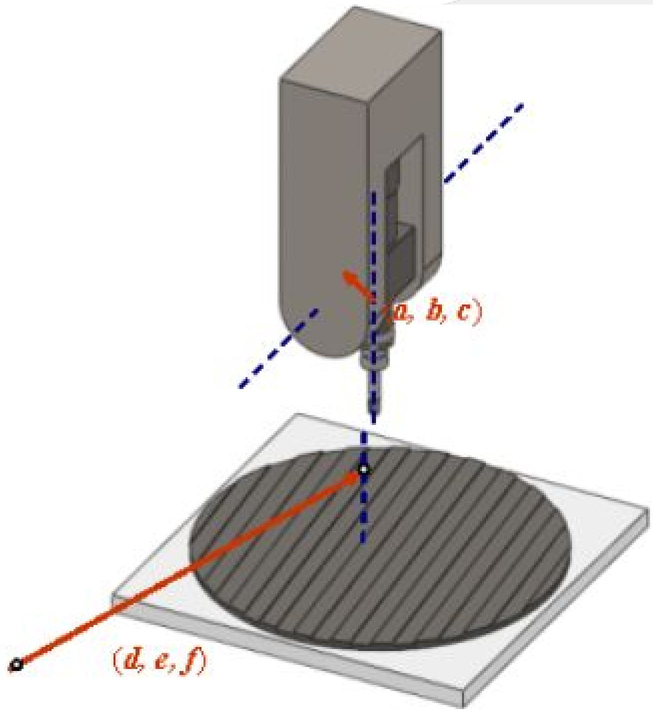


Figure 24

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3051		Enable smooth RTCP function (0:No;1:Yes)	[0,1]	-	0	Reset
V12	P20202	N1~N4					

Description:

0: Disable STCP (Smooth RTCP) Function

1: Enable STCP (Smooth RTCP) Function

When this parameter is set as 1, user can neglect L argument in NC program and STCP function will be enabled automatically.

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3052		First rotation axis smoothness tolerance (0.001deg)	[1, 179999]	0.001 deg	500	Reset
V12	P20203	N1~N4					
V10	Pr3053		Second rotation axis smoothness tolerance (0.001deg)	[1, 179999]	0.001 deg	500	Reset
V12	P20204	N1~N4					

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3054		1st RTCP interpolation mode (0: Five axis simultaneously; 1: Tool vector)	[0,1]	-	0	Reset
V12	P20201	N1~N4					

Description:

0: Five Axis Simultaneous
1: Tool Vector

Five Axes Simultaneous:

If the tool moves from vector V_1 to vector V_2, the tool direction and posture will not be considered, all axis will move simultaneously with the normal interpolation mode.

Tool Vector:

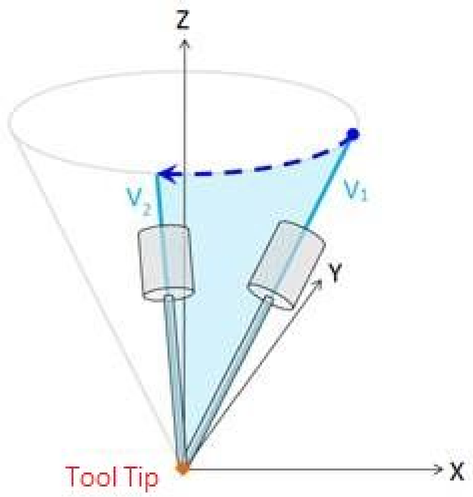
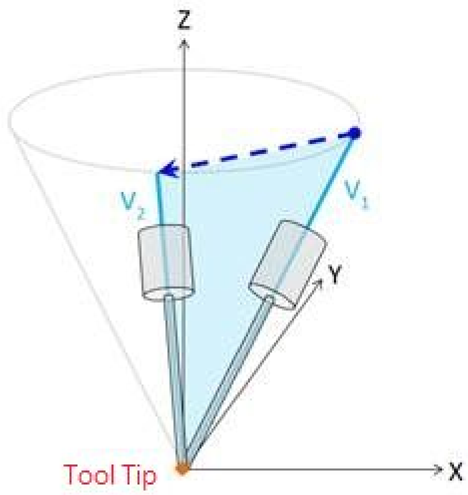
If the tool moves from vector V_1 to vector V_2, considering the tool posture during movement, the tool vector must stay on the plane composed of V_1 and V_2.

Note

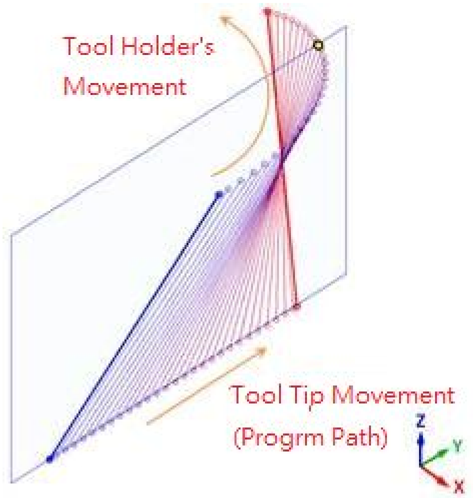
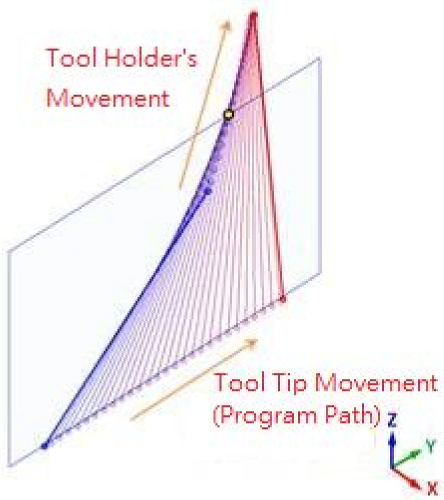
1. Tool vector interpolation mode is only valid for cutting G codes, such as G01, G02, G03, G02.4, G03.4. Axial movement generated from other G codes, such as G00, will not be affected by this parameter.

2. Under Pr3054 = 1, NC blocks will be interpreted as tool vectors.
   Therefore, the axes might not arrive the designated positions after executing a block, and their behavior might not follow the rules defined by axial types(Pr221~) as well.
   ex. After executing the NC block: " A10. C0. ", the axes might stop at " A-10. C180. " since these two sets of rotary position represent the same tool vector.
3. Under Pr3054 = 1, If the angle between the start and end tool vectors is close to 180 degrees, an alarm will be triggered: 【COR-183 Angle between start and end tool vector approach 180 during tool vector interpolation mode】.

> Example 1
> G43.4 H1 [Enable RTCP]
> G90 G01 X0. Y0. Z0. B30. C0. [Initial tool posture]
> G01 C-90. F1000 [Execute motion command]

Five Axis Simultaneous **Tool posture changes during movement**	**Tool Vector** **Tool posture stays on the same plane**
	
Because there is no command for B axis, so B axis stays still; the tool posture is slightly off from the plane composed of V_1 and V_2.	To stay on the plane composed of V_1 and V_2 during movement, B axis rotates without command and eventually returns to its original angle.

> Example 2
> G90 G00 B30. C45. [Initial tool posture]
> G43.4 H1 [Enable RTCP]
> X0. Y0. Z0. [Positioning]
> G01 Y100. B60. C135. F1000 [Execute motion command]

Five Axis Simultaneous Tool posture changes during movement	Tool Vector Tool posture stays on the same plane
	
Because tool posture is not considered, B and C axis will rotate simultaneously. Therefore tool posture deflects.	Tool posture stays on the plane composed of V_1 and V_2 during movement. B and C axis will reach target coordinate at the same time even they don't rotate simultaneously.

1.4.1 Related Alarm

Alarm ID	**COR-183**	**Alarm Title**	**【Angle between start and end tool vector approach 180 during tool vector interpolation mode】**
Description	When the RTCP interpolation mode is set to tool vector, the angle between the starting and ending tool vectors approaches 180.		
Reason	When the RTCP interpolation mode is set to tool vector and the tool vector at the end point of a segment is parallel to the current tool vector, it is not possible to determine a single movement plane.		
Solution	1. Please modify the NC file to ensure that the angle between tool vector at the end point of a segment and the current tool vector is not close to 180 degrees. 2. Set Pr3054 to 0.		

<table>
<tr><th>Version</th><th colspan="2">No</th><th>Descriptions</th><th>Range</th><th>Unit</th><th>Default</th><th>Effective</th></tr>
<tr><td>V10</td><td colspan="2">Pr3055</td><td rowspan="2">Reference datum of rotation axis offset in coordinate transformation</td><td rowspan="2">[0,1]</td><td rowspan="2">-</td><td rowspan="2">0</td><td rowspan="2">Reset</td></tr>
<tr><td>V12</td><td>P20 205</td><td>N1~N4</td></tr>
</table>

Value

0—zero position of machine coordinate

1—zero position of workpiece coordinate

Description

Reference datum of rotation axis offset in coordinate transformation is exclusive to RTCP and only exist during RTCP is enabled.

Parameter	**Physical meaning**
Pr3055=1	Reference datum of rotation axis offset in coordinate transformation is zero position of workpiece coordinate
Pr3055=0	Reference datum of rotation axis offset in coordinate transformation is zero position of machine coordinate

- Difference in application:

If Pr3055 is 1, user should fill the specific angle into G54 offset before enabling RTCP.

Examples are made to explain the specification in detail.

Notice

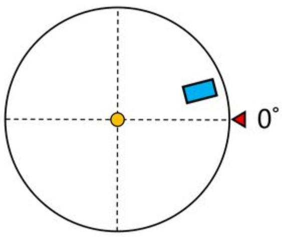
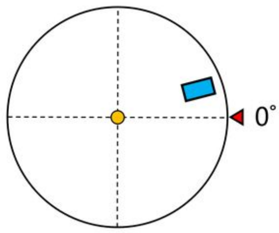
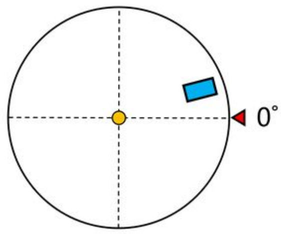
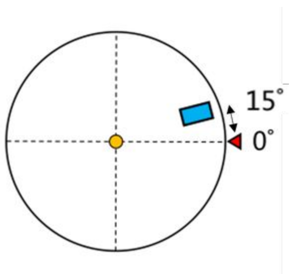
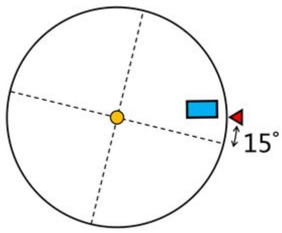
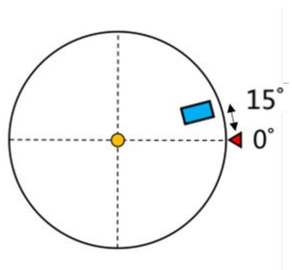
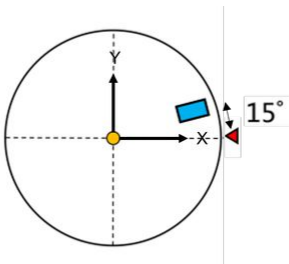
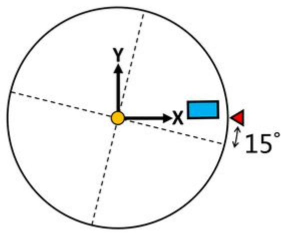
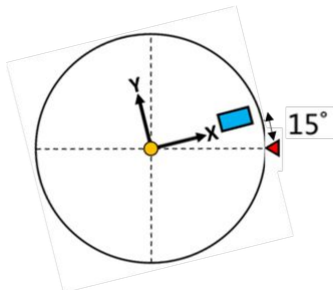
- If the parameter is modified when RTCP is enabled, it will take effect when RTCP being enabled next time.

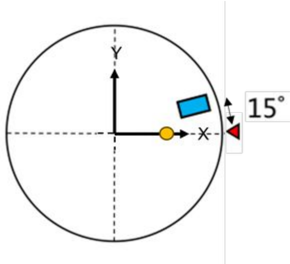
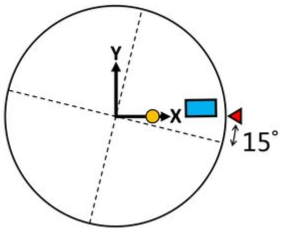
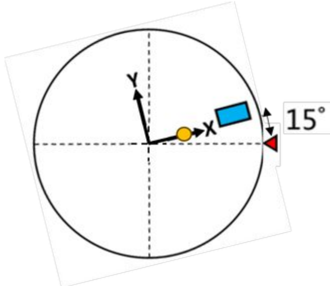
Example

Legend

- The table rotation should follow left-hand rule.
- Yellow circle: position of tool tip
- Triangle: zero position of C axis in machine coordinate
- Square: Workpiece

Example	**A**	**B**	**C**
Pr3055	**0**	**1**	**1**
G54 Offset	C+15.	C+15.	C+15.

NC Command	(directly enable RTCP) G90 G49 G54 N1 // do nothing N2 G43.4 H1 Z0. // RTCP ON N3 X10. Y0. Z0. // move tooltip	(locate C axis, then enable RTCP) G90 G49 G54 N1 C0. // C positioning N2 G43.4 H1 Z0. // RTCP ON N3 X10. Y0. Z0. // tool tip moves	(directly enable RTCP) G90 G49 G54 N1 // do nothing N2 G43.4 H1 Z0. // RTCP ON N3 X10. Y0. Z0. // tool tip moves
Initial position			
N1	// do nothing 	C0. // C positioning 	// do nothing 
N2	G43.4 H1 Z0. // RTCP ON 	G43.4 H1 Z0. // RTCP ON 	G43.4 H1 Z0. // RTCP ON 

N3	X10. Y0. Z0. // tool tip moves 	X10. Y0. Z0. // tool tip moves 	X10. Y0. Z0. // tool tip moves 

Before version 10.118.82

Parameter	**Physical meaning**
Table Type Pr3055=0	Reference datum of rotation axis offset in coordinate transformation is RTCP start position
Table Type Pr3055=1	Reference datum of rotation axis offset in coordinate transformation is zero position of workpiece coordinate
Spindle Type = 0/1	Reference datum of rotation axis offset in coordinate transformation is zero position of machine coordinate

2 2. Rotation Tool Center Point (RTCP)

RTCP function will be introduced in this chapter.

2.1 2.1 Introduction of RTCP Function

RTCP (Rotation Tool Center Point) is the function to control the tool center point.

When RTCP function is enabled, the control point will change from the tool holder to the tool center, the object of all commands is the coordinate of tool center, it's exclusive for 5-axis machines.

Before RTCP function, CAM software is required to generate NC program based on current tool length, one NC program for one tool length.

If the tool length changes, a new NC program is required, and be criticized with its inefficiency.

With RTCP function, CAM software only needs to calculate the coordinate of workpiece contour, the tool length and tool wear are considered by the controller automatically.

The tool center point will always work along the workpiece contour, no matter how the tool length or tool wear varies.

There are two paths in figure 25, the orange one shows the control point is the tool holder without RTCP enabled; the red one shows the control point is the tool center point with RTCP enabled.

It's also shown in figure 25 that the tool posture changes continuously, when the tool length or tool wear is updated, CAM software is required to generate new path without RTCP enabled.

If RTCP function is enabled, we can update the tool length and wear in the table directly, then the controller will complete the compensation automatically.

Therefore, it's suggested to apply RTCP function when machining with 5-axis machines, which increases the precision and efficiency and also makes the greatest use of the machine.

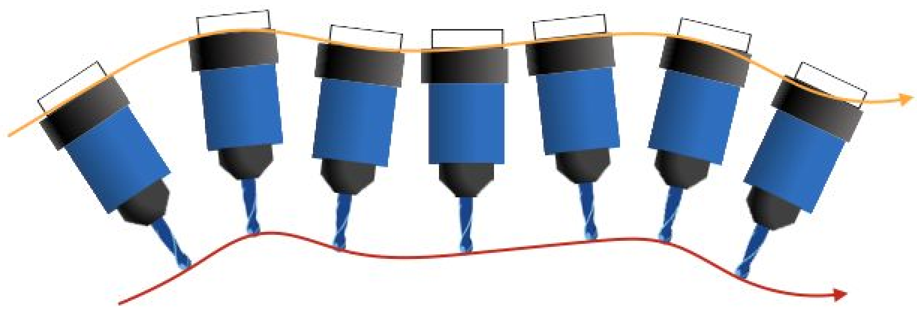


Fig.25

For now, Syntec controller provides two types of command format to enable RTCP, Type1 and Type2.

The difference is the way to define the tool posture.

Type 1 determines the tool posture with angle of 1st & 2nd rotary axis; Type 2 determines the tool posture with tool vector.

More details are introduced in the following chapters.

2.2 2.2 RTCP Type1

2.2.1 Command Format

```
G43.4 H_;
G49;

G43.4: enable RTCP Type1;
G49: disable RTCP Type1;
H : tool compensation number;
```

2.2.2 Limitations

1. Do not apply with G41, G42 tool radius compensation function
2. Do not apply with G43, G44, G43.5 tool length compensation function
3. The tool length should be positive
4. Disable RTCP mode with G49 before applying G53, G28, G29 or G30 to avoid abnormal motions
5. Enable HPCC function with G05 P10000 in RTCP mode will trigger alarm 【COR-140 G05 HPCC cannot apply under RTCP mode】

2.2.3 Example

Two program examples shown below explain the difference of machine motion between RTCP disabled and enabled.

For first program, G43.4 command is not given; but for second program, RTCP is enabled in the very beginning line.

Program with RTCP disabled:

```
G00 X0. Y0. Z0. B-45. C0.
G01 X50. Y0. Z0. B45. C0.
```

Fig.26 shows the machine motion with RTCP disabled.

Program with RTCP enabled:

```
G43.4 H1
G00 X0 Y0 Z0 B-45 C0
G01 X50. Y0 Z0 B45. C0
```

Fig.27 shows the machine motion with RTCP enabled.

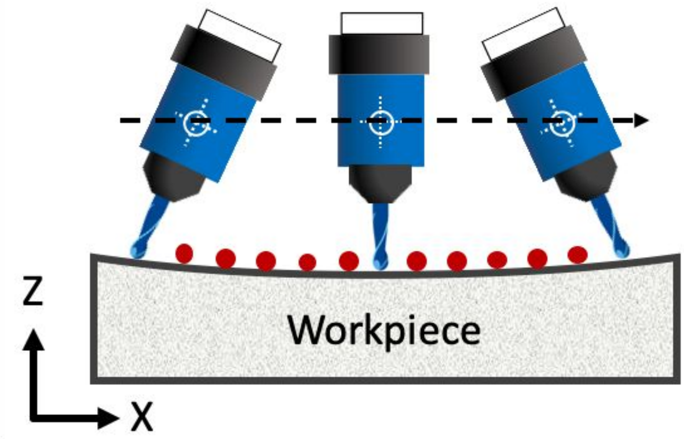


Fig.26

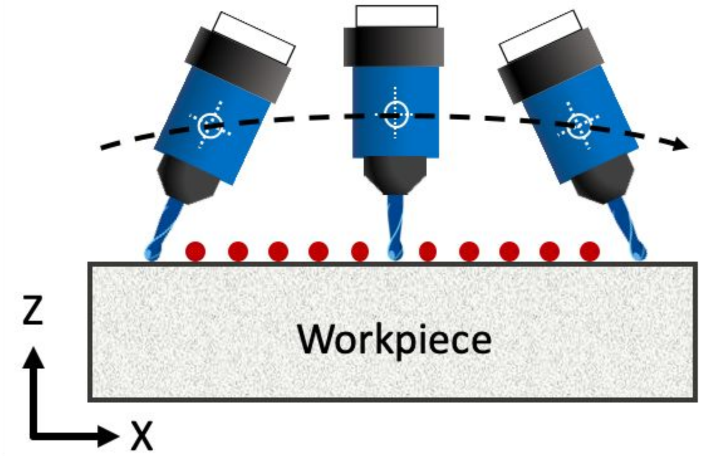


Fig.27

2.3 2.3 RTCP Type2

2.3.1 Command Format

```
G43.5 H_;
X_ Y_ Z_ I_ J_ K_;
G49;
G43.5: enable RTCP Type2;
G49: disable RTCP Type2;
H : tool compensation number;
X_ Y_ Z_: coordinate of moving block for tool center point in program coordinate
system
I_ J_ K_: tool vector of moving block at end point in program coordinate system
(refer to Fig.28 for tool vector definition)
```

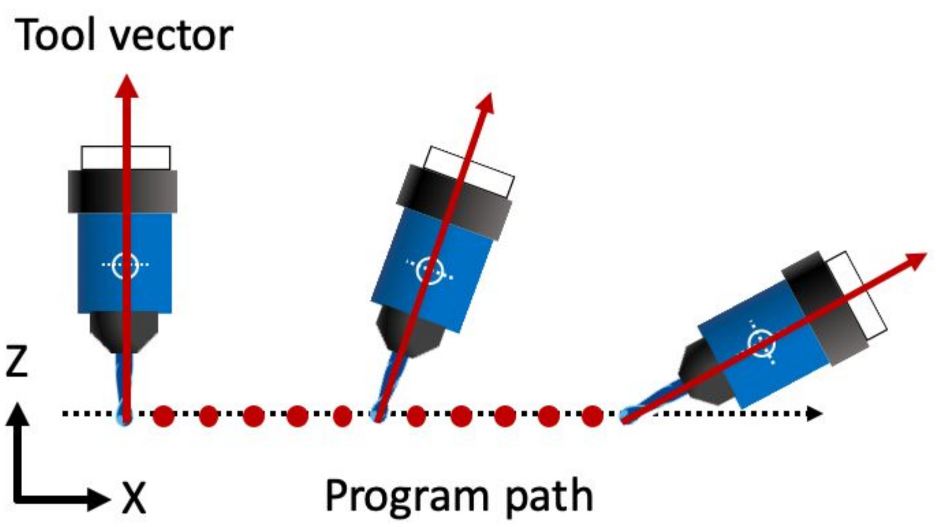


Fig.28

2.3.2 Limitations

1. Do not apply with G41, G42 tool radius compensation function
2. Do not apply with G43, G44, G43.4 tool length compensation function
3. Do not apply with G91 incremental command
4. The tool length should be positive
5. Disable RTCP mode with G49 before applying G53, G28, G29 or G30 to avoid abnormal motions
6. Enable HPCC function with G05 P10000 in RTCP mode will trigger alarm 【COR-140 G05 HPCC cannot apply under RTCP mode】
7. Execute 1st/2nd rotary axis rotating commands in RTCP Type2 mode will trigger alarm 【COR-158 Master and slave rotation angle command is inhibit in G43.5 mode】
8. The arguments will be regarded as 0 when one of the arguments I, J, K is omitted; if I, J, K are all omitted then the tool posture will be the same as previous block
9. The tool vector shall not be a 0 vector, if I, J, K are all 0. then alarm 【COR-159 The tool vector is invalid】 will be triggered
10. STCP function (Smooth Tool Center Point) is not supported

2.3.3 Example

Two program examples shown below explain the difference of machine motion between RTCP disabled and enabled.

For first program, G43.5 command is not given; but for second program, RTCP is enabled in the very beginning line.

Program with RTCP disabled:

```
G00 X0 Y0 Z0 B-45. C0.
G01 X50. Y0 Z0 B45. C0.
```

Fig.29 shows the machine motion with RTCP disabled.

Program with RTCP enabled:

```
G43.5 H1
G00 X0. Y0. Z0. I-1. J0. K1.
G01 X50. Y0. Z0. I1. J0. K1.
```

Fig.30 shows the machine motion with RTCP enabled.

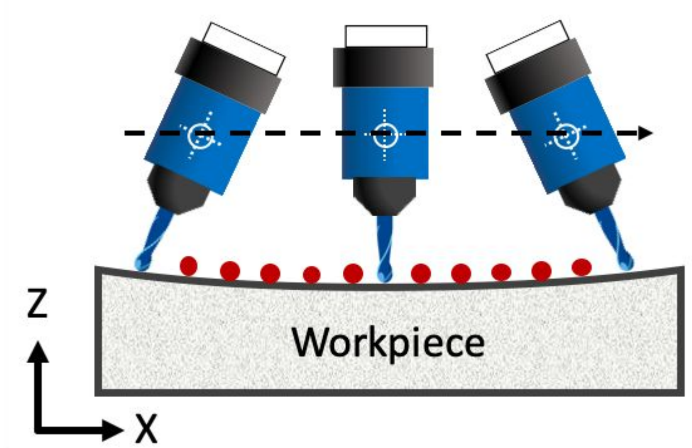


Fig.29

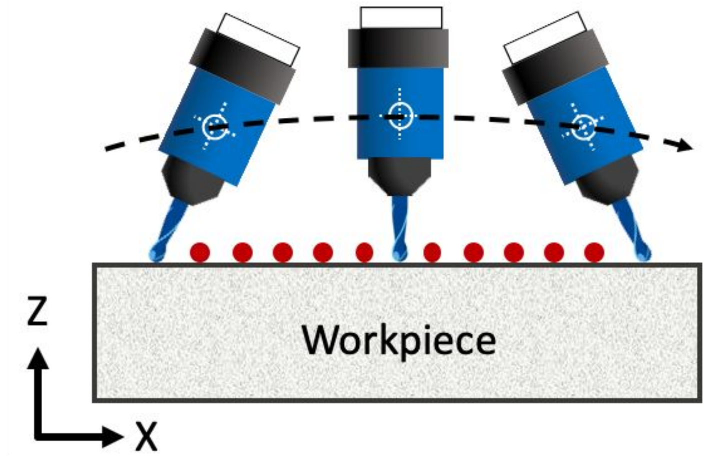
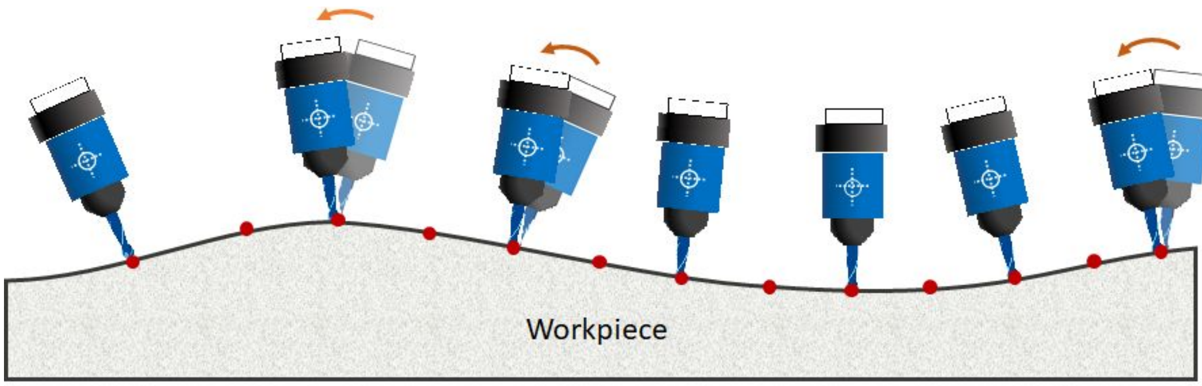


Fig.30

2.4 2.4 Smooth Tool Orientation Function (STO)

STO (Smooth Tool Orientation) is a function for smoothing tool orientation control. When STO is enabled, the rotary axis commands in the machining program are processed smoothly, resulting in a more stable machining process and improved surface quality.



STO (On) G43.4 L2

2.4.1 Command Format

G43.4 H_: For detailed usage, refer to the relevant documentation(2.2 G43.4 RTCP Type1 (ENG)). This document explains the functions related to G43.4 L2.

- **G43.4 L2 E_ R_**; // Enable L2 function
- **G49**; // Disable **G43.4** and **L2** functions
- **G43.4 L0**; // Disable L2 function

- **E**: Allowable error for the first and second rotary axes. Controls the deviation between smoothed and unsmoothed rotary axis commands. Unit: IU (deg). Input range: **0.001–179.999**.
  - If **E** is not specified, the allowable errors for the first and second rotary axes default to **Pr3052** and **Pr3053**, respectively.
- **R**: Ball-end tool tip radius. Unit: IU (mm or inch).
  - Required only when machining files use **tool tip machining** with a **ball-end tool**.
  - Not needed for **ball center machining** or **non-ball-end tools**.

2.4.2 Description

Supported Versions:10.118.48N, 10.118.52H, 10.118.56B, 10.118.57 and later.

This function smooths **G01 rotary axis commands** in the machining file **without affecting the tool tip position**. The smoothing is based on either the user-defined **allowable error parameter (E)** or the default **allowable errors (Pr3052, Pr3053)**.

Related Parameters:

- **Pr3052, Pr3053**: Allowable error for the first and second rotary axes.

2.4.3 Notes

1. This function requires **RTCP** or **4-axis RTCP** option to be enabled.
2. t only applies to the **first axis group**; other axis groups will not be smoothed.
3. **Parameter Pr3809 must not be set to 1** (UVW treated as XYZ increments), otherwise, smoothing will not take effect.
4. Only **short G01 segments (< 4mm)** are smoothed.
5. Allowable Error Settings
   a. **G43.4 L2:**

    - If **E is specified**, the allowable errors for the first and second rotary axes are set to **E**.
    - If **E is not specified**, the allowable errors default to **Pr3052, Pr3053**.
  b. After executing **G43.4 L2 E_**, the error value remains until **G49, G43.4 L0, or RESET** is executed.
  c. **E value range: 0.001 – 179.999 (deg)**. Exceeding this range triggers **COR-170 G43.4 L2 illegal argument value**
  d. If **G43.4 L2 is not closed** before the next execution, the **previous E value is retained** (See Example 4).

6. Ball Radius (R) Settings
    - **R is always treated as an absolute value**, regardless of **G90/G91 mode**.
    - **Negative R values will trigger an alarm** (**COR-170 G43.4 L2 Argument Error**).
    - If using a ball-end tool with a **tool-tip path from CAD/CAM**, the **R value must be set**.
    - After **G43.4 L0, G49, or RESET**, the **R value resets to 0** (See Example 3).
    - If **G43.4 L2 is not closed**, the **previous R value is retained** in the next execution (See Example 4).
7. **Not affected by Pr415**: On **RESET**, L2 functionality ends, and **E reverts to Pr3052/3053** while **R resets to 0**.
8. **Not compatible with**: Workpiece coordinate setting, Coordinate transformation, Kinematic chain transformation commands
    a. G10 L2/L10/L11/L21/L1050/L1051/L5000/L1300/L1301
    b. G12.1
    c. G17/18/19
    d. G52/54-59/G59.1-G59.9
    e. G68/G68.2/G68.3/G69/G92/G92.1

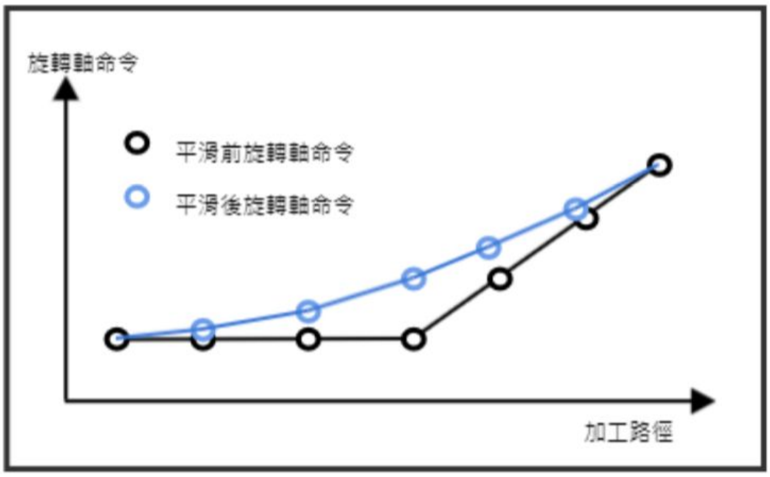
2.4.4 Related Parameters

Parameter	Description	Range	Unit	Default	Effective After Modification
3052	Allowable smoothing angle for the 1st rotary axis	[1,179999]	0.001deg	500	Effective after Reset
3053	Allowable smoothing angle for the 2nd rotary axis	[1,179999]	0.001deg	500	Effective after Reset

2.4.5 Smoothing Effect

1. This function smooths the rotary axis path without altering the geometric position commands.

2. The effect is illustrated in the following figure:



2.4.6 Program Example

Program Example1

```
N001 G0 X0. Y0. Z0. A6. C360.
N002 G43.4 L2 // Enable rotary axis smoothing, with allowable error values for the
first and second rotary axes set to Pr3052 and Pr3053, without setting the tool ball
radius (no R argument).
N003 X-132.35 A6.001 C359.266
N004 X-132.45 A6.001 C359.119
N005 X-132.55 A6.001 C358.973
N006 X-132.65 A6.001 C358.826
N007 X-132.75 A6.002 C358.679
N008 X-132.85 A6.002 C358.533
N009 X-132.95 A6.002 C358.386
N010 X-133.05 A6.003 C358.239
N011 G49 // Disable rotary axis smoothing
N012 M30
```

Program Example2

```
N001 G0 X0. Y0. Z0. A6. C360.
N002 G43.4 L2 E0.2 R1 // Enable rotary axis smoothing, set the allowable error for
the first and second rotary axes to 0.2deg, and set the ball radius to 1 (mm or
inch).
N003 X-132.35 A6.001 C359.266
N004 X-132.45 A6.001 C359.119
N005 X-132.55 A6.001 C358.973
N006 X-132.65 A6.001 C358.826
N007 X-132.75 A6.002 C358.679
N008 X-132.85 A6.002 C358.533
N009 X-132.95 A6.002 C358.386
```

```
N010 X-133.05 A6.003 C358.239
N011 G49 // Disable rotary axis smoothing
N012 M30
```

Program Example3

```
N001 G0 X0. Y0. Z0. A6. C360.
N002 G43.4 L2 E2 R3 // Enable rotary axis smoothing, set the allowable error for the
first and second rotary axes to 2deg, and set the ball radius to 3 (mm or inch).
N003 X-132.35 A6.001 C359.266
N004 X-132.45 A6.001 C359.119
N005 X-132.55 A6.001 C358.973
N006 X-132.65 A6.001 C358.826
N007 X-132.75 A6.002 C358.679
N008 X-132.85 A6.002 C358.533
N009 X-132.95 A6.002 C358.386
N010 X-133.05 A6.003 C358.239
N011 G43.4 L0 // Disable rotary axis smoothing
N012 G43.4 L2 // Enable rotary axis smoothing, with allowable error values for the
first and second rotary axes set to Pr3052 and Pr3053, without setting the tool ball
radius (no R argument).
N013 X-132.35 A6.001 C359.266
N014 X-132.45 A6.001 C359.119
N015 X-132.55 A6.001 C358.973
N016 X-132.65 A6.001 C358.826
N017 X-132.75 A6.002 C358.679
N018 X-132.85 A6.002 C358.533
N019 X-132.95 A6.002 C358.386
N020 X-133.05 A6.003 C358.239
N021 G49 // Disable rotary axis smoothing
N022 M30
```

Program Example4

```
N001 G0 X0. Y0. Z0. A6. C360.
N002 G43.4 L2 E2 R3 // Enable rotary axis smoothing, set the allowable error for the
first and second rotary axes to 2deg, and set the ball radius to 3 (mm or inch).
N003 X-132.35 A6.001 C359.266
N004 X-132.45 A6.001 C359.119
N005 X-132.55 A6.001 C358.973
N006 X-132.65 A6.001 C358.826
N007 X-132.75 A6.002 C358.679
N008 X-132.85 A6.002 C358.533
N009 X-132.95 A6.002 C358.386
N010 X-133.05 A6.003 C358.239
N012 G43.4 L2 // Enable rotary axis smoothing. Since G43.4 L2 has not been turned off
(via G49, G43.4 L0, or RESET), the allowable error values for the first and second
rotary axes remain at 2deg, and the ball radius remains at 3 (mm or inch).
N013 X-132.35 A6.001 C359.266
N014 X-132.45 A6.001 C359.119
```

```
N015 X-132.55 A6.001 C358.973
N016 X-132.65 A6.001 C358.826
N017 X-132.75 A6.002 C358.679
N018 X-132.85 A6.002 C358.533
N019 X-132.95 A6.002 C358.386
N020 X-133.05 A6.003 C358.239
N021 G49 // Disable rotary axis smoothing
N022 M30
```

2.5 2.5 Manual RTCP Functions

RTCP function can not only be enabled with G43.4 command in machining, but also be enabled with PLC function in manual operations (MPG, JOG, IncJOG).

2.5.1 Function Description

Manual RTCP functions are controlled by R518 & R519, the functions are introduced below:

R518

R518 is the register used to select the coordinate when linear axis is moving manually, the value and corresponding coordinates are listed below:

1. R518 = 0, the linear axis is moving manually based on machine coordinate.
2. R518 = 1, the linear axis is moving manually based on program coordinate.
3. R518 = 2, the linear axis is moving manually based on tool coordinate.

R519

R519 is the register only for manual function of 5-axis machines, it's not effective when applied to non-5-axis machines.

When operating with manual function of 5-axis machines, RTCP will be enabled with R519 = 1 and be disabled with R519 = 0.

When RTCP is enabled, the program coordinate of tool center point will remain the same when the rotary axis is rotating manually, but the tool (or the table) posture will change, as shown in Fig.32.

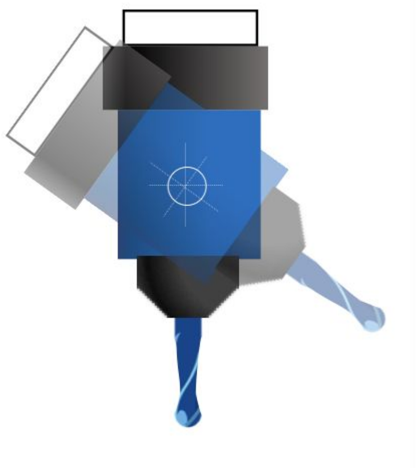
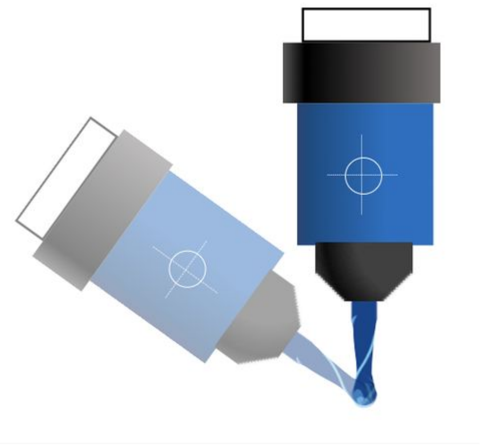
	
R519 = 0	R519 = 1

Fig.32

Tool Coordinate

When both rotary axis are at 0 degree, the definition of the tool orientation is shown as the table below.

Fig.33 shows the tool coordinate when Pr3002=3 and both rotary axis are at 0 degree.

Pr3002	Tool Axis Direction	Tool Axis Direction 1	Tool Axis Direction 2
1	+X	+Y	+Z
2	+Y	+Z	+X
3	+Z	+X	+Y

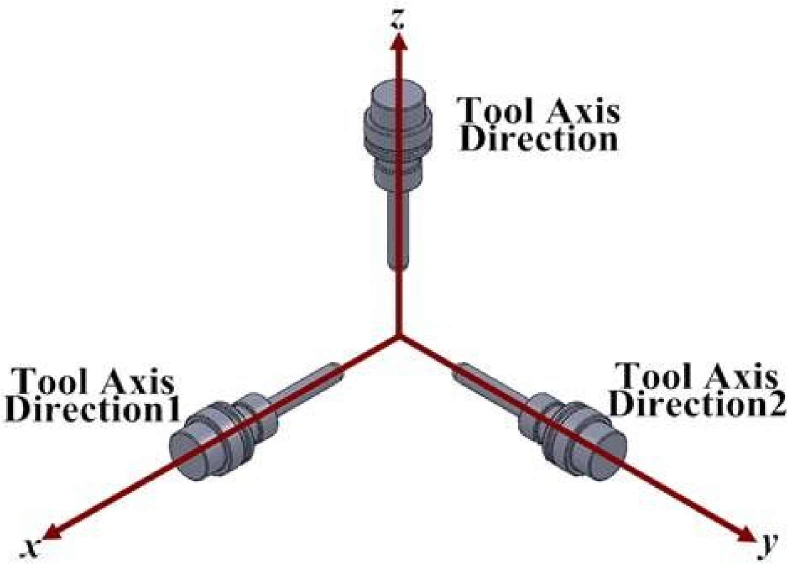


Fig.33

When the rotary axis is not at 0 degree, the tool orientation means the direction pointing from tool tip to tool holder instead of +Z.

Manual function with tool coordinate is only applicable when there's a rotary axis on spindle side, such as spindle type or mix type 5-axis machines.

The tool orientation of table type 5-axis machines is unchangeable thus the tool coordinate won't change.

When the rotary axis are both at 0 degree, the tool coordinate overlaps with the machine coordinate.

The tool coordinate rotates when the rotary axis rotates.

As shown on the left of Fig.34, when the tool rotates along X axis, the new tool coordinate is shown on the right of Fig.34.

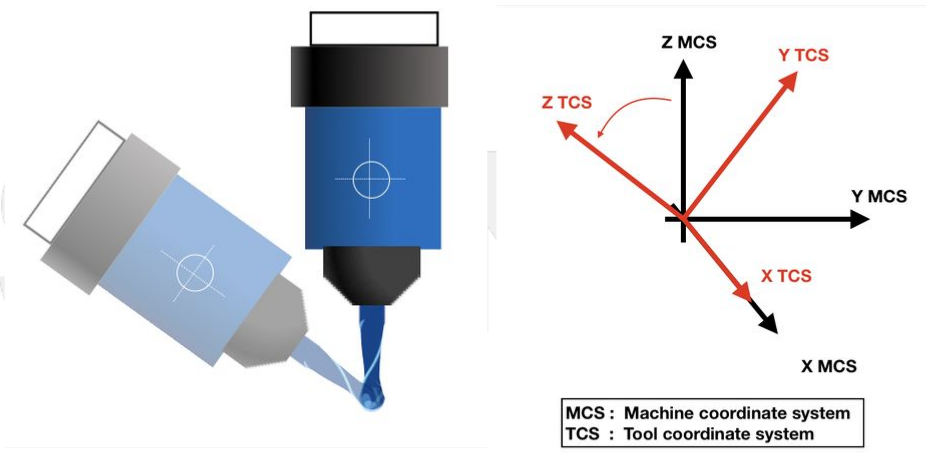


Fig.34

2.5.2 Notifications

1. R518 only affects the linear axis, the motion of rotary axis will be the same with all values of R518.
2. R519 only affects the rotary axis, the motion of linear axis will be the same with all values of R519.
3. Before applying manual RTCP functions, remember to add R518 and R519 in the Ladder to enable the functions.
4. To enable manual RTCP functions, besides R519=1, the coordinate set by R518 should be confirmed, then switch to MDI mode and execute G43.4 command, finally switch to MPG mode.
5. With manual RTCP functions enabled, the machine coordinate of XYZ and all coordinates of rotary axis will change when the rotary axis rotate manually, but the program coordinate of XYZ won't.

2.5.3 Function Test

R518

Spindle Type

With rotary axis on spindle side, set R518 to 2. Since the tool coordinate changes but the program coordinate (workpiece coordinate) don't, it's meaningless to set R518 to 1.

Rotate the rotary axis to an arbitrary angle and change the tool orientation.

Since the tool coordinate is following the rotary axis, so the linear axis will be moving along to the new directions of XYZ.

If the motions are not changed then it means the manual RTCP function is not enabled.

Table Type

With rotary axis on table side, set R518 to 1. Sine the program coordinate (workpiece coordinate) changes but the tool coordinate don't, it's meaningless to set R518 to 2.

Rotate the table to an arbitrary angle and change the orientation of workpiece coordinate.

The linear axis will be moving along to the new workpiece coordinate.

If the motions are not changed then it means the manual RTCP function is not enabled.

Mix Type

With rotary axis on both sides, R518 can be set to 1 or 2, please refer to previous sections for the test method.

Example of Coordinate Setting：

R518 = 0

Switch to X axis and rotate the MPG, the motion of the machine is shown in Fig.35.

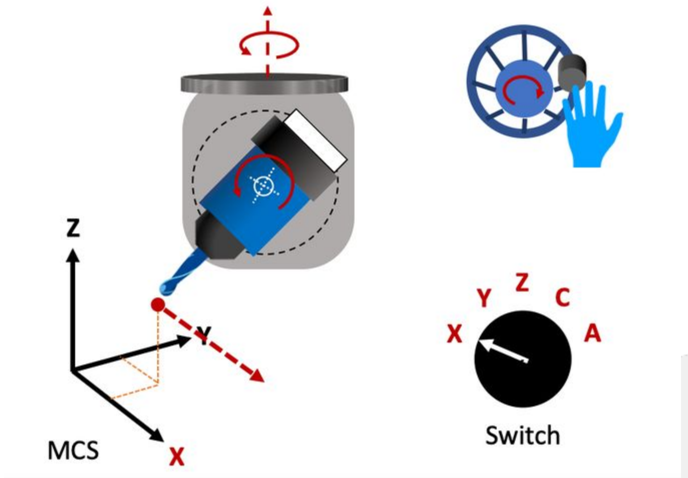


Fig.35

R518 = 1

Switch to Y axis and rotate the MPG, the motion of the machine is shown in Fig.36.

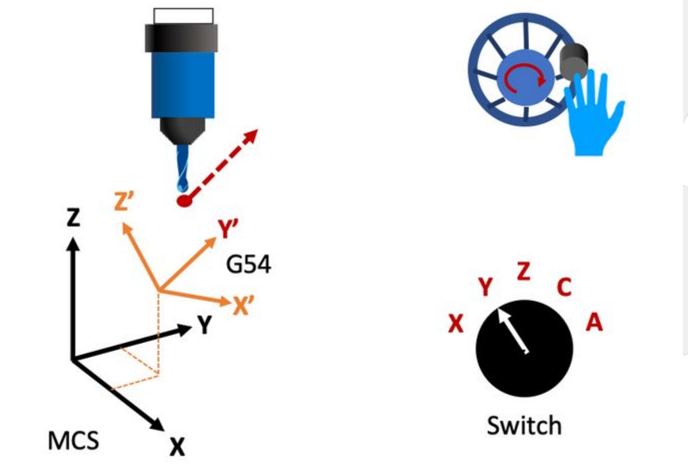


Fig.36

R518 = 2

Switch to Z axis and rotate the MPG, the motion of the machine is shown in Fig.37.

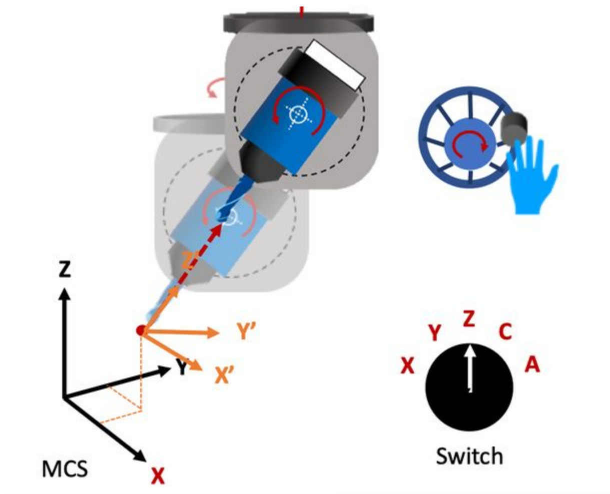


Fig.37

R519

Set R519 to 1 and execute G43.4 to enable RTCP.

Spindle Type

Move the spindle (or tool) to an appropriate position and rotate the rotary axis, if the tool length compensation is executed properly, the tool center point won't move.

If the tool length is not set, then the spindle nose won't move.

The program coordinate of XYZ won't change during the rotation, but those of rotary axis and the machine coordinate of all axis will.

For example, if B axis rotates, the machine coordinate of XZ will change but the program coordinate of XZ won't, and both coordinates of B axis will change.

Table Type

Move the spindle (or tool) to an appropriate position and rotate the rotary axis, if the tool length compensation is executed properly, the relative position of tool center point and the table will remain.

If the tool length is not set, then the relative position of spindle nose and the table will remain.

The program coordinate of XYZ won't change during the rotation, but those of rotary axis and the machine coordinate of all axis will.

For example, if B axis rotates, the machine coordinate of XZ will change but the program coordinate of XZ won't, and both coordinates of B axis will change.

Mix Type

Please refer to the test methods above.

3 3. Tilted Working Plane Machining

The applications, operation specifications and examples of tilted working plane will be introduced in this chapter.

3.1 3.1 G68.2 Tilted Working Plane Machining (Euler Angle)

3.1.1 Function Introduction

Tilted Working Plane (or so-called Feature Coordinate) function can build a program coordinate on arbitrary tilted plane, thus the machining can be executed just like on a horizontal plane.

Tilted Working Plane should be defined with G54 coordinate, in other words, the origin of Tilted Working Plane is set relative to G54 coordinate, and the tilted angle is set by Euler angle.

The relations are shown in Fig.43.

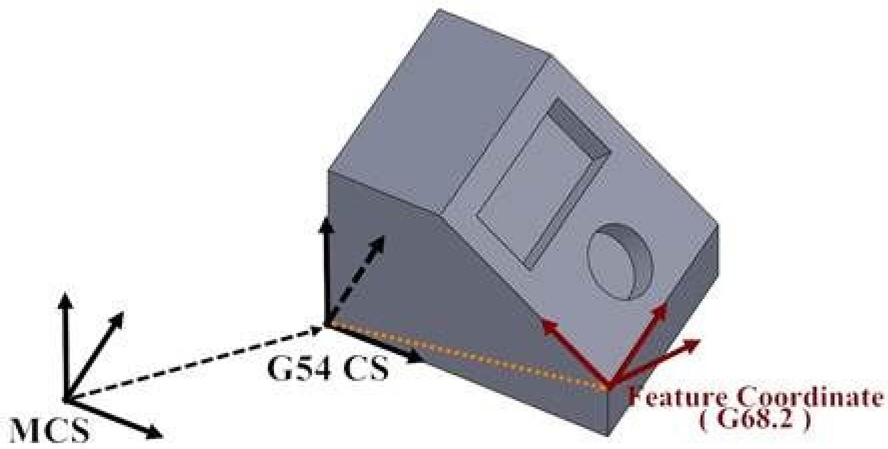


Fig.43

3.1.2 Definition of Euler Angle

Euler angle is used to define Tilted Working Plane with rotation of axis in the order of Z-X-Z.

At first, rotates around Z axis for angle I, then rotates around the new X' axis for angle J, and finally rotates around the new Z' axis for angle K.

The direction of rotation for Euler angle I, J, K is defined by the right-hand rule.

Further details are explained below.

Euler angle I is defined as the rotating angle around Z axis.

As shown in Fig.44, a new coordinate X'Y'Z is created after the coordinate XYZ rotates around Z axis for angle I.

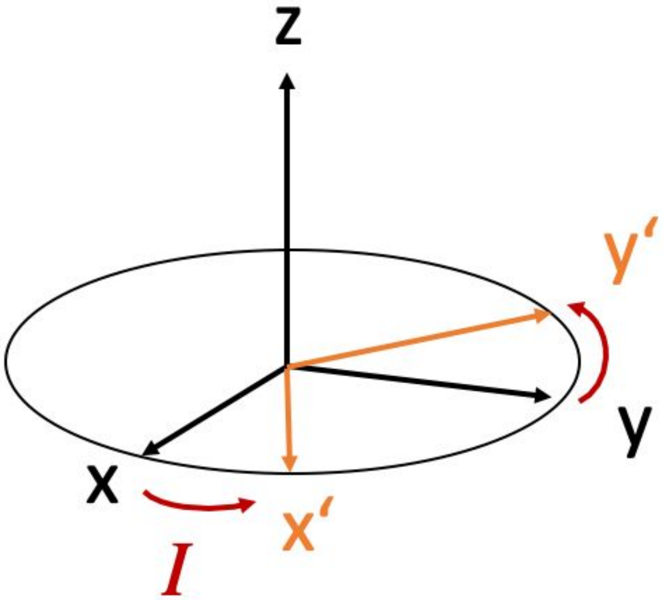


Fig.44

Then based on X'Y'Z coordinate, Euler angle J is defined as the rotating angle around X' axis.

As shown in Fig.45, a new coordinate X'Y''Z'' is created after the coordinate X'Y'Z rotates around X' axis for angle J.

The Z'' here is thus the Zc axis of Tilted Working Plane.

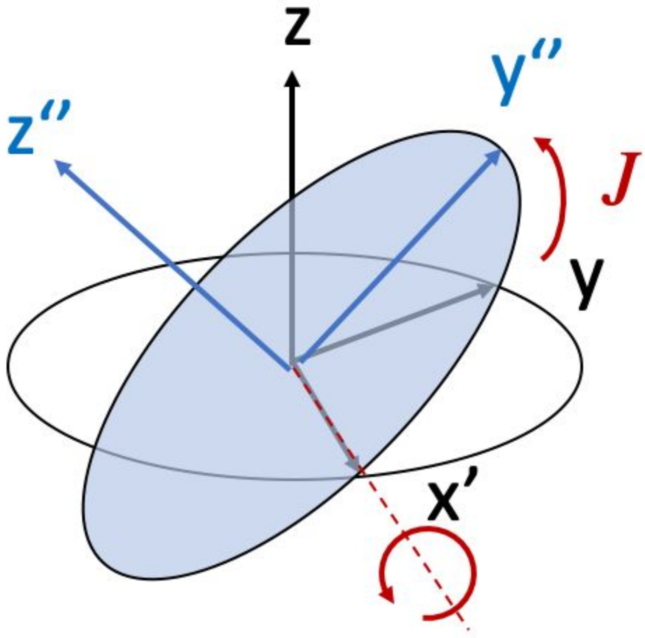


Fig.45

At last, based on X'Y''Zc coordinate, Euler angle K is defined as the rotating angle around Zc axis.

As shown in Fig.46, we obtain XcYcZc of Tilted Working Plane after the coordinate X'Y''Zc rotates around Zc axis for angle K.

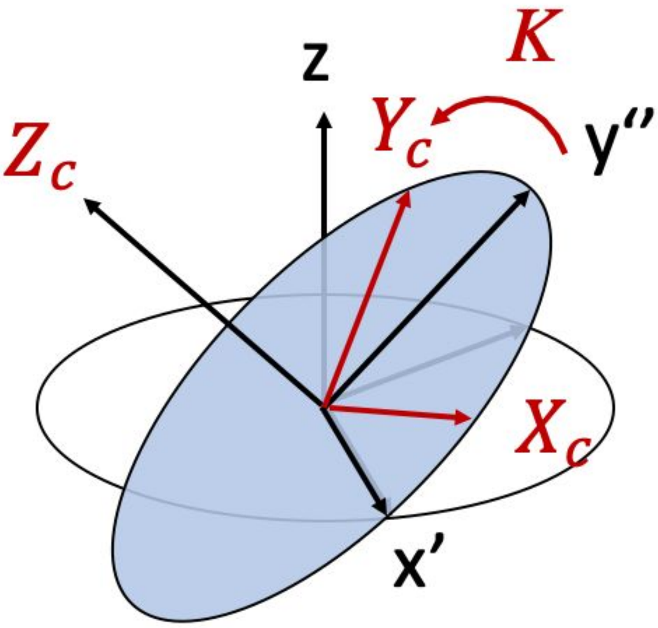


Fig.46

3.1.3 Command Format

With G68.2, the reference coordinate of NC program will be transformed to Tilted Working Plane.

Before G69 is executed, all commands will be seen as the commands for Tilted Working Plane and be executed based on it.

After G68.2 is executed, it's able to control the tool orientation to align to Tilted Working Plane with G53.1 or G53.3 or G53.6 command.

Command format of G68.2 will be explained below:

```
G68.2 X_ Y_ Z_ I_ J_ K_ ; // to set up Tilted Working Plane
G53.1 ; //tool alignment function
G43 H_ ; //tool length compensation, the control point will be changed to the tool
tip.
...
G49 ; //disable tool length compensation
G69 ; //disable Tilted Working Plane function
```

G68.2： enable Tilted Working Plane function;

G69： disable Tilted Working Plane function;

X_ Y_ Z_： the origin of Tilted Working Plane (relative to the origin of G54 coordinate);

I_ J_ K_： Euler angle of Tilted Working Plane;

3.1.4 Application Limitations

1. G68.2 can be executed for multiple times.
2. Each setting is relative to G54 coordinate.
3. Tool length compensation(G43) can't be enabled before G68.2 is executed.

3.1.5 Related Parameters

<table>
<tr><th>Version</th><th colspan="2">No</th><th>Descriptions</th><th>Range</th><th>Unit</th><th>Details</th><th>Effective</th></tr>
<tr><td>V10</td><td colspan="2">Pr3014</td><td rowspan="2">Feature coordinate persist mode</td><td rowspan="2">[0,2]</td><td rowspan="2">-</td><td rowspan="2">0: Do NOT preserve feature coordinate status defined by G68.2/G68.3 after reset & reboot.
1: Preserve feature coordinate status defined by G68.2/G68.3 after reset only.
2: Preserve feature coordinate status defined by G68.2/G68.3 after reset and reboot.</td><td rowspan="2">Reset</td></tr>
<tr><td>V12</td><td>P902</td><td>N1</td></tr>
</table>

3.2 3.2 G53.1 Tool Alignment Function for Tilted Working Plane

3.2.1 **Command Format**

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

G68.2: enable Tilted Working Plane function;
G53.1: tool alignment function;

P: define the rotating direction of the rotary axis, 0: shortest path for 1st rotary axis (Master axis) (default); 1: positive direction for 1st rotary axis; 2: negative direction for 1st rotary axis

After G68.2 is executed and before the cutting commands(EX: G01), G53.1 or G53.3 or G53.6 is required for the tool to align to Tilted Working Plane.

3.2.2 Description

After Tilted Working Plane is enabled, G53.1 is required for the tool to align to Tilted Working Plane, thus this G code is attached under G68.2 and should exist at the same time.

3.2.3 Notifications

1. G53.1 can't be executed before G68.2.
2. Please apply positive tool length. (G43 should be executed after G53.1)
3. After G43 is executed, the control object of the program coordinate is the tool tip. User should apply G49 when cutting is finish to cancel the tool tip control.
4. The P argument will be 0 in default if it's not specified.
5. If the value of P argument is out of range, alarm 【COR-149 G53.1/G53.6 P Argument over range】 will occur.

6. When P is 0, the system will search for the shortest path for 1st rotary axis (Master axis) first. If the target angle or the path is out of range (defined by Pr3009~), the other target angle will be applied instead; if both target angles or paths are out of range, alarm 【COR-153 no solution for this tool direction】 will occur.
7. When P is 1 or 2, if the target angle or the path is out of range (defined by Pr3009~), alarm 【COR-153 no solution for this tool direction】 will occur.
8. For the definitions of the rotary axis corresponding to different mechanisms, please refers to 1.3 Definitions of Rotary Axis and 1.4 Parameter Descriptions.

	0 (default)	**1**	**2**
Spindle/Table/ Mix	shortest path for 1st rotary axis (Master axis)	positive direction for 1st rotary axis	negative direction for 1st rotary axis

Command format of G53.1 is explained below:

```
G68.2 X_ Y_ Z_ I_ J_ K_;
G53.1;
G43 H_;
...
G49;
G69;
```

3.2.4 Program Example

Take the NC program below as example to explain the basic actions of Tilted Working Plane.

```
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 actions of the NC program will be explain line by line:

```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of G54 coordinate by G01 in speed of 1000 mm/min.
```

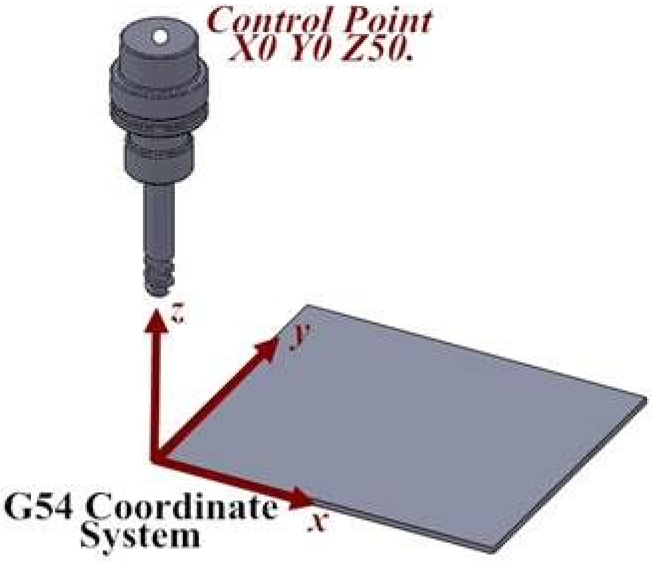


Fig.47

```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
// Specify X100. Y100. Z50. relative to the origin of G54 coordinate as the origin of
Tilted Working Plane, and the Euler angles are I30. J15. K20.
// The program coordinate will transform to Tilted Working Plane after G68.2 is
executed.
```

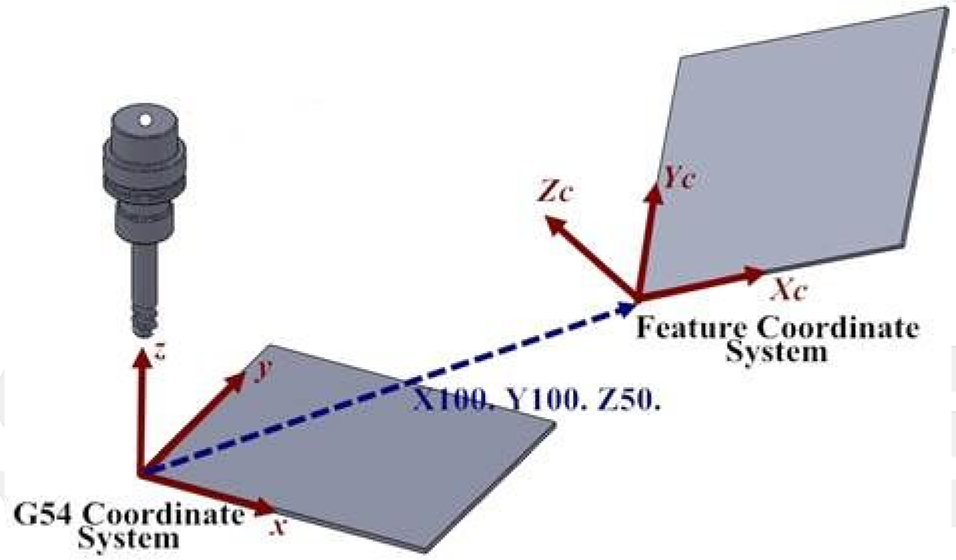


Fig.48

```
N3 G01 X0 Y0 Z50. F1000;
```

```
// Moves to Z50. of Tilted Working Plane by G01 in speed of 1000 mm/min, but the tool direction remains the same.
```

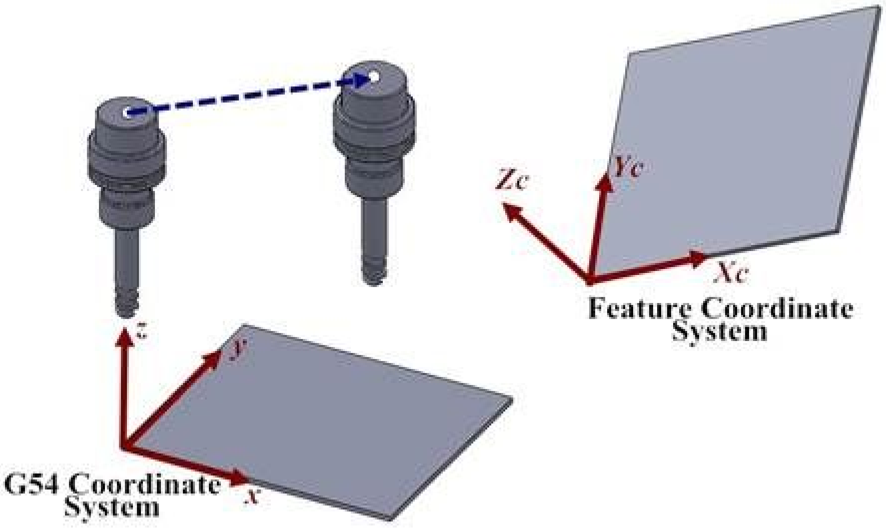


Fig.49

```
N4 G53.1;
// The tool direction aligns to the Z axis of Tilted Working Plane.
```

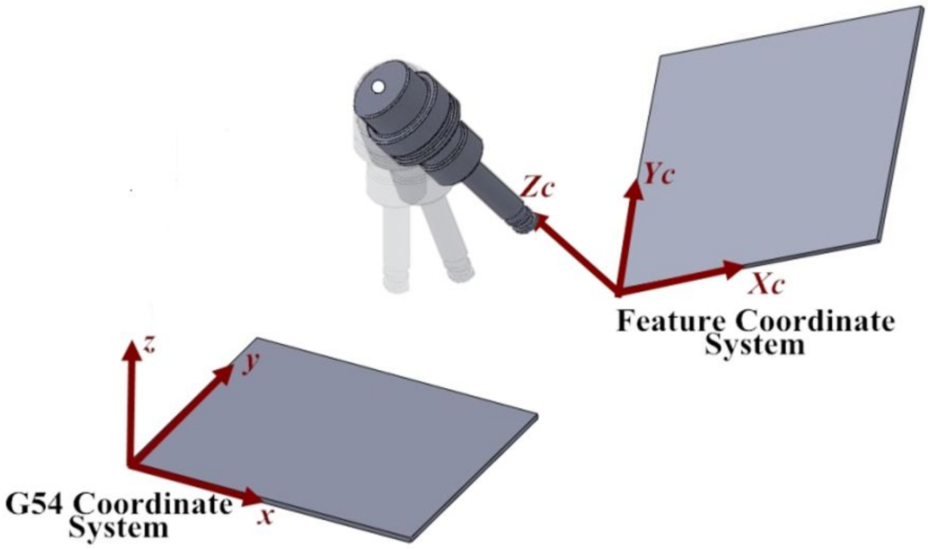


Fig.50

```
N5 G43 H1;
// Tool length compensation, the control point changes to the tool tip.

N6 G01 X0 Y0 Z0;
```

```
// The tool tip moves to X0 Y0 Z0 of Tilted Working Plane.
```

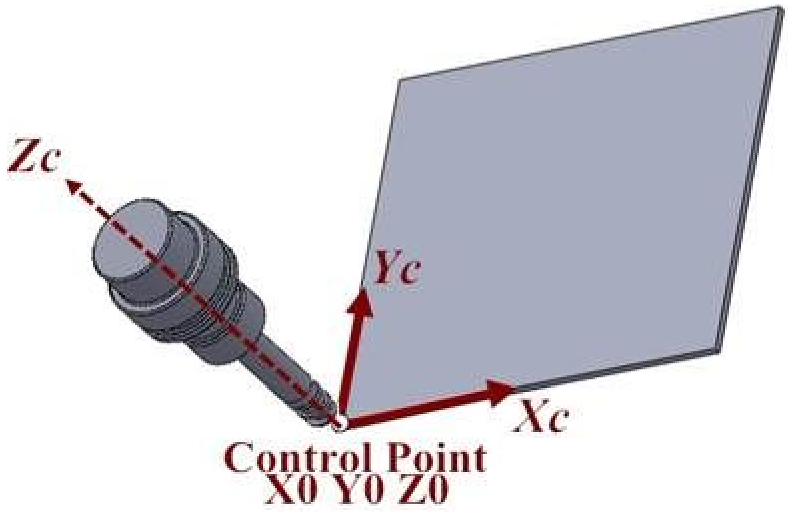


Fig.51

```
N98 G49;
// Cancel tool tip control.
N99 G69;
// Cancel Tilted Working Plane.
N100 G01 X0. Y0. Z50.;
// Moves to X0. Y0. Z50. of G54 coordinate.
```

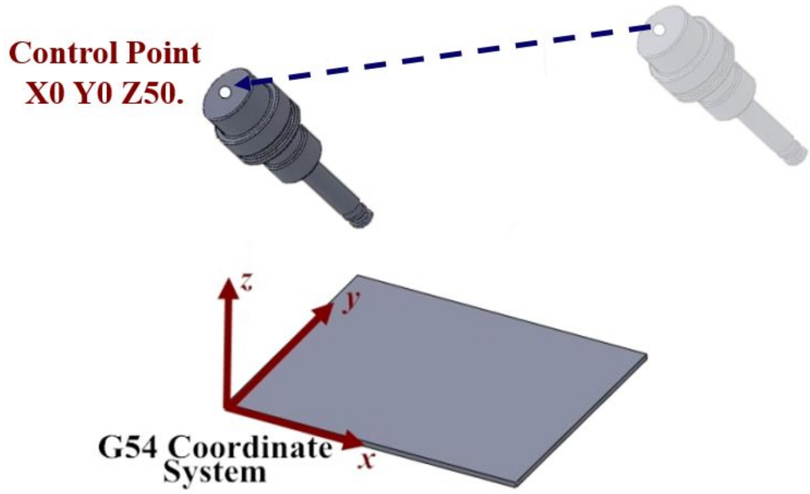


Fig.52

3.3 3.3 G53.3 Tool Alignment Function for Tilted Working Plane (5-Axis simultaneous motion)

3.3.1 **Command Format**

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

G68.2： enable Tilted Working Plane function;
G53.3： tool alignment and positioning function;

X、Y、Z: specified position.

H: Tool number;

P: define the rotating direction of the rotary axis

- 0: shortest path for 1st rotary axis (Master axis) (default);
- 1: positive direction for 1st rotary axis;
- 2: negative direction for 1st rotary axis

After G68.2 is executed and before the cutting commands(EX: G01), G53.1 or G53.3 or G53.6 is required for the tool to align to Tilted Working Plane.

3.3.2 Description

Applying G53.3 after Tilted Working Plane is enabled will lead to the following actions simultaneously:

1. Activate tool length compensation with positive tool length. The number of the tool length is the same as the H argument of G53.3.
2. The tool aligns to Tilted Working Plane.
3. Moves to the specified position of Tilted Working Plane which is specified by XYZ arguments in the speed of G00.

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

3.3.3 Notifications

1. G53.3 can't be executed before G68.2.
2. After G53.3 is executed, the control object of the program coordinate is the tool tip. User should apply G49 when cutting is finish to cancel the tool tip control.
3. The P argument will be 0 in default if it's not specified.
4. If the value of P argument is out of range, alarm 【COR-149 G53.1/G53.6 P Argument over range】 will occur.
5. When P is 0, the system will search for the shortest path for 1st rotary axis (Master axis) first. If the target angle or the path is out of range (defined by Pr3009~), the other target angle will be applied instead; if both target angles or paths are out of range, alarm 【COR-153 no solution for this tool direction】 will occur.
6. When P is 1 or 2, if the target angle or the path is out of range (defined by Pr3009~), alarm 【COR-153 no solution for this tool direction】 will occur.
7. For the definitions of the rotary axis corresponding to different mechanisms, please refers to 1.3 Definitions of Rotary Axis and 1.4 Parameter Descriptions.

	0(default)	**1**	**2**
Spindle/Table/ Mix	shortest path for 1st rotary axis (Master axis)	positive direction for 1st rotary axis	negative direction for 1st rotary axis

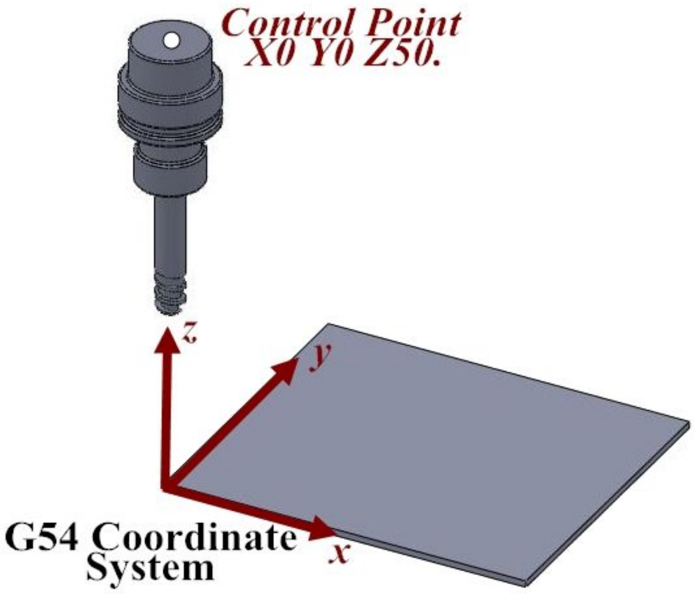
3.3.4 Program Example

Take the NC program below as example to explain the basic actions of Tilted Working Plane.

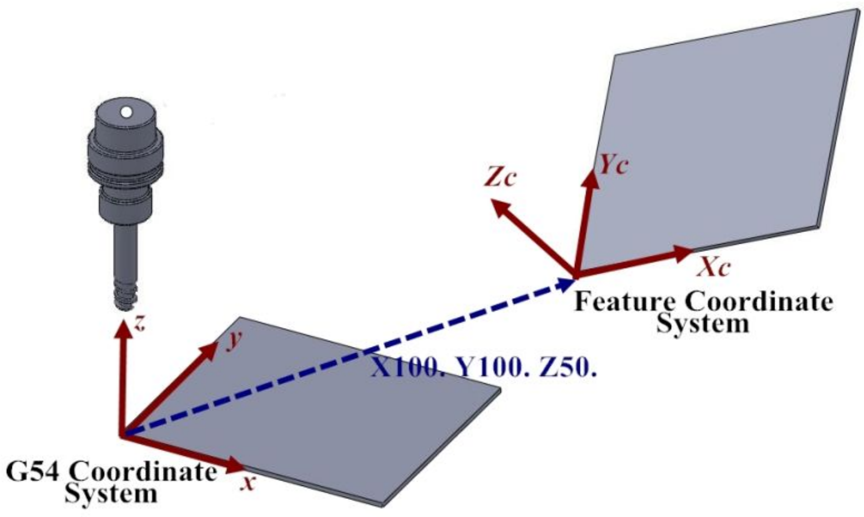
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
N3 G01 X0 Y0 Z50. F1000;
N4 G53.3 X0 Y0 Z0 H1;

... //  Tilted Working Plane machining

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

The actions of the NC program will be explain line by line:

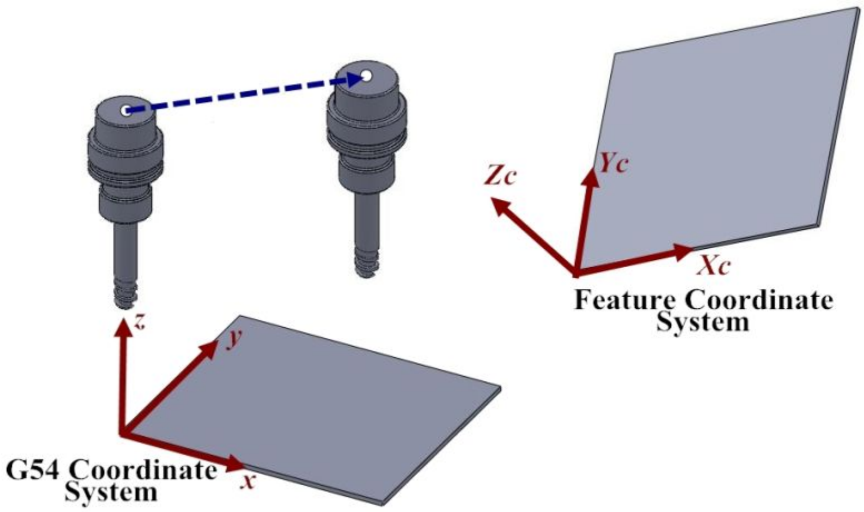
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of G54 coordinate by G01 in speed of 1000 mm/min.
```



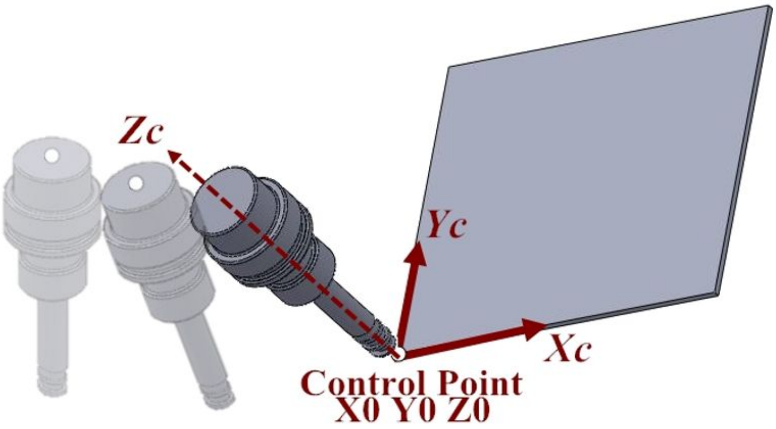

```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
// Specify X100. Y100. Z50. relative to the origin of G54 coordinate as the origin of
Tilted Working Plane, and the Euler angles are I30. J15. K20.
// The program coordinate will transform to Tilted Working Plane after G68.2 is
executed.
```



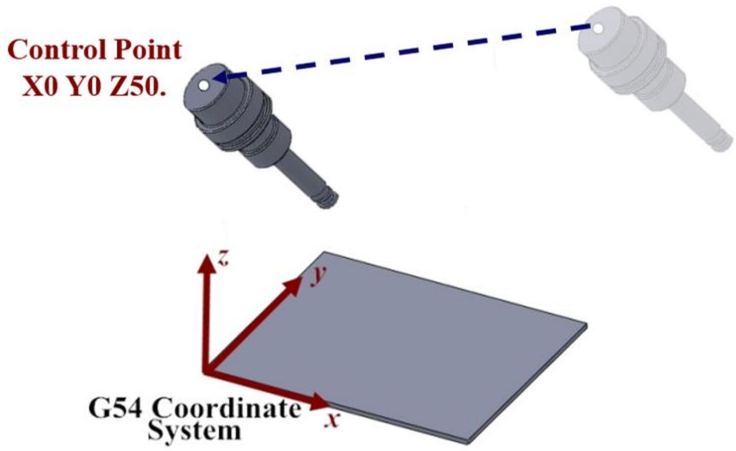

```
N3 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of Tilted Working Plane by G01 in speed of 1000 mm/min, but the tool direction remains the same.
```




```
N4 G53.3 X0 Y0 Z0 H1;
// Tool length compensation is enabled, the control point changes to the tool tip.
// The tool direction aligns to the Z axis of Tilted Working Plane, and moves to X0 Y0 Z0 of Tilted Working Plane in the speed of G00.
```




```
N98 G49;
// Cancel tool tip control.
N99 G69;
// Cancel Tilted Working Plane.
N100 G01 X0. Y0. Z50.;
// Moves to X0. Y0. Z50. of G54 coordinate.
```



3.4 3.4 G53.6 Tool Alignment Function for Tilted Working Plane (TCP/Rotation Center)

3.4.1 **Command Format**

G68.2 X_ Y_ Z_ I_ J_ K_;
G53.6 [H_] [P_] [R_];

G68.2: enable Tilted Working Plane function;
G53.6: tool alignment function (TCP/ Rotation Center);

H: tool number, using the previous tool number when H code is not given, if there's no previous tool number then alarm "MAR-407 Tool number can not be 0 while using G53.6" will be triggered.
P: define the rotating direction of the rotary axis, 0: shortest path for 1st rotary axis (Master axis) (default); 1: positive direction for 1st rotary axis; 2: negative direction for 1st rotary axis
R: the distance from tool center point to rotation center;

After G68.2 is executed and before the cutting commands(EX: G01), G53.1 or G53.3 or G53.6 is required for the tool to align to Tilted Working Plane.

3.4.2 Description

1. After Tilted Working Plane is enabled, G53.6 is required for the tool to align to Tilted Working Plane, thus this G code is attached under G68.2 and should exist at the same time.
2. Both G53.6 and G53.1 control the tool direction and align it to Tilted Working Plane, but the distance between tool center and rotation center will be the same during the alignment though G53.6. The distance can be assigned with G53.6 (by argument R). The figures below show the difference which is made by argument R :

- **Without R:** the tool center point keeps in place while the rotary axis is rotating.

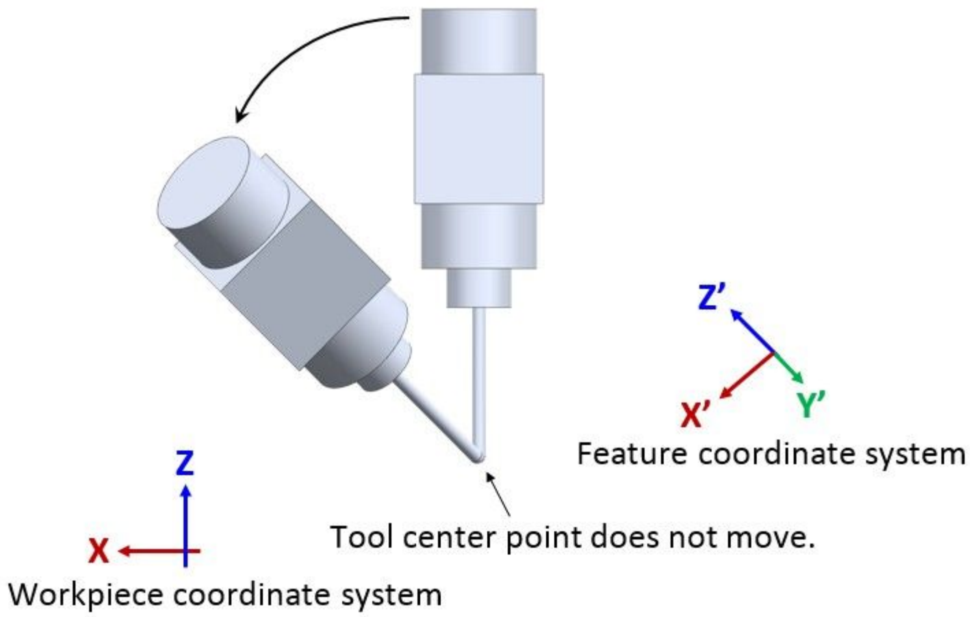


Fig.52

- **With R (Rr):** the rotation center, which was extended from the tool center point for distance r, keeps in place while the rotary axis is rotating.

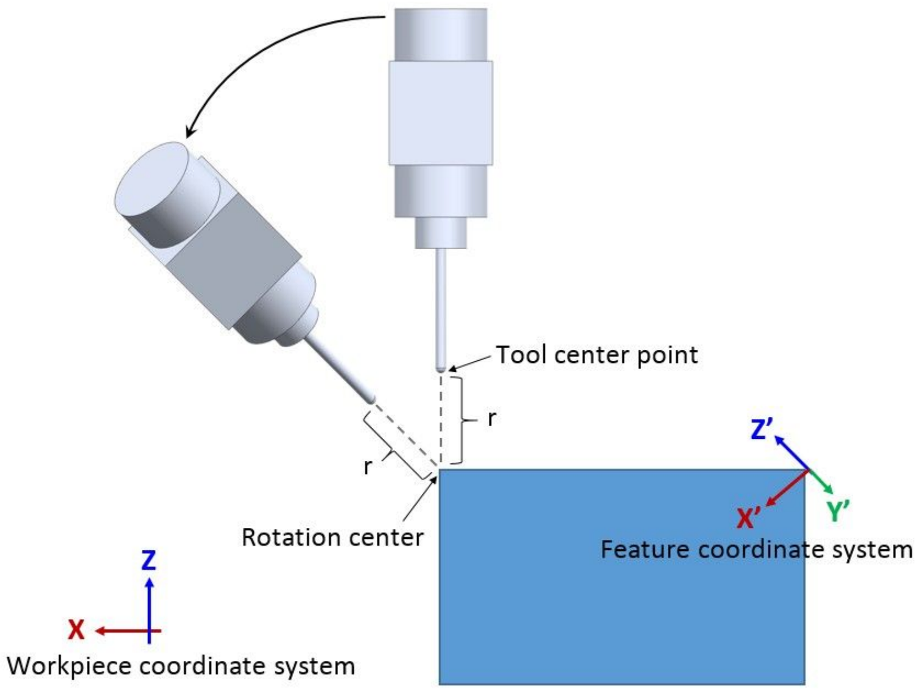


Fig.53

3.4.3 Notifications

1. G53.6 can't be executed before G68.2.
2. Please apply positive tool length (G53.6 could assign the tool number with H code).
3. The tool rotation will be executed in the way of RTCP after G53.6 is executed, the control object of the follow-up commands is the tool tip. User should apply G49 when cutting is finish to cancel the tool tip control.
4. Do not execute G41, G42 before G53.6, or alarm 【MAR-406 G53.6 must be enabled in G40 mode】 will occur.
5. If G53.6 is executed without H argument and the current tool number is 0, alarm 【MAR-407 Tool number can not be 0 while using G53.6】 will occur.
6. The P argument will be 0 in default if it's not specified.
7. If the value of P argument is out of range, alarm 【COR-149 G53.1/G53.6 P Argument over range】 will occur.
8. When P is 0, the system will search for the shortest path for 1st rotary axis (Master axis) first. If the target angle or the path is out of range (defined by Pr3009~), the other target angle will be applied instead; if both target angles or paths are out of range, alarm 【COR-153 no solution for this tool direction】 will occur.
9. When P is 1 or 2, if the target angle or the path is out of range (defined by Pr3009~), alarm 【COR-153 no solution for this tool direction】 will occur.
10. For the definitions of the rotary axis corresponding to different mechanisms, please refers to 1.3 Definitions of Rotary Axis and 1.4 Parameter Descriptions.

	0(default)	**1**	**2**
Spindle/Table/ Mix	shortest path for 1st rotary axis (Master axis)	positive direction for 1st rotary axis	negative direction for 1st rotary axis

3.4.4 Program Example

Take the NC program below as example to explain the basic actions of Tilted Working Plane.

```
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 actions of the NC program will be explain line by line:

```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of G54 coordinate by G01 in speed of 1000 mm/min.
```

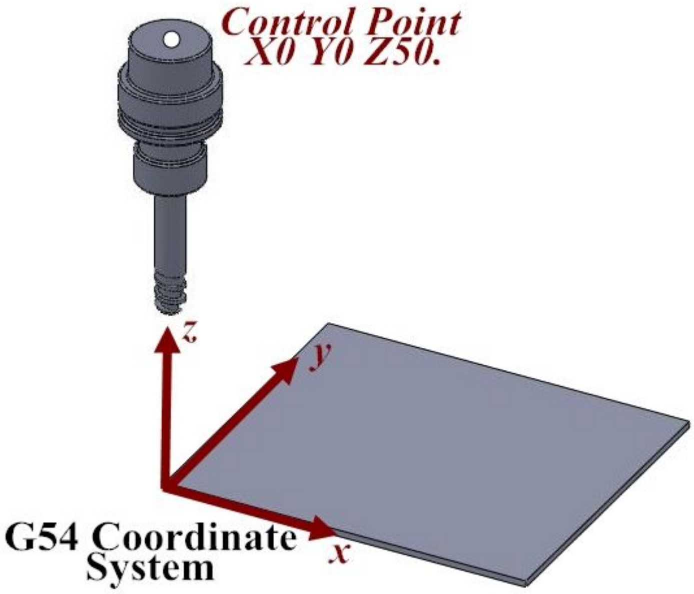


Fig.54

```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
// Specify X100. Y100. Z50. relative to the origin of G54 coordinate as the origin of Tilted Working Plane, and the Euler angles are I30. J15. K20.
// The program coordinate will transform to Tilted Working Plane after G68.2 is executed.
```

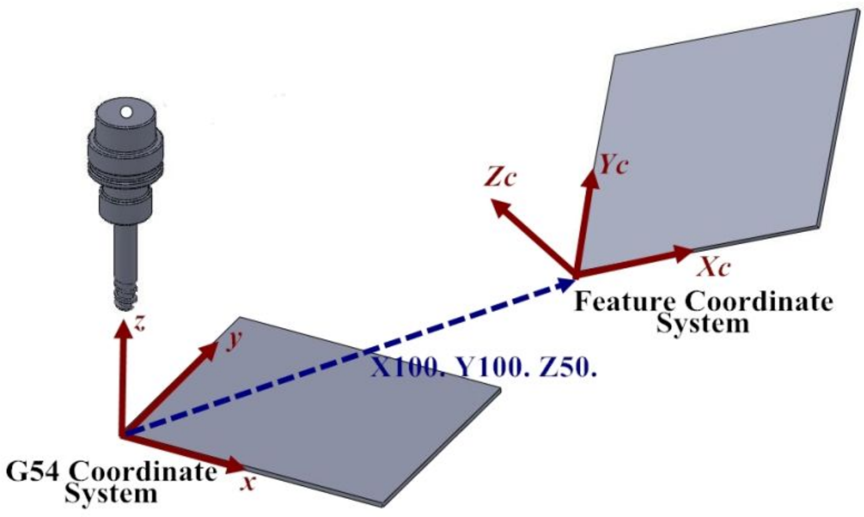


Fig.55

```
N3 G53.6;
// The tool direction aligns to the Z axis of Tilted Working Plane.
```

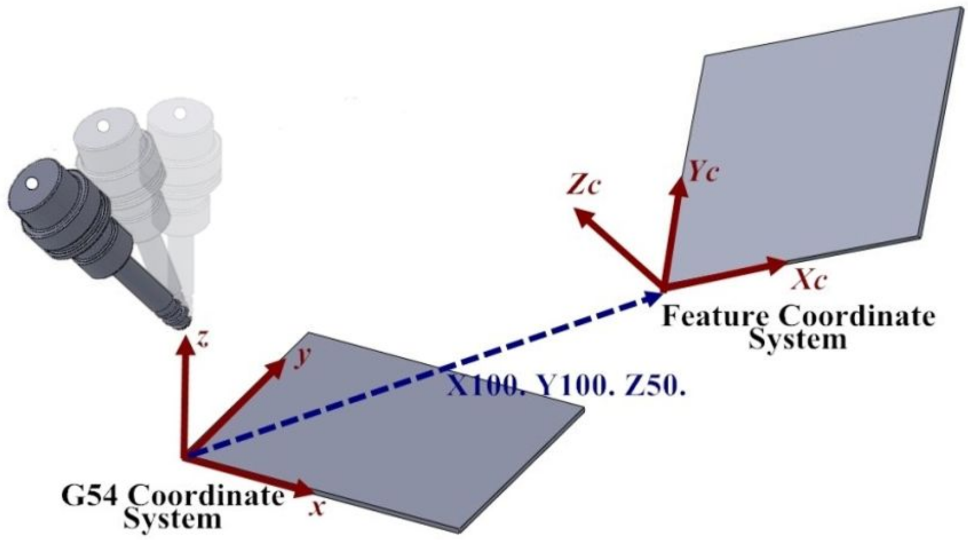


Fig.56

```
N4 G01 X0 Y0 Z0;
// The tool tip moves to X0 Y0 Z0 of Tilted Working Plane.
```

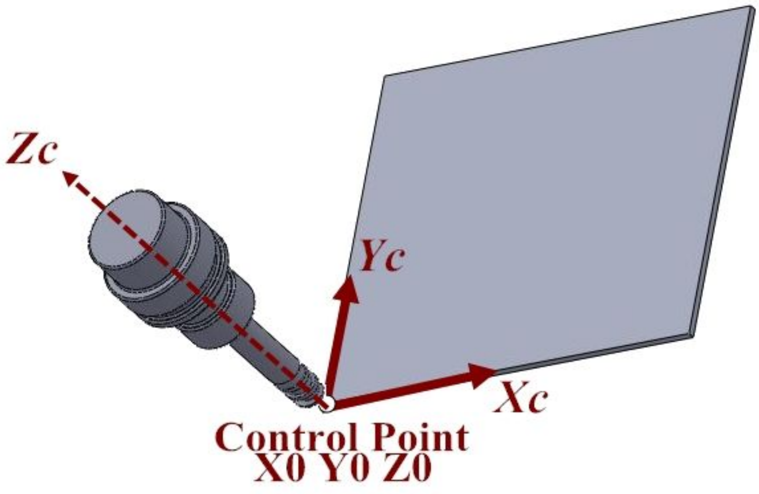


Fig.57

```
N98 G49;
// Cancel tool tip control.
N99 G69;
// Cancel Tilted Working Plane.
N100 G01 X0. Y0. Z50.;
// Moves to X0. Y0. Z50. of G54 coordinate.
```

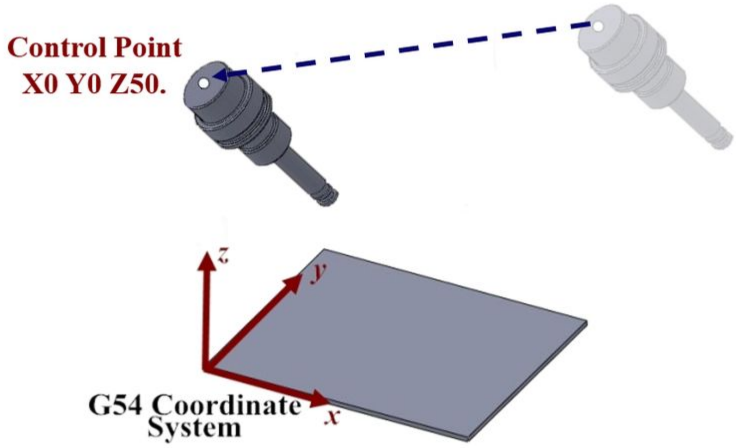
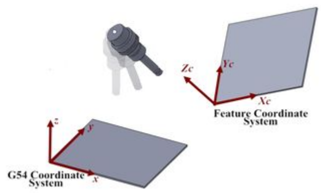
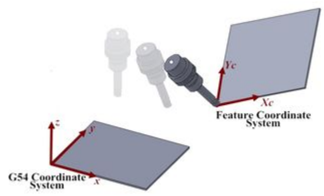
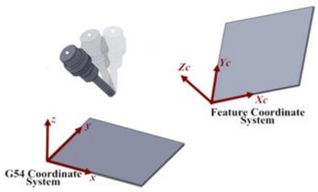


Fig.58

3.5 3.5 Comparison Between Tool Alignment Functions

Command Format	Description	Figure	Application	Valid Version
G53.1 [P_]	• Rotary axis rotate and nothing more. • The tool will align to Tilted Working Plane with G00. • The position of tool tip varies during alignment.		For machines prone to interference, this command is usually used for the tool to align to Tilted Working Plane after retracting to a safety height.	G code itself : • from start Allows to choose rotating direction : • 10.116.54I • 10.118.0E • 10.118.5
G53.3 [X_] [Y_] [Z_] [H_] [P_]	• Rotary axis rotate and the position of tool tip can be specified. • The tool will align to Tilted Working Plane with G00, and the target position of tool tip is specified as **(X_Y_Z_)** on Tilted Working Plane.		Positioning and alignment are completed simultaneously to save time. For rapid tool changing on the special mechanism such as the machines with multi sets of RTCP.	• 10.118.28G • 10.118.33

Command Format	Description	Figure	Application	Valid Version
G53.6 [H_] [P_] [R_]	• The tool will align to Tilted Working Plane with G00, and the tool tip keeps in place during alignment.		For measurement of 5-axis machine.	G code itself : • 10.116.45 Allows to choose rotating direction : • 10.118.28G • 10.118.33

3.6 3.6 G68.3 Tilted Working Plane Machining (Tool Direction)

G68.2 determines Tilted Working Plane with Euler angle, and G68.3 takes the tool direction as Z axis of Tilted Working Plane and generates XY plane perpendicular to Z axis automatically.

Rotates the tool till it's perpendicular to the tilted plane on the workpiece and determine Tilted Working Plane with G68.3, then we can process 3-axis machining on the tilted plane as shown in Fig.57.

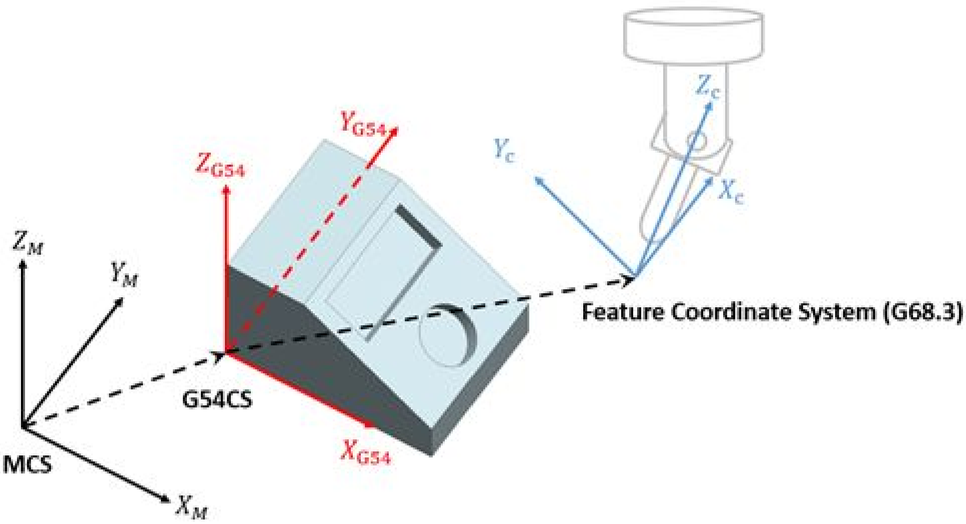


Fig.57

3.6.1 Command Format

With G68.3, the reference coordinate of NC program will be transformed to Tilted Working Plane.

Before G69 is executed, all commands will be seen as the commands for Tilted Working Plane and be executed based on it.

There are 2 formats for G68.3 function.

Type 1 :

```
G68.3 X_ Y_ Z_ R_;
G69;
```

G68.3: enable Tilted Working Plane function, Tilted Working Plane is defined by outer product;

G69: disable Tilted Working Plane function;

X_ Y_ Z_: the origin of Tilted Working Plane (relative to the origin of G54 coordinate);

R_: after outer product, rotate along the tool vector (Z axis) for angle R.

Description with picture:

G68.3 X***X0*** Y***Y0*** Z***Z0*** R***R***

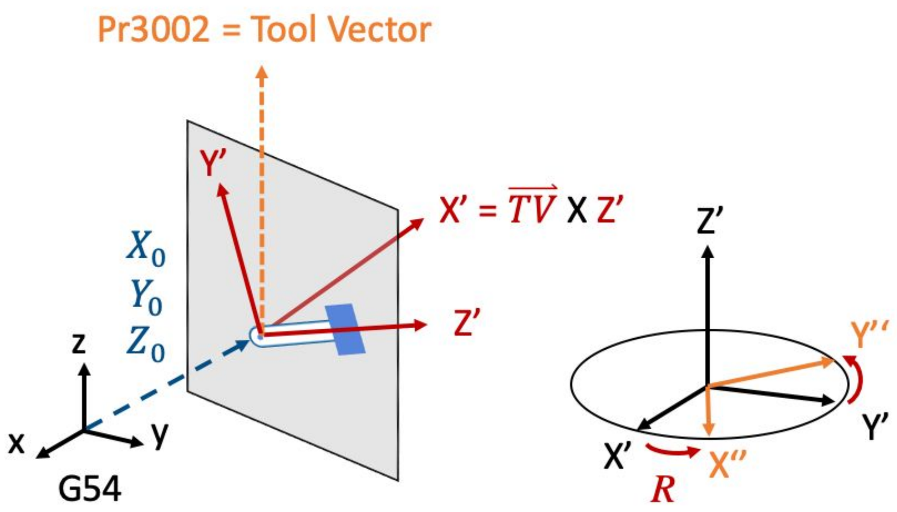


Fig.58

Type 2:

```
G68.3 P1 X_ Y_ Z_;
G69;
```

G68.3: enable Tilted Working Plane function;

P1: define Tilted Working Plane with the rotation angle of the tool;

G69: disable Tilted Working Plane function;

X_ Y_ Z_: the origin of Tilted Working Plane (relative to the origin of G54 coordinate);

Description with picture:

Before G68.3 P1 is executed, the angles of the tool are C45° and B10°.

G68.3 P1 X***X0*** Y***Y0*** Z***Z0***

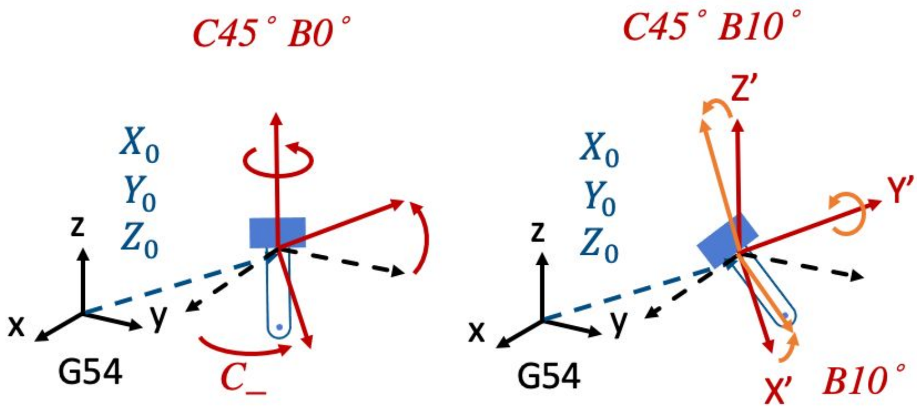


Fig.59

3.6.2 Limitations

1. When G68.3 is executed, all of XYZ need to exist or not exist at the same time or alarm 【COR-141 Illegal G68.3 input argument】 will be triggered.
2. If XYZ is not given, then current position will be taken as the origin of Tilted Working Plane.
3. Restrictions on combining with other commands:
    a. G43 should be executed after G68.3.
    b. G68.3 can be used after RTCP (G43.4/G43.5) is turned on; however RTCP cannot be used after G68.3 is turned on,otherwise, COR-381 G68.3 function usage error will be issued.
    c. G68.3 can not use with G54.4, otherwise, COR-381 G68.3 function usage error will be issued.
    d. COR-381 available version: 10.118.86S, 10.120.6B, 10.120.12Z, 10.120.16S, 10.120.24F, 10.120.28B, 10.120.32 and later versions.
4. G43 should be executed after G68.3.
5. G68.3 can NOT be executed while RTCP (G43.4/G43.5) is enabled.
6. When G68.3 P1 is executed, R argument will be ignored if provided.
7. G68.3 can be executed for multiple times and each setting is relative to G54 coordinate.
8. G68.3 is only for 5-axis machines with option-13 enabled at the same time.

3.6.3 Related Parameters

Parameters below defines the initial tool direction:

No.	Description	Range	Unit	Initial Value	Take Effect
3001	*1st organization for five axis machine	[0,3]	-	0	Reboot
3002	1st direction of Tool	[0,3]	-	0	Reset
3003	1st incline Angle of direction of Tool (RA)	[0,360000]	BLU	0	Reset
3004	1st incline Angle of direction of Tool (RB)	[0,360000]	BLU	0	Reset
3013	1st tool Holder Offset	[0,999999999]	BLU	0	Reset

No.	Description	Range	Unit	Application Introduction
3014	feature coordinate persist mode	[0,2]	-	0: Do NOT preserve feature coordinate status defined by G68.2/G68.3 after reset & reboot. 1: Preserve feature coordinate status defined by G68.2/G68.3 after reset only. 2: Preserve feature coordinate status defined by G68.2/G68.3 after reset and reboot.

3.6.4 Program Example:

3.6.5 (1)

NC program and Fig.60~63 are used to explain the relations of coordinate transformation when G68.3 & tool length compensation are applied.

```
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;        // Define Tilted Working Plane with outer product according to the
tool direction and the control point is point A.
N4 G43 H1;       // Control point transforms to tool center point C.
N5 X0. Y0. Z0.;
```

```
N6 G69 G49;
```

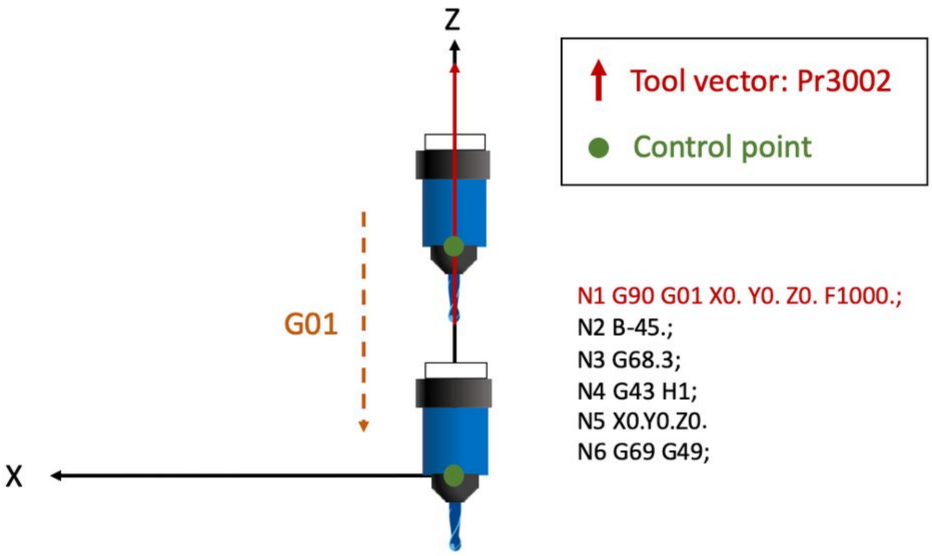


Fig.60

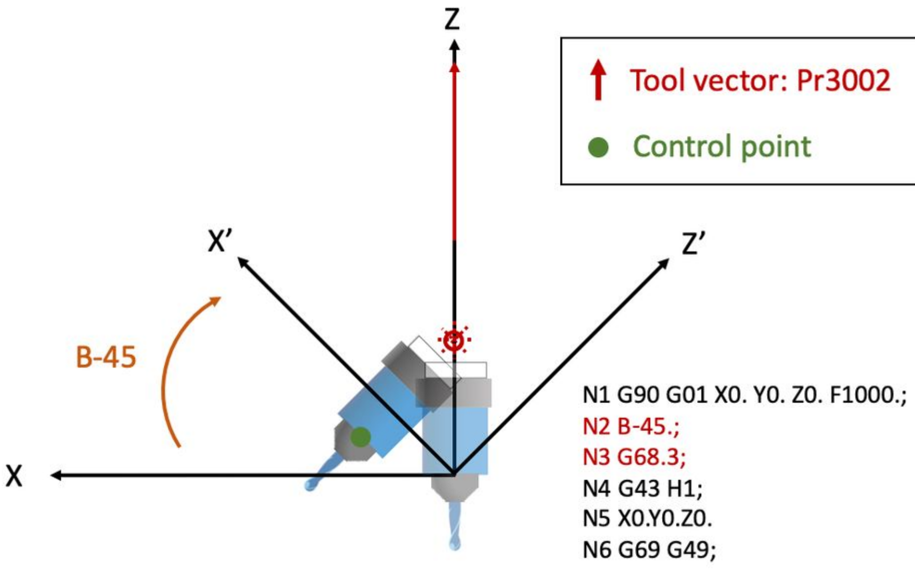


Fig.61

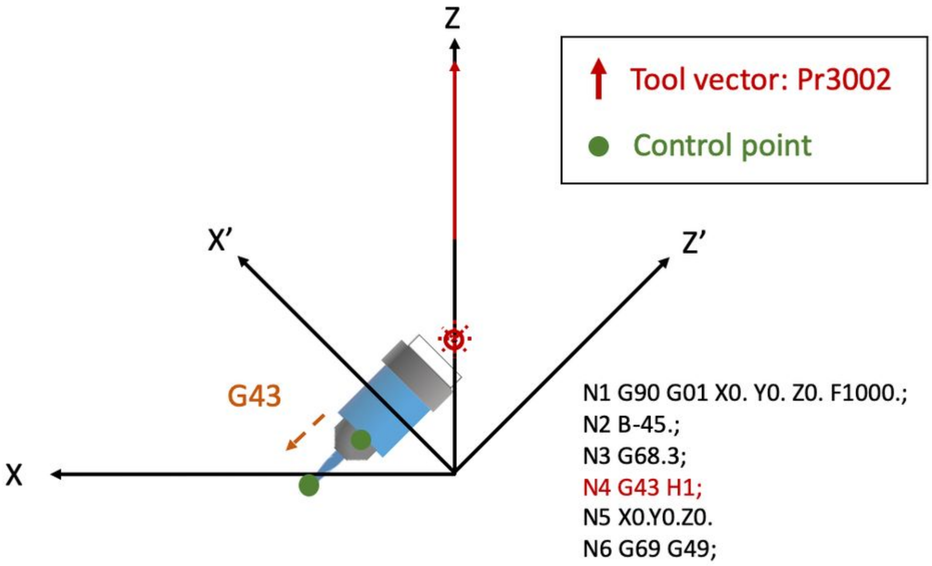


Fig.62

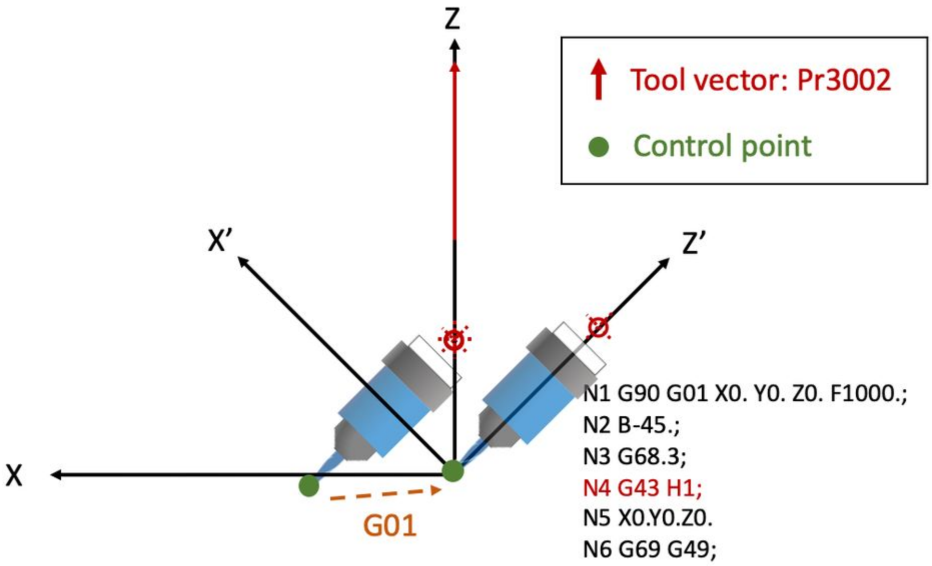


Fig.63

3.6.6 (2)

NC program and Fig.64~66 are used to explain the motions within Tilted Working Plane when G68.3 is enabled.

```
N1 G54 G90 G00 B0. C0.;
N2 C45.;
N3 B90.;
```

```
N4 G68.3 P1 X0. Y0. Z0.;       // Define Tilted Working Plane with the rotation angle
of the tool.
N5 G01 Y10.;                   // Moves to Y10. within Tilted Working Plane, but for
G54 coordinate it moves to X7.071 Y7.071.
N6 G69;
N7 G00 X0. Y0. Z0. B0. C0.;
N8 M30;
```

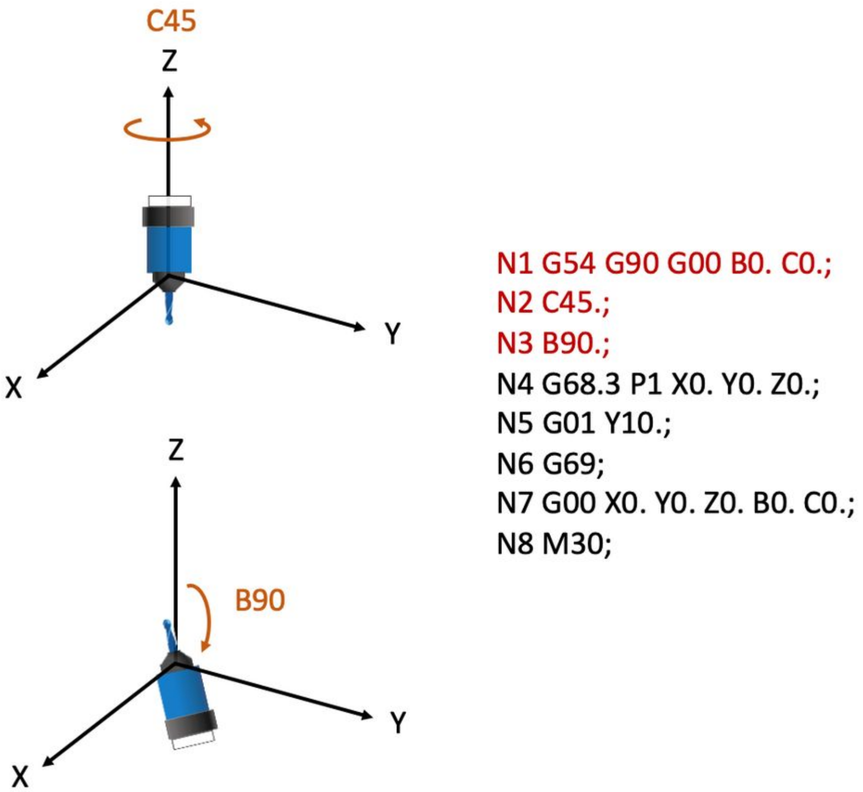
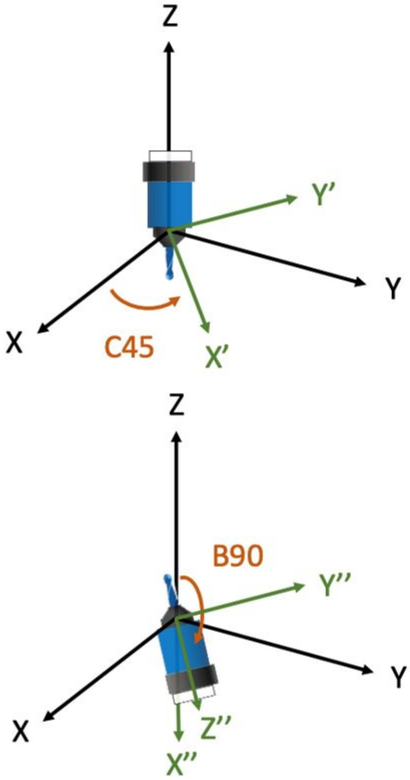
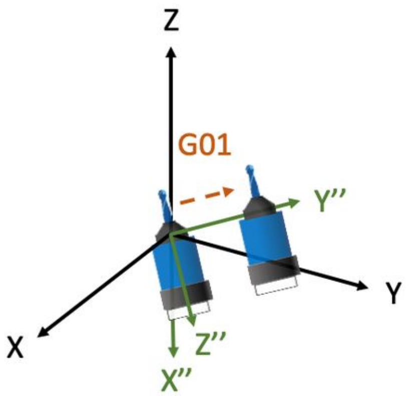


Fig.64




```
N1 G54 G90 G00 B0. C0.;
N2 C45.;
N3 B90.;
N4 G68.3 P1 X0. Y0. Z0.;
N5 G01 Y10.;
N6 G69;
N7 G00 X0. Y0. Z0. B0. C0.;
N8 M30;
```

Fig.65




```
N1 G54 G90 G00 B0. C0.;
N2 C45.;
N3 B90.;
N4 G68.3 P1 X0. Y0. Z0.;
N5 G01 Y10.;
N6 G69;
N7 G00 X0. Y0. Z0. B0. C0.;
N8 M30;
```

Fig.66

3.6.7 （3）

NC program and Fig.67、68 are used to explain the motions when G68.3 is executed for multiple times.

```
N1 G55;
N2 G01 A90. F1000.;        // Tool rotates, A axis rotates for 90 degrees. (right-hand rule)
N3 G68.3 X0 Y0 Z0 R0;      // Define Tilted Working Plane with outer product. (green coordinate)
N4 X10. Y0. Z0.;           // Moves to X10. Y0. Z0. within Tilted Working Plane.
N5 C90.;                   // Tool rotates, C axis rotates for 90 degrees. (right-hand rule)
N6 G68.3 X10. Y0. Z0. R0;  // Define Tilted Working Plane (purple coordinate) according to the new tool direction.
N7 X0. Y0. Z0.;            // Moves to X0. Y0. Z0. within Tilted Working Plane.
N8 G69;
```

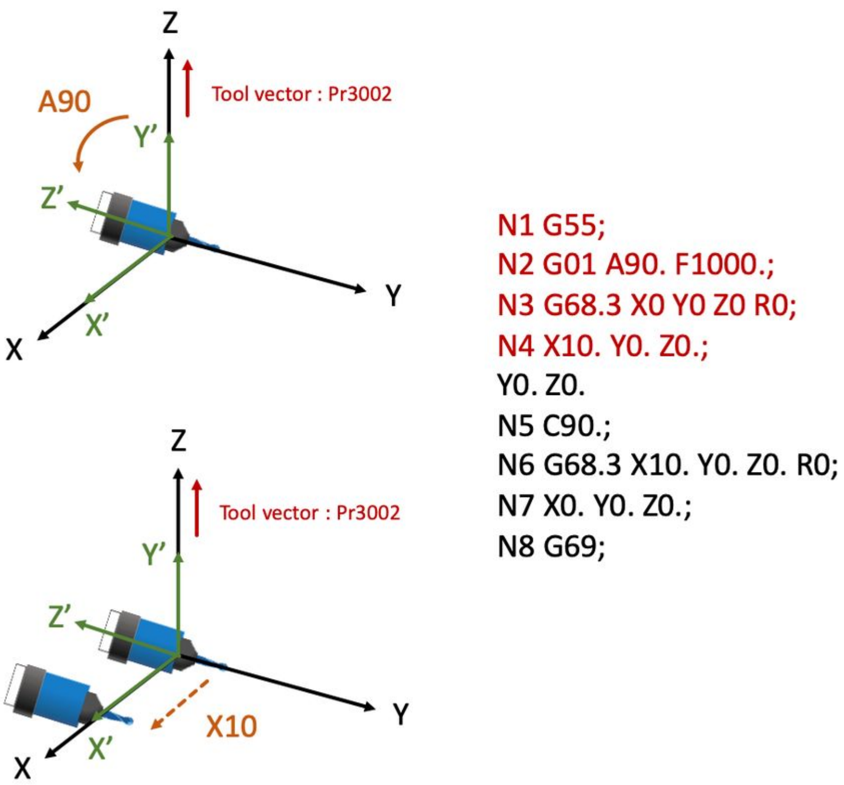
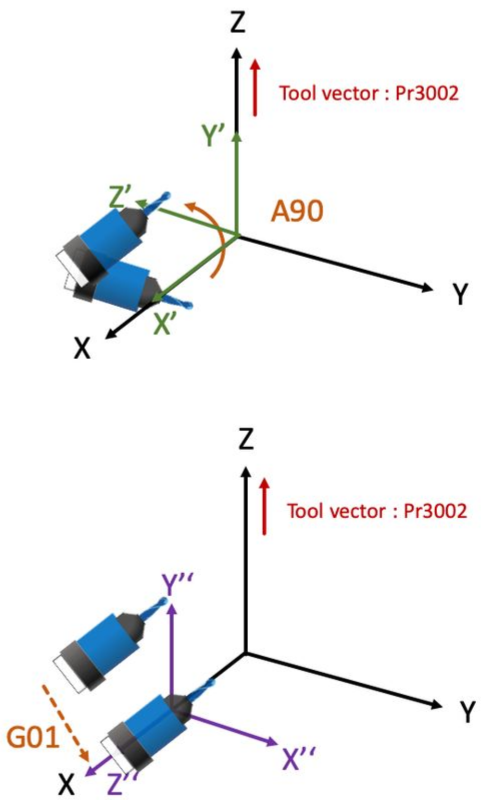


Fig.67




```
N1 G55;
N2 G01 A90. F1000.;
N3 G68.3 X0 Y0 Z0 R0;
N4 X10. Y0. Z0.;
Y0. Z0.
N5 C90.;
N6 G68.3 X10. Y0. Z0. R0;
N7 X0. Y0. Z0.;
N8 G69;
```

Fig.68

3.7 3.7 Program Example of Tilted Working Plane Machining

3.7.1 Example Description

The key of Tilted Working Plane machining is to define Tilted Working Plane, and it actually takes only two blocks to complete this action.

The remaining part of NC program is totally the same as 3-axis machining, so there's no need to generate the NC program for Tilted Working Plane by CAM additionally.

We will explain how to modify a 3-axis machining program into a Tilted Working Plane machining program in this section.

As shown in Fig.69, there's a workpiece with 100mm in length & width, and two 15° inclined planes cross on the top.

And now we are going to carve a line of word with same depth on each plane.

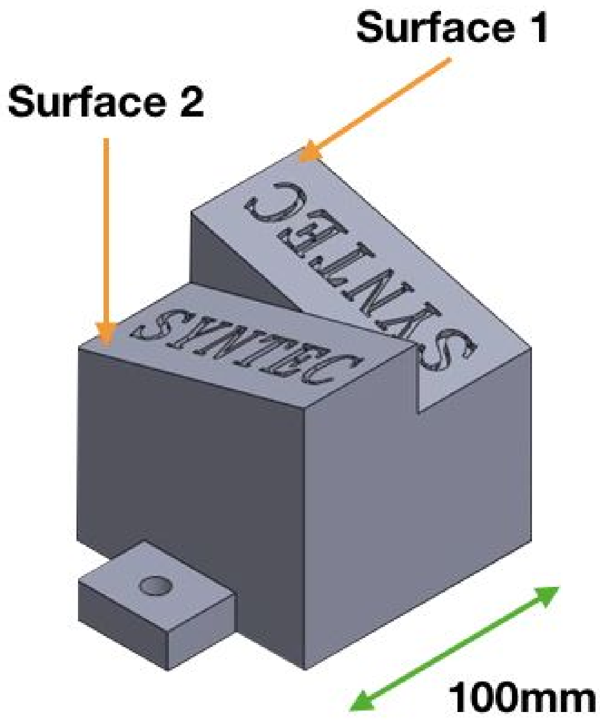


Fig.69

3.7.2 Program Modification

Generate a 3-axis NC program by CAM, the origin of the program is at the bottom left corner of the inclined plane (Fig.70), the program is shown below.

```
G71
G17 G40 G49 G90 G80
G91 G28 Z0.0
G90 G54 G00 X11.4608 Y24.1067
G43 G00 Z10. H01
S20000 M03
G01 X11.4608 Y24.1067 F1000.
Z-.15
………
```

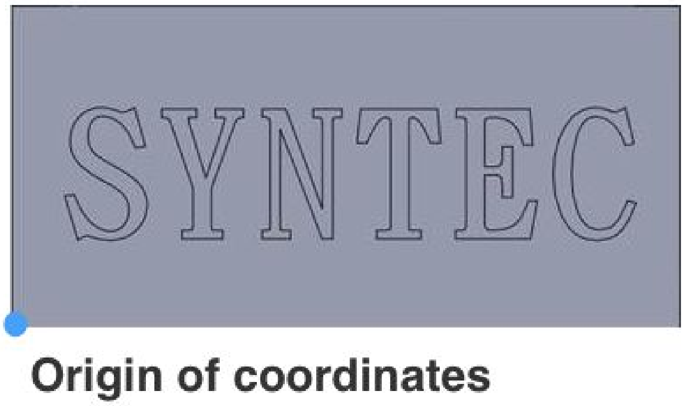


Fig.70

After G54 coordinate is set, insert related commands (G68.2, G53.1/G53.6) to execute Tilted Working Plane machining, the program is shown below.

```
G71
G17 G40 G49 G90 G80
G91 G28 Z0.0
G90 G54 G00 X11.4608 Y24.1067
G68.2 X100. Y0. Z-26.7 I0. J15. K90.
G53.1
G43 G00 Z10. H01
S20000 M03
G01 X11.4608 Y24.1067 F1000.
Z-.15
.........
```

3.7.3 Setting Origin of Tilted Working Plane

The origin of Tilted Working Plane is assigned relative to the origin of G54 coordinate, and there is no need to consider the direction.

Take Surface 1 as example, the origin of Tilted Working Plane offsets in X and Z direction from the origin of G54 coordinate (Fig.71, Fig.72), thus set X100. Y0. Z-26.7 (100*Sin15°).

For Surface 2, the offset will become X0. Y100. Z-26.7.

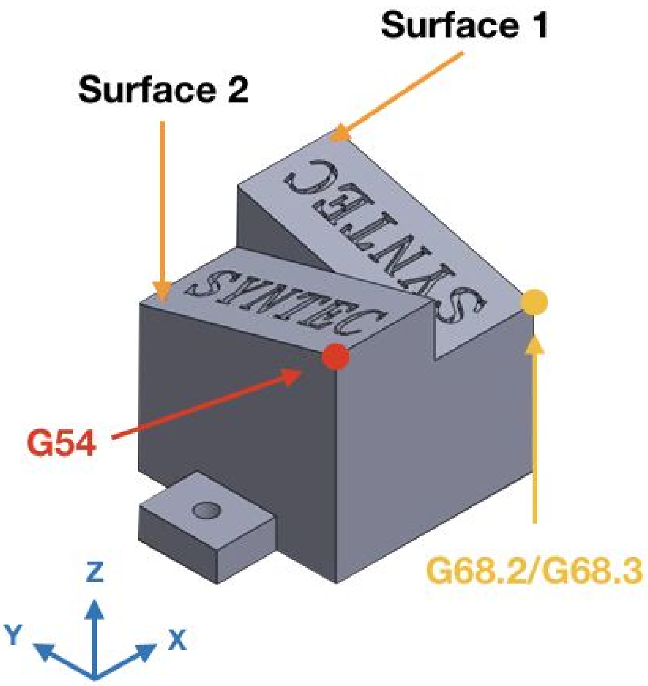


Fig.71

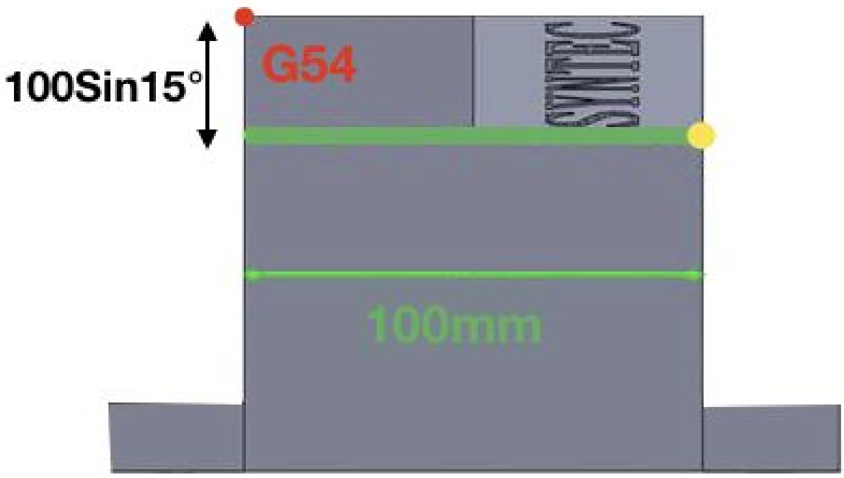


Fig.72

3.7.4 **Setting Euler Angle**

As shown in Fig.73, the program coordinate on Surface 1 is different from the machine coordinate, thus it requires to be transformed by Euler angle.

We can refer to the definition of Euler angle (I, J, K), and then find that they're 0, 15, 90 for Surface 1; 0, -15, 270 for Surface 2.

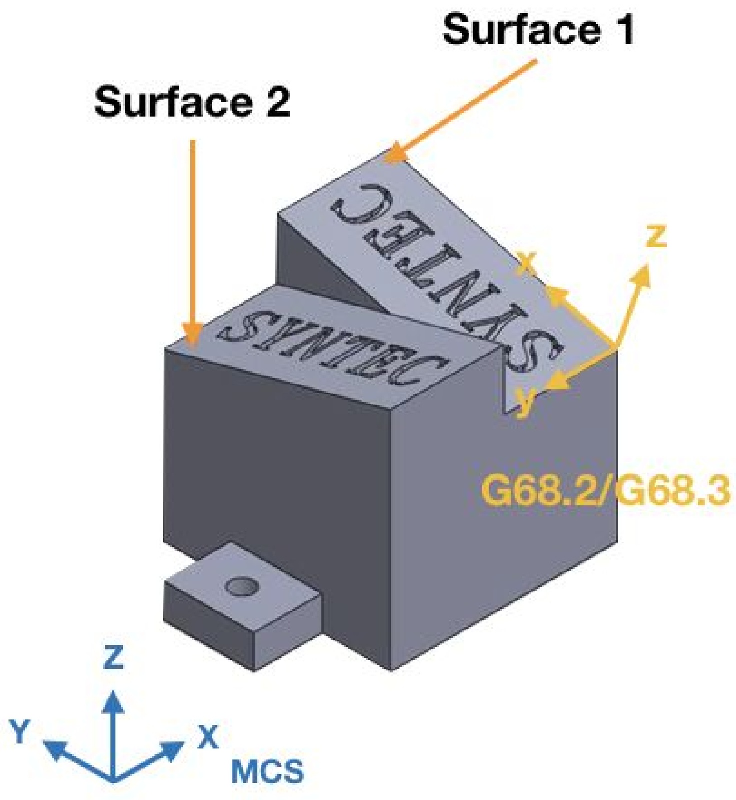


Fig.73

3.7.5 **Tool Alignment**

Please remember to give G53.1 or G53.3 or G53.6 after setting the origin and Euler angle, or the tool will have correct tool tip position but not align to the machining surface.

For table type 5-axis machines, when executing G53.1 or G53.3 or G53.6, the table will rotate till the machining surface aligns to the tool.

For spindle type 5-axis machines, the tool will rotate till it aligns to the machining surface.

3.8 3.8 Tilted Working Plane Related Parameters

3.8.1 Parameter Description

Version	No	Descriptions	Range	Unit	Details	Effective

V10	Pr3014		Feature coordinate persist mode	[0,2]	-	0: Do NOT preserve feature coordinate status defined by G68.2/G68.3 after reset & reboot. 1: Preserve feature coordinate status defined by G68.2/G68.3 after reset only. 2: Preserve feature coordinate status defined by G68.2/G68.3 after reset and reboot.	Reset
V12	P90 2	N 1					

3.9 3.9 Tilted Working Plane Teach Function

3.9.1 Function Description

Tilted Working Plane (or so-called Feature Coordinate) Teach function is placed in "Offset / Setting", the function screen is shown as Fig.38:

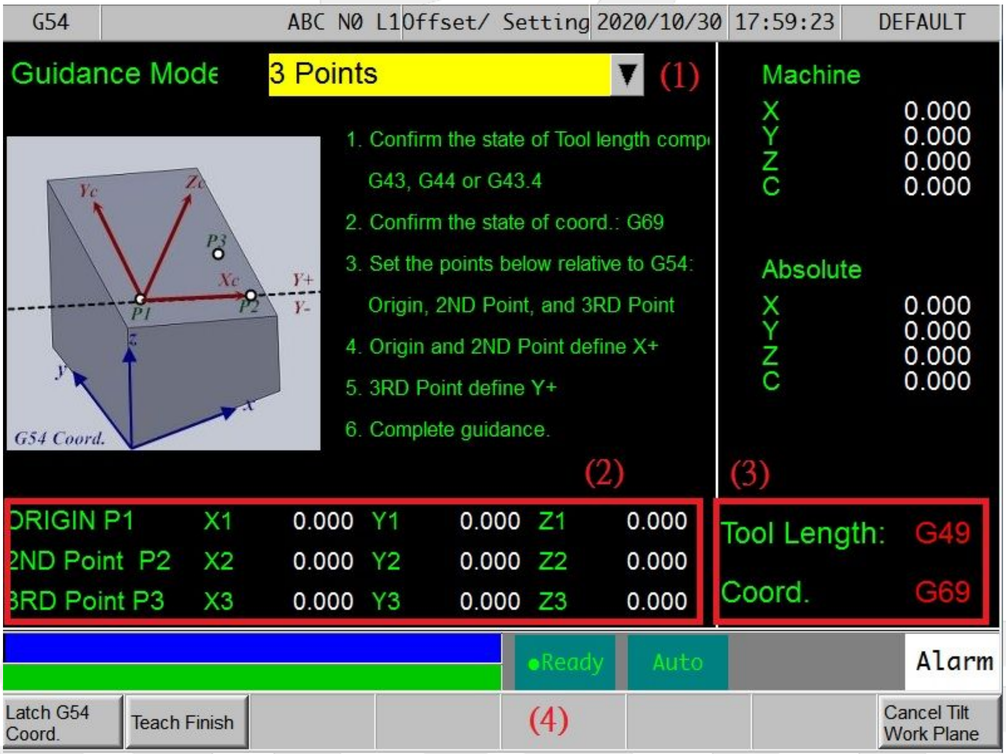


Fig.38

Function description:

1. Guidance mode: To select the teach mode.
2. Setting area: To set the required value according to different teach modes.
3. State display area: To show the current state of tool length compensation and coordinate transformation.
4. Function key

F1： Latch G54 Coordinate: To set the current "absolute coordinate" to the input box specified.

F2： Teach Finish: To transform the current coordinate to the tilted working plane coordinate just taught, it's effective before executing G69 command.

F8： Cancel Tilt Work Plane: To reset the coordinate back to G69.

[Note]
1. Before applying "F1 Latch G54 Coordinate", please execute G43.4, G43 or G44 first. It's not available when the state of tool length compensation is G49.
2. "F1 Latch G54 Coordinate" is only available with coordinate state being G69.

3.9.2 Three Points

Define the directions of X, Y, Z on the tilted working plane by setting coordinates of 3 individual points on tilted working plane.

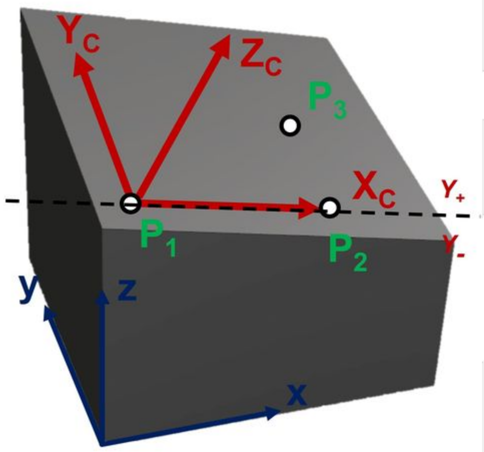


Fig.39

Setting Data

#	Name	Teach Input	Description
P1	Origin of tilted working plane	Yes	Define as the origin of the tilted working plane.
P2	Second point of tilted working plane	Yes	The direction from P1 to P2 will define X+ direction of the tilted working plane. The X axis cuts the tilted working plane into two areas, Y+ and Y-.
P3	Third point of tilted working plane	Yes	Determine Y+ direction of the tilted working plane.

> **[Note]**
> The teach will fail if 3 setting points are collinear, and the coordinate status will remain in G69 mode.

3.9.3 Tool Direction

Define the directions of X, Y, Z on the tilted working plane with current tool direction.

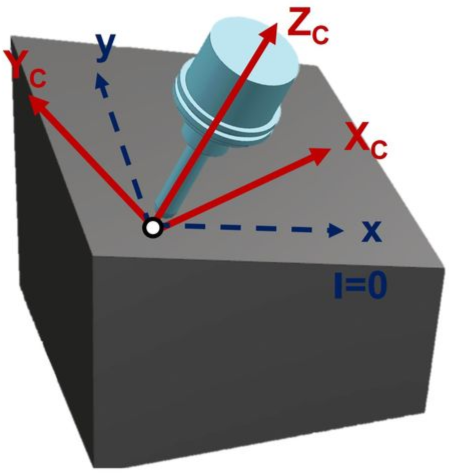


Fig.40

Setting Data

#	Name	Teach Input	Description
P1	Origin of tilted working plane	Yes	Define as the origin of the tilted working plane. Assume facing tool tip from tool holder, the right hand direction is defined as X+ direction. The tool axis is defined as Z axis, thus a XYZ coordinate is defined.
I	Rotation angle of tool	No	The X, Y, Z directions of tilted working plane are determined after rotating the coordinate for angle I.

3.9.4 Euler Angle

Define the directions of X, Y, Z on tilted working plane by setting Euler angles.

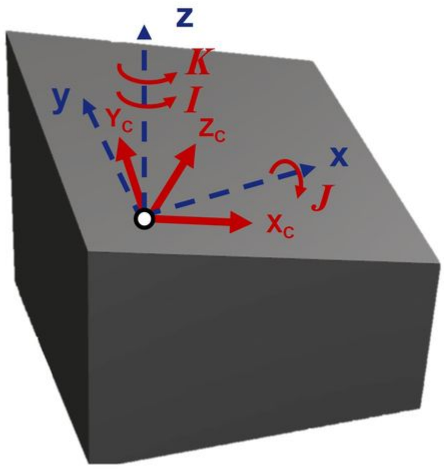


Fig.41

Setting Data

#	Name	Teach Input	Description
P1	Origin of tilted working plane	Yes	Define as the origin of the tilted working plane.
I	1st Euler angle	No	Rotation angle around Z axis, XYZ coordinate becomes to X'Y'Z after rotation.
J	2nd Euler angle	No	Rotation angle around X' axis, X'Y'Z coordinate becomes to X'Y"Z' after rotation.
K	3rd Euler angle	No	Rotation angle around Z' axis, X'Y''Z' coordinate becomes to XcYcZc after rotation, which is the directions of XYZ on tilted working plane.

[Note]
Please refers to 3.1 G68.2 Tilted Working Plane Machining (Euler Angle) for the definition of Euler angle.

3.9.5 2 Vectors

Define the tilted working plane by setting the X axis and Z axis of the tilted working plane.

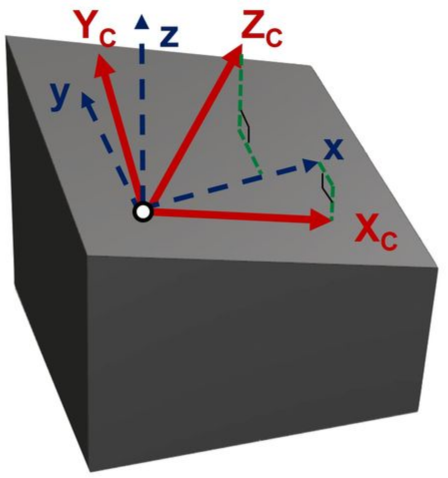


Fig.42

Setting Data

#	Name	Teach Input	Description
P1	Origin of tilted working plane	Yes	Define as the origin of the tilted working plane.
Xc	X axis of tilted working plane	No	Vector components of X axis on tilted working plane related to G54 coordinate.
Zc	Z axis of tilted working plane	No	Vector components of Z axis on tilted working plane related to G54 coordinate.

[Note]
The teach will fail if the situations below are met:
1. The setting X axis and Z axis are not orthogonal.
2. The setting X axis or Z axis is a zero-vector.

4 4. Workpiece Coordinate Rotation Function

This chapter will introduce the usage of workpiece coordinate rotation function, along with its operational specifications and examples.

4.1 4.1 Command Format and Description

4.1.1 Command Format

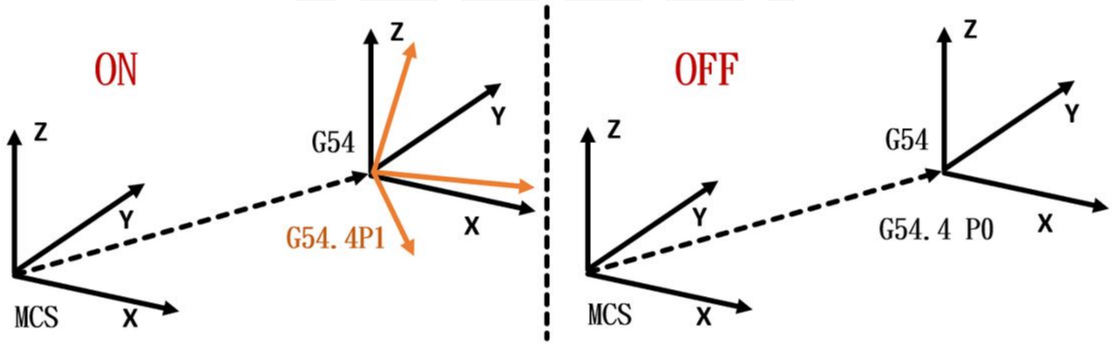
G54.4 P_;

P: Workpiece Coordinate Rotation function switch. Setting **0** disables the current Workpiece Coordinate Rotation value, and setting **1** enables the current Workpiece Coordinate Rotation value.

4.1.2 Description

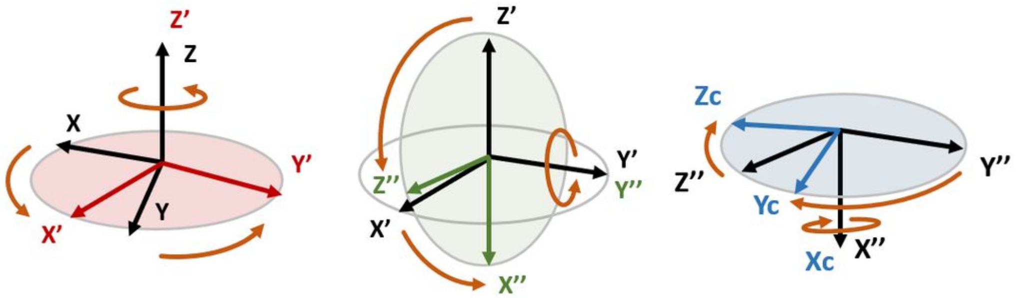
G54.4 command extends the usability of the workpiece coordinate system (G54) by allowing temporary deactivation of the Workpiece Coordinate Rotation function within the NC file. This enhances customization flexibility and convenience.

1. The program coordinate system can rotate along the XYZ axes, enabling it to align with any desired angle:
    a. Illustration of enabling/disabling rotation: (Enabled/Disabled).



2. Limitations when using G54.4 P1 for coordinate system rotation:
    a. A specific G54 P_ coordinate system must be designated. If none is specified, the current coordinate system rotation will be activated.
    b. After enabling, if the machining operation is not simultaneous five-axis machining (G43.4/G43.5), the coordinate system must first be aligned (G53.1/G53.3/G53.6).
    c. After enabling, tool length compensation (positive tool length) must be activated to compensate the coordinate system from the tool holder to the tool tip.
    d. Positive tool length compensation includes G43, G43.4, and G43.5.
3. This feature is available only on specific milling machine models. For other models without the setting interface, the function can still be used:
    a. 210MA-E5, 210MB-E5, 610MA-E5, 610MA-H5
    b. 22MA(PLUS), 220MA(PLUS), 220MB(PLUS)
    c. 220MA-5(PLUS), 220MB-5(PLUS)
4. The Euler angle settings follow a defined sequence. The three angles on the Workpiece Coordinate Rotation interface (XYZ) rotate in the order Z-axis → Y-axis → X-axis:
    a. First, rotate the original XYZ Cartesian coordinate system around the Z-axis by a certain angle to obtain the new coordinate system X'Y'Z'. This angle is defined as the Z angle.

b. Then, rotate X'Y'Z' around the Y'-axis by a certain angle to obtain X"Y"Z". This angle is defined as the Y angle.
c. Finally, rotate X"Y"Z" around the X"-axis by a certain angle to obtain the final coordinate system XcYcZc. This angle is defined as the X angle.
d. XcYcZc represents the final inclined plane coordinate system.



4.2 4.2 Program Examples

4.2.1 Example 1

Using a Macro during the tool setting process as an example, the G54.4 P_ command can be used to switch the Workpiece Coordinate Rotation function on or off:

Tool Setting Macro

```
N1 Backup Modes();                            // Backup all modes before
Macro execution
N2 #100 = #1849;                              // Backup the current
coordinate rotation state

N3 G54.4
P0;                                   // Disable coordinate rotation
N4
G91;                                        // Use incremental mode for the
tool setting program
N5 Measure Workpiece Inclination();           // Measure the rotation values
for the coordinate system using multiple G31 commands
N6 Set Coordinate Rotation Values();          // Assign the measured
rotation values to corresponding # variables

N7 G54.4
P1;                                   // Enable coordinate rotation
N8 Centering Program();                       // Measure the center of the
workpiece using multiple G31 commands
N9 Set Coordinate Translation Values();       // Assign the measured center
to the workpiece coordinate origin

N10 G54.4 P#100;                              // Restore the coordinate
rotation state before Macro execution
```

```
N11 Restore Modes();                        // Restore all modes before Macro execution
```

4.2.2 Example 2

Application scenarios for **G54.4 P1** coordinate system rotation:

<table>
<tr><th>G54.4 P1(General Machining)</th><th>G54.4 P1+G68.2(Inclined Plane Machining)</th><th>G54.4 P1+G43.4/G43.5(Simultaneous Five-Axis Machining)</th></tr>
<tr>
<td>

Machining Macro

```
N1 G54 P1   // Specify the coordinate system
N2 G54.4 P1 // Enable coordinate rotation
N3 G53.1    // Align the coordinate system
N4 G43 H1   // Activate tool length compensation
N5 ......   // Start machining
```

</td>
<td>

Machining Macro

```
N1 G54 P1   // Specify the coordinate system
N2 G54.4 P1 // Enable coordinate rotation
N3 G68.2    // Activate inclined plane machining
N4 G53.1    // Align the coordinate system
N5 G43 H1   // Activate tool length compensation
N6 ......   // Start machining
```

</td>
<td>

Machining Macro

```
N1 G54 P1   // Specify the coordinate system
N2 G54.4 P1 // Enable coordinate rotation
N3 G43.4 H1 // Activate tool length compensation
N4 ......   // Start machining
```

</td>
</tr>
</table>

4.2.3 Example 3

When using **G54.4 P1** for coordinate system rotation, if the rotation values are modified during operation, tool length compensation must be reactivated:

重新启用刀长补偿 Macro

```
N1 G54 P1          // Specify the coordinate system
N2 G54.4 P1        // Enable coordinate rotation
N3 G53.1           // Align the coordinate system
N4 G43 H1          // Activate tool length compensation
N5 ......          // (Modify coordinate rotation values)
N6 G49             // Cancel tool length compensation
N7 G43 H1          // Reactivate tool length compensation
N8 ......          // Start machining
```

4.3 4.3 Precautions

4.3.1 Pr3229

Pr3229 Disable workpiece coordinate

The function of **Pr3229** is to disable the Workpiece Coordinate System feature. When Pr3229 = 1, the G54.4 command will not take effect.

	Pr3229 = 0	**Pr3229 = 1**
G54.4 = 0	X	X
G54.4 = 1	O	X

O: Workpiece Coordinate Rotation On
X: Workpiece Coordinate Rotation Off

Warning

If the Workpiece Coordinate Rotation is active, setting Pr3229 = 1 will disable the Workpiece Coordinate Rotation feature.

Default Activation State of Workpiece Coordinate Rotation

※ *Upon system startup or enabling the Workpiece Coordinate System function (Pr3229 = 1), if Pr3839 is set to 1, the Workpiece Coordinate Rotation function will be automatically enabled.*
For additional details, refer to Pr3839 Enabled Mode of Workpiece Coordinate System Rotation

5 5. Error and Compensation of 5-Axis Machine

This chapter will introduce how to measure and compensate the errors of 5-axis machine.

5.1 5.1 Measurememt and Compensation

Because there are two more rotary axis on the machine, the possibility that mechanism error happens shall increase.

Besides linear motions, the cause of the errors will also be more complicated during operation.

Whether these errors are compensated or not, the precision will be affected to varying degrees.

Moreover, since the motions of 5-axis machine are so complicated, the measurement also becomes a huge project.

5.1.1 Introduction of Error terms

There are two types of error for 5-axis machine, position error and component error.

Position errors occur due to the difference between ideal and actual position of each axis, it's classified as static error and the error value is a constant.

Component errors occur due to the difference between ideal and actual movement, it's classified as dynamic error and the error value is the function of the position.

According to ISO 230-1, the errors will be named with 3 characters, such as EAX or XOC, each of them has its own meaning, the definitions of position error and component error are also different.

Position Error:

EX:

AOY

1st character-A： the error direction is A axis.

2nd character-O： always be O, stands for Position Error.

3rd character-Y： the axis under consideration is Y axis.

Explanation：

AOY means Y axis has an angle error in A axis direction (around X axis).

Component Error:

EX:

EXY

1st character-E： always be E, stands for Component Error.

2nd character-X： the error direction is X axis.

3rd character-Y： the axis under consideration is Y axis.

Explanation：

EXY means Y axis has a straightness error in X axis direction.

A total of 43 error terms of 5-axis machine are listed below, and will be explained in the following section.

	Error Type	Error of Each Axis	Number of Axis	Total Errors
Linear Axis	Position Error	-	-	3
	Component Error	6	3	18
Rotary Axis	Position Error	5	2	10
	Component Error	6	2	12

Error of Linear Axis

The position error of linear axis is the squareness of the machine, as shown in Fig.55.

The ideal angles between XYZ axis should be 90 degrees, but errors might occur due to parts precision or assembling mistakes.

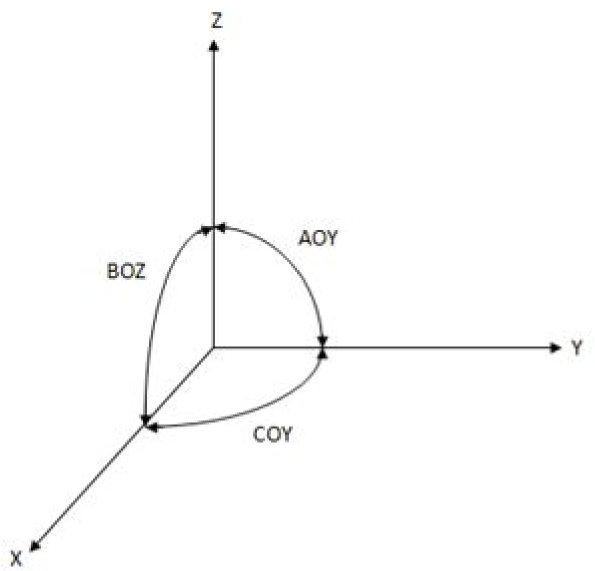


Fig.55

The component error of linear axis is the function of position, including translational deviation and rotational deviation.

For each linear axis, there are 3 error terms for both linear and rotary dimension respectively, thus there are total 18 error terms for 3 axis.

Fig.56 takes Y axis as an example.

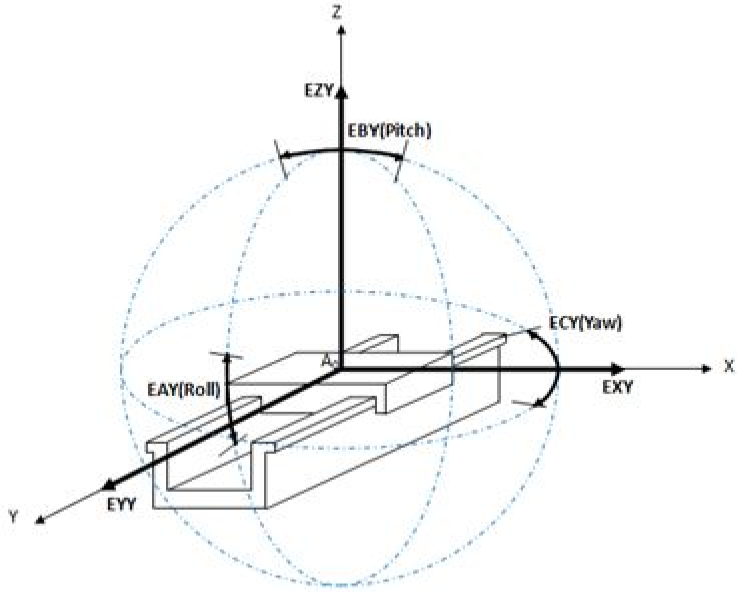


Fig.56

Error of Rotary Axis

The position error of rotary axis includes position deviation in the direction of the other 2 axis and angle deviation around all 3 axis.

Thus there are total 10 error terms for 2 rotary axis.

Fig.57 takes C axis as an example.

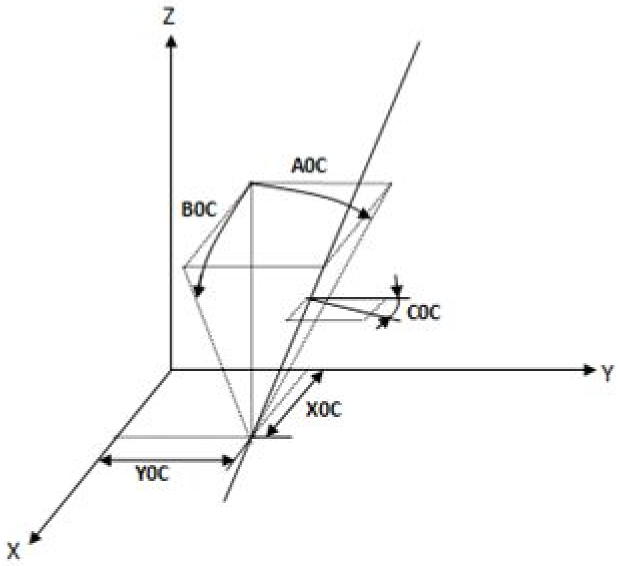


Fig.57

The component error of rotary axis is the function of the position of tool tip.

Therefore, when the tool is longer or the cutting area is far away from the rotary axis, the error varies with the tool length and the distance.

For each rotary axis, there are 3 error terms for both linear and rotary dimension respectively, thus there are total 12 error terms for 2 axis.

Fig.58 takes C axis as an example.

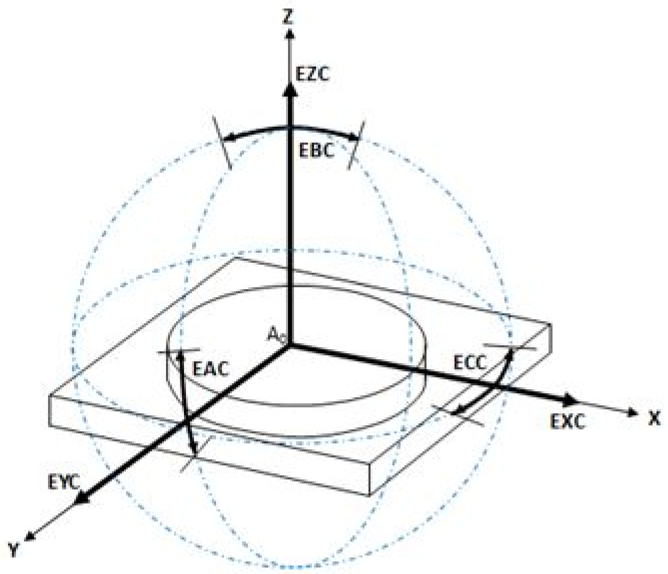


Fig.58

5.1.2 Related Parameters of Compensation for Syntec Controller

The definitions of errors for 5-axis machine are explained above and all 43 error terms are listed below.

15 of them (in red) are the errors which can be compensated by Syntec controller for now.

Linear Axis				**Rotary Axis (2 out of 3 axis)**			
Position Error	Component Error			Position Error		Component Error	
	X	Y	Z	B	C	B	C
AOY	EXX	EYY	EZZ	XOB	XOC	EXB	EXC
COY	EYX	EXY	EXZ	ZOB	YOC	EYB	EYC
BOZ	EZX	EZY	EYZ	AOB	AOC	EZB	EZC
	EAX	EAY	EAZ	BOB	BOC	EAB	EAC

	EBX	EBY	EBZ	COB	COC	EBB	EBC
	ECX	ECY	ECZ			ECB	ECC

EXX, EYY, EZZ, EBB, ECC can be sorted into pitch error, which can be compensated with the pitch compensation function of Syntec controllers directly.

The related parameters are Pr8001~Pr10000, please refers to the corresponding manual for further details.

XOB, ZOB, XOC, YOC are the position errors of the center of the rotary axis, need to be measured with instruments.

AOB, BOB, COB, AOC, BOC, COC are the angular errors of the rotary axis, can be compensated by Pr3015~Pr3020, but still need to be measured with instruments.

For now the parameters about the error compensation combine the errors and the mechanical dimensions.

For example, if the distance between 1st and 2nd axis is originally designed to be 150mm, but turns out to be 150.03mm after measuring, which means a 0.03mm error occurred.

With Syntec controller, it only needs to input 150.03, no need to input 150 and 0.03 respectively.

The table below shows all corresponding parameters, Pr3021~Pr3026 are for spindle type; Pr3031~Pr3036 are for table type; Pr3041~Pr3046 are for mix type.

These parameters are separated into XYZ components respectively.

No.	**Descriptions**	**Range**	**Unit**	**Default**	**Take Effect**
Spindle Type					
3021	1st x-component of Offset from tool holder to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3022	1st y-component of Offset from tool holder to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3023	1st z-component of Offset from tool holder to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3024	1st x-component of Offset from second rotation axis to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3025	1st y-component of Offset from second rotation axis to first rotation axis	[-999999999,999999999]	BLU	0	Reset

3026	1st z-component of Offset from second rotation axis to first rotation axis	[-999999999,999999999]	BLU	0	Reset
Table Type					
3031	1st x-component of Offset from first rotation axis to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3032	1st y-component of Offset from first rotation axis to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3033	1st z-component of Offset from first rotation axis to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3034	1st x-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3035	1st y-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3036	1st z-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
Mix Type					
3041	1st x-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3042	1st y-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3043	1st z-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3044	1st x-component of Offset from machine to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3045	1st y-component of Offset from machine to second rotation axis	[-999999999,999999999]	BLU	0	Reset
3046	1st z-component of Offset from machine to second rotation axis	[-999999999,999999999]	BLU	0	Reset

Spindle Type for 4-Axis Machine					
3041	1st x-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3042	1st y-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3043	1st z-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
Table Type for 4-Axis Machine					
3034	1st x-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3035	1st y-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
3036	1st z-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset

6 6. 3D Tool Radius Compensation

For machines equipped with multiple rotary axes capable of freely controlling the tool axis direction, this feature calculates the tool vector based on the rotary axis positions. It then computes the compensation vector on the plane perpendicular to the tool vector (compensation plane), enabling 3D tool radius compensation.

Supported Versions: Version **10.120.27** and above. This feature is available after enabling the software option **Option-61**.

6.1 6.1 Command Format

Type 1:

```
G41.2 ( or G42.2 ) X_Y_Z_A_B_C_ D_;
G40;
```

- **G41.2**: 3D tool radius compensation, offset to the **left**.
- **G42.2**: 3D tool radius compensation, offset to the **right**.
- **G40**: **Cancel** tool radius compensation.
- **X_Y_Z_**: Axis movement commands, tool tip coordinates.
- **A_B_C_**: Rotary axis movement commands.
- **D**: Specifies the tool diameter or tool number for compensation.

Type 2:

```
G41.6 ( or G42.6 ) X_Y_Z_I_J_K_D_:
G40;
```

- **G41.6**: 3D tool radius compensation, offset to the **left**.
- **G42.6**: 3D tool radius compensation, offset to the **right**.
- **G40**: **Cancel** tool radius compensation.
- **X_Y_Z_**: Axis movement commands, tool tip coordinates.
- **I_J_K_**: Tool vector.
- **D**: Specifies the tool diameter or tool number for compensation.

6.2 6.2 Precaution

1. Supported G-codes for 3D Tool Diameter Compensation (Cutting/Positioning/Interpolation):
    a. **G00**: Rapid linear positioning
    b. **G01**: Linear interpolation
2. Tool Radius Compensation Formula: Tool Radius = Tool Diameter + Wear Compensation
    a. Overcut detection is currently not supported.
3. **XYZABC** movement commands will use either **G00** or **G01**, depending on the mode of the previous block.。
4. For **G41.6 (Type 2)**, issuing **A_B_C_** rotary axis movement commands will trigger an alarm: **[COR-158: G43.5 mode does not support the first or second rotary axis command].**
5. If the parameters **I, J, K** are partially omitted, the omitted parameters are treated as **0**. If all are omitted, the tool direction is assumed to be the same as the previous block.

6. The tool direction cannot be a zero vector. If **I0. J0. K0.** is specified, an alarm will be triggered: **[COR-159: Invalid tool vector].**
7. Neither **Type 1** nor **Type 2** tool diameter compensation currently supports **Pr3057 - Tool attitude reference coordinate system**
8. **G40** must be issued to cancel tool compensation before issuing **G49** to disable RTCP.

6.3 6.3 Limitations

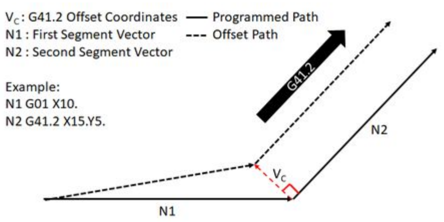
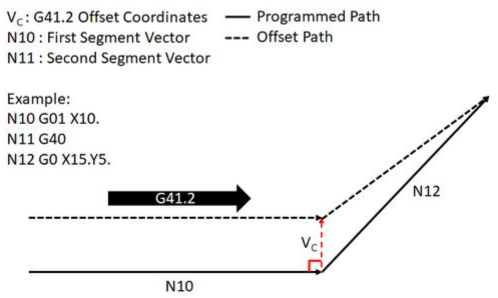
- Handwheel simulation can be used between program blocks with 3D tool radius compensation enabled, but reversing the handwheel has no effect.
- Changing the compensation mode while tool radius compensation is enabled is not allowed.
    - If **G41/G42** tool diameter compensation is enabled, using 3D tool diameter compensation will trigger the **COR-605** alarm
    - If **G41.2/G42.2** 3D tool diameter compensation is enabled, using tool diameter compensation will trigger the **COR-605** alarm.
- Before enabling 3D tool diameter compensation, **RTCP tool tip control** must first be enabled
    - If tool tip control is not enabled during tool diameter compensation, the **COR-601** alarm will be triggered: *"Tool diameter compensation cannot be used before RTCP is enabled."*
- Disabling tool tip control (**G49**) during tool diameter compensation is not allowed
    - If tool tip control is disabled during compensation, the **COR-603** alarm will be triggered: *"Tool tip control cannot be disabled during 3D tool diameter compensation."*
- Changing the work plane (**G17~G19**) during tool diameter compensation is not allowed.
    - If a work plane change is attempted during compensation, the **COR-604** alarm will be triggered: "Tool diameter compensation cannot change during operation."
- Changing coordinates during tool radius compensation is not allowed.
    - Using the following commands to change coordinates during compensation will trigger the **COR-603** alarm:
        i. **G52**: Local coordinate setting.
        ii. **G54-G59.9**: Work coordinate setting.
        iii. **G54**: Coordinate rotation.
        iv. **G68**: Coordinate rotation.
        v. **G68.2/G68.3**: Inclined plane machining.
            1. The inclined plane machining tool alignment commands **G53.1/G53.6** are not available because inclined plane machining is blocked.
        vi. **G92/G92.1**: Absolute zero coordinate setting.
- These commands are not compatible with 3D tool radius compensation
    - During 3D tool radius compensation, using the following commands will trigger the alarm: **"COR-602 3D tool radius compensation is enabled, using unsupported commands."**
        i. Arc interpolation: **G02/G03/G02.1/G03.1/G02.2/G03.2**
        ii. Polar coordinate interpolation: **G12.1/G13.1**
        iii. Polar coordinate command: **G16**
        iv. Reference point return: **G28/G29/G30**
        v. High-speed positioning: **G28.1**
        vi. Skip functions: **G31/G31.10/G31.11**
        vii. Thread turning command: **G33**
        viii. Cutter compensation (tool radius) control: **G41.1/G42.1**
        ix. Machine coordinate positioning: **G53**
        x. Tapping mode: **G63**
        xi. Feed per revolution: **G95**
        xii. Constant surface speed control: **G96**
- The following commands, when enabled, prevent the use of 3D tool radius compensation:
    - Tangential Control: **G41.1/G42.1**

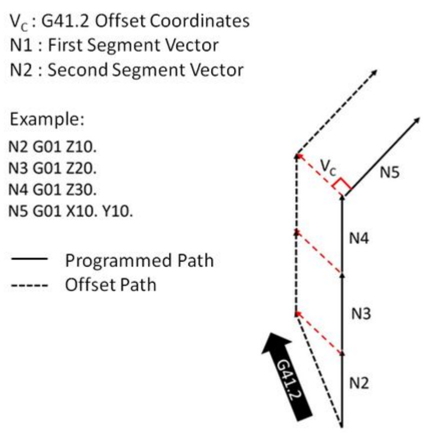
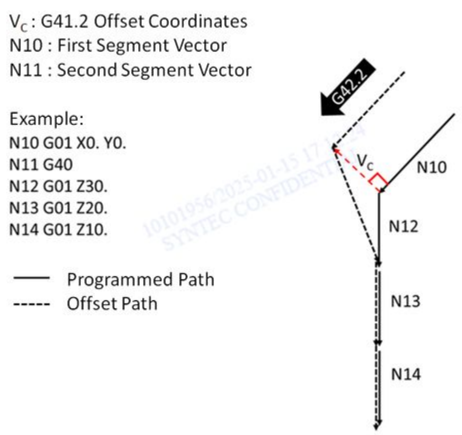
    - If 3D tool radius compensation is used while cutter compensation control is active, alarm **COR-182** will be triggered.
  - Tapping mode: **G63**
  - Feed per revolution: **G95**
  - Constant surface speed control: **G96**
- When coordinate rotation is enabled, 3D tool radius compensation cannot be used
  - If 3D tool radius compensation is used while the following coordinate transformations are active, the alarm **"COR-606 Coordinate rotation enabled, 3D tool radius compensation cannot be used"** will be triggered:
    1. Polar coordinate interpolation: **G12.1/G13.1**
    2. Polar coordinate command: **G16**
    3. Tool Alignment for Tilted Working Plane Machining: **G53.1/G53.6**
    4. Enabling coordinate rotation G54 under Pr3057 - Tool attitude reference coordinate system = 1
    5. Coordinate rotation: **G68**
    6. Tilted Working Plane Machining: **G68.2/G68.3**。
- If an unprocessable tool compensation amount occurs during the tool compensation process, the alarm **"COR-607 This block cannot compute 3D tool compensation value"** will be triggered.
  - When using the intersection method, the tool vector is coplanar with the path.
- If too many non-processable blocks occur during the tool compensation process
  - After enabling tool compensation, any non-processable blocks before the first move block are not problematic
  - Once the first move block occurs, if there are 8 or more non-processable blocks between the moving blocks, the alarm **"COR-609"** will be triggered.
  - Non-processable blocks include:
    - **G00/G01 blocks** that only specify movement of the A, B, C rotary axes.
    - **G00/G01 blocks** where the tool tip coordinates do not change from the previous block (i.e., the tool tip does not move).
    - **G00/G01 blocks** where the tool tip coordinates cause the tool tip to move in a direction parallel to the tool, meaning the block is used for tool parallel entry or exit from the workpiece.
    - G-codes other than **G00/G01**.
    - **M-codes**, **H-codes**, **T-codes**, etc.
    - **G-code macros**. (If a macro is used during the tool compensation process, the number of blocks occupied by the macro will be equal to the number of instructions expanded from the macro.)
- The following functions are prohibited when 3D tool radius compensation is enabled:
  - Indexing axis
    - If the indexing axis participates in five-axis linkage, the alarm **COR-369** will be triggered.
  - Tilting axis
    - If the tilting axis participates in five-axis linkage, the alarm **COR-171** will be triggered.
- The following functions are not recommended during the tool compensation process:
  - Various cycle functions **G73-G89**.
  - Various cycle functions **G135~G137.1**.
  - Commands in MDI mode after a pause.
  - Use of handwheel offset.
  - Use of breakpoint return function to return to a block in 3D tool compensation.
  - **Overlapping**.

6.4 6.4 Action Description

6.4.1 The actions of **tool engagement** and **tool withdrawal**

- Not affected by **Pr3815 Tool Radius Compensation Mode**
- Tool engagement/withdrawal does not check for collisions. When writing the processing program, users must be cautious of potential risks.

		Tool Engagement	**Tool Withdrawal**
Scenario 1	Description	• **The first move block (N2) following a tool compensation command (e.g., G41.2) is the tool-on block.** • The move block after the tool-on block is the first effective tool compensation block (N3). • The first effective tool compensation block (N3) has a compensation vector (Vc) at the starting point of the block, perpendicular to the direction of the block. • **The endpoint of the tool-on block (N2) will reach the starting position of the first effective tool compensation block (N3) with compensation applied.**	• **The first move block (N12) after disabling tool compensation (G40) is the tool-off block.** • The move block before the tool-off block is the last effective tool compensation block (N10). • The last effective tool compensation block (N10) has a compensation vector (Vc) at the endpoint of the block, perpendicular to the direction of the block. • **The starting point of the tool-off block (N12) will start from the compensated position at the endpoint of the last effective tool compensation block (N10).**
	Illustration		

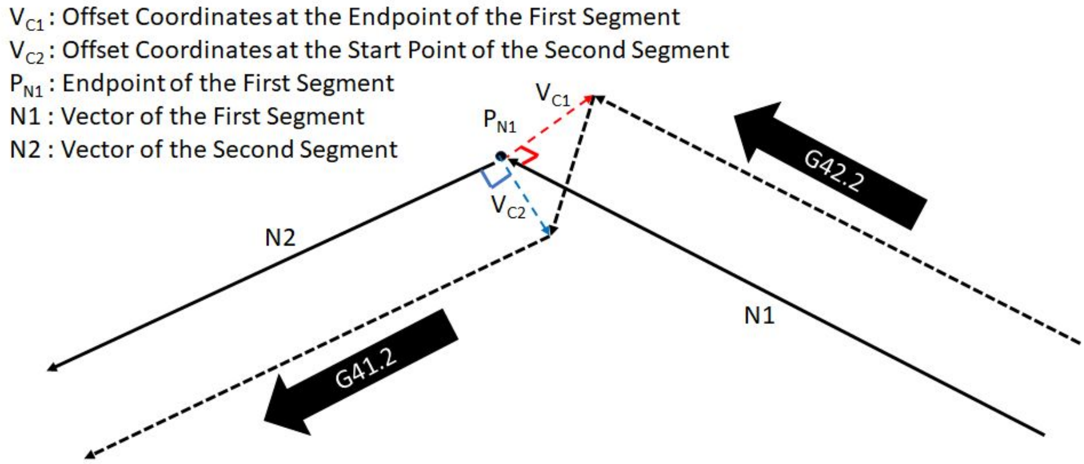
<table>
<tr><th></th><th></th><th>Tool Engagement</th><th>Tool Withdrawal</th></tr>
<tr><td rowspan="2">Scenario 2</td><td>Description</td><td>• The block that is parallel to the tool direction can be considered as the tool-on block. The tool-on timing is shown in the following diagram.</td><td>• The block that is parallel to the tool direction can be considered as the tool-off block. The tool-off timing is shown in the following diagram.</td></tr>
<tr><td>Illustration</td><td>
</td><td>
</td></tr>
</table>

6.4.2 Compensation Actions

Tool Compensation Behavior

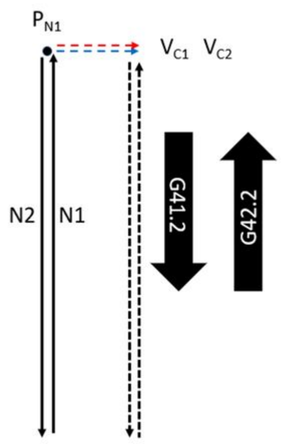
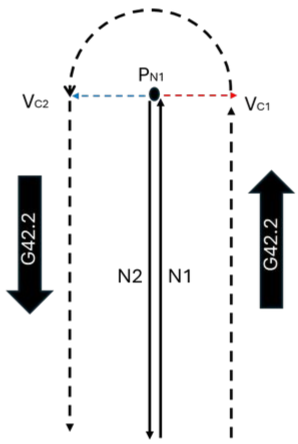
- For blocks with geometric axis movement and tool posture changes, the controller ensures that the cutting point (programming point) of the tool remains in contact with the block at all times.
- It is allowed to change the tool compensation side during the compensation process.
  - When changing the compensation direction, the blocks between the two modes are not considered as corners.
  - Action Method: Insert a block with the same insertion mode (G01/G02/G03) as the last block of the previous tool compensation mode. From the endpoint of the last block in the previous tool compensation mode, move to the starting point of the next tool compensation mode's first block by offsetting the coordinates.
  - No over-cut check is performed during the replacement process. (When the movement direction of the front and back blocks is opposite, over-cutting may occur at the reverse point due to the simultaneous presence of two compensation vector directions.)

Example: Corner



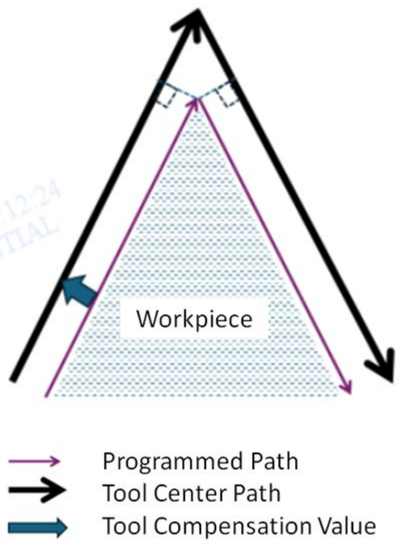
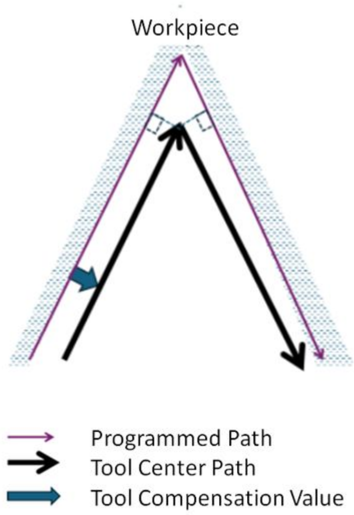

```
( G42.2方式 )
..
N1 G91 G01 X90. Z20. ;
N2 G41.2 X60. Z0. ;
..
```

- When the tool path is reversed compared to the previous program segment, the offset direction can be changed via G-code to match the path of the previous program segment.

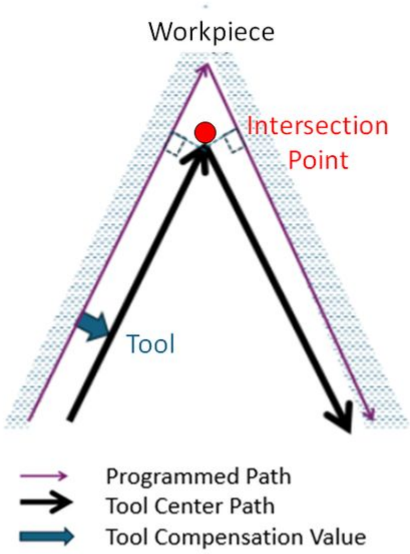
Example: Reversal	**The offset direction is changed via G-code**	**If the offset direction is not changed via G-code, an arc will be used for the connection**
V_{C1} : 1st Segment Endpoint Offset Coordinate V_{C2} : 2nd Segment Start Point Offset Coordinate P_{N1} : 1st Segment Endpoint N1 : 1st Segment Vector N2 : 2nd Segment Vector		

Corner Behavior

- Corner Classification

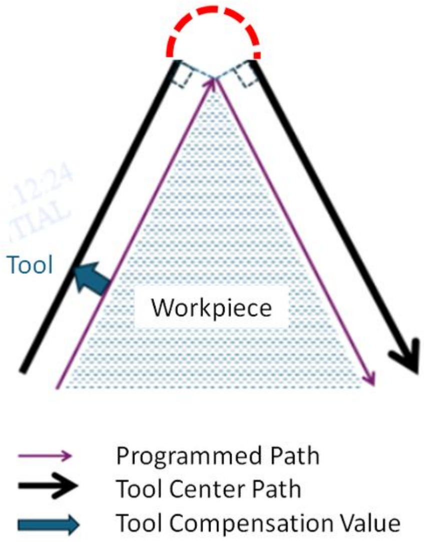
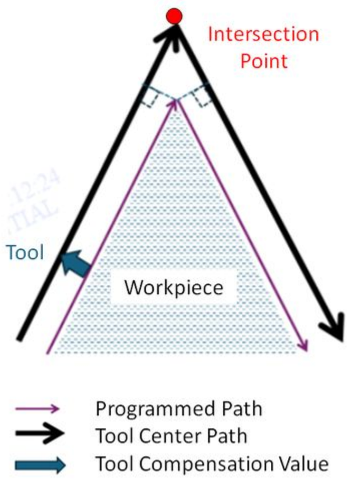
	Outer Corner	**Inner Corner**
Illustration		
Definition	The tool compensation offset path is outside the programmed path	The tool compensation offset path is inside the programmed path

- Inner corner path processing method：
    - Project the offset paths of the previous and next segments onto the compensation plane formed by the tool vector at the endpoint of the first segment to find the intersection.

Illustration


- Outer corner path processing method：
  - 提Two processing methods are provided for outer corners: Arc transition for outer corner, Extend the path to find the intersection for outer corner. The user can specify the method using G-code (see the table below).
  - If the method for outer corner transition is not specified, the default method is to extend the path to find the intersection.

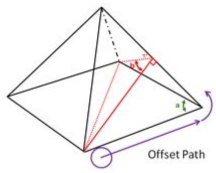
G Code	G450	G451
Definition	Outer corner arc transition	Outer corner with path extension to find intersection
Description	At the outer corner, an arc segment with a radius equal to the tool radius is inserted. This arc starts from the tool compensation position at the endpoint of the segment before the corner and rotates to the tool compensation position at the starting point of the new segment. The arc transition always ensures that the **cutting point remains at the corner (programming point)**.	At the outer corner, the offset paths (tool tip paths) of the previous and next segments are projected onto the tool compensation plane of the segment before the corner (an imaginary plane with the tool orientation as the normal vector). The offset path is extended to form an intersection.

G Code	G450	G451
Illustration		

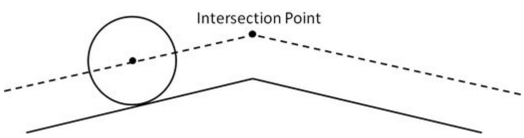
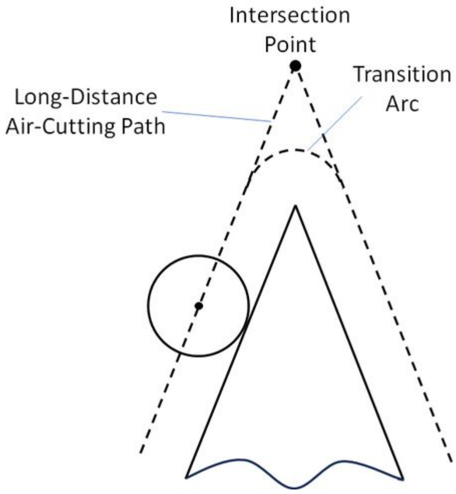
G Code	G450	G451

Sample Program 1	```//O100 (Sample Program1;
	N10 G55; // Select work coordinate system 5
	N20 G90 X0 Y0 Z300.0 B0 C0; // Move to the starting point (absolute coordinates)
	N30 G01 G43.4 H01 Z40.0 F500.0; // Activate RTCP (Rotation Tool Center Point), set tool offset, and feed rate
	N40 G41.2 D01; // Enable 3D tool compensation
	N50 G450; // Specify corner mode
	N60 X50.0 Y50.0 Z20.0 B30.0 C45.0; // Move to the position with specified coordinates and angles
	N70 X-50.0 C135.0;
	N80 X-100.0 Y-100.0 C225.0;
	N90 X100.0 C315.0;
	N100 G451; // Change corner mode
	N110 X50.0 Y50.0 C405.0;
	N120 X0 Y0 Z40.0 B0 C360.0; // Return to initial position
	N130 G40; // Disable 3D tool compensation
	N140 G49; // Disable RTCP (Rotation Tool Center Point)
	N150 G00 Z300.0; // Return to the starting point (rapid move)
	N160 M30; // End of program```	

- Angle Range Setting

Workpiece Angle Definition	
	• **Program Path**: The path specified in the NC machining file, which can be either manually written by the user or generated by CAD/CAM software. This path typically represents the cutting point trajectory of the tool. • **Offset Path**: The path obtained after applying the specified tool radius compensation. The offset path is equivalent to the tool tip path. It is derived from the offset of the program path by the tool radius. • In 3D space, when machining the side of a tapered four-corner pyramid with a side milling cutter, the workpiece angle is defined as the angle (b) formed by the machined surface, as shown in the diagram, rather than the angle (a) formed by the NC path.

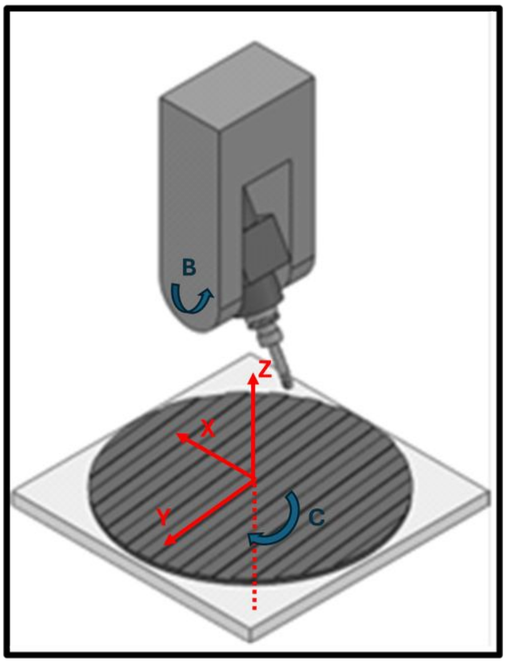
	Maximum Angle for Arc Path Planning	**Minimum Angle for Intersection Path Planning**
Description	When using the arc transition method, if the workpiece angle is too large, the workpiece profile becomes overly flat, and the transition arc and the extended intersection point become very close. In such cases, the advantage of using the arc transition method becomes less significant. Therefore, by adjusting parameters to set a threshold, the system can automatically switch to the intersection method when the workpiece angle exceeds the threshold, avoiding the slowdown caused by the arc transition and helping to reduce processing time.	When using the path extension to find the intersection method, if the workpiece angle is too small, the workpiece profile becomes too sharp, potentially leading to a long empty cutting path extending to the intersection point. Therefore, by adjusting parameters to set a threshold, the system can automatically switch to a transition arc when the workpiece angle is smaller than the threshold, helping to reduce processing time.

<table>
<tr><th></th><th>Maximum Angle for Arc Path Planning</th><th>Minimum Angle for Intersection Path Planning</th></tr>
<tr><td>Illustration</td><td>
</td><td>
</td></tr>
<tr><td>Parameter Specifications</td><td><table>
<tr><th>Parameter Number</th><th>Description</th><th>Range</th><th>Unit</th><th>Initial Value</th><th>When Changes Take Effect</th></tr>
<tr><td>3992</td><td>The maximum angle for using arc path planning for outer corners</td><td>[0~1500]</td><td>0.1deg</td><td>1500</td><td>Reset</td></tr>
</table></td><td><table>
<tr><th>Parameter Number</th><th>Description</th><th>Range</th><th>Unit</th><th>Initial Value</th><th>When Changes Take Effect</th></tr>
<tr><td>Pr3991</td><td>The minimum angle for using intersection path planning for outer corners</td><td>[0~1500]</td><td>0.1deg</td><td>300</td><td>Reset</td></tr>
</table></td></tr>
</table>

<table>
<tr><th></th><th>Maximum Angle for Arc Path Planning</th><th>Minimum Angle for Intersection Path Planning</th></tr>
<tr><td>Application</td><td>When using the arc transition method, the following conditions apply:
<table><tr><th>Workpiece angle > Pr3992</th><th>Workpiece angle < Pr3992</th></tr><tr><td>Intersection path extension</td><td>Arc transition</td></tr></table></td><td>When using the path extension to intersection method, the following conditions apply:
<table><tr><th>Workpiece angle > Pr3991</th><th>Workpiece angle < Pr3991</th></tr><tr><td>Intersection path extension</td><td>Arc transition</td></tr></table></td></tr>
</table>

- Note
  - G450 / G451 support arbitrary switching during the tool compensation process.
  - Regardless of whether it's an inner or outer corner, when the tool posture is not parallel to the normal vector of the corner vector plane, when the intersection is back-projected to the original segment, the intersection point of the second segment is not at the tool tip but at a certain distance along the tool axis. Depending on whether the projection point is above or below the tool tip, the tool will either be inserted or withdrawn along the tool axis in the next segment to compensate for this distance difference.

6.5 6.5 Program Example

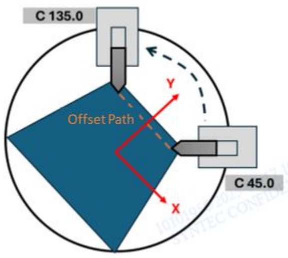
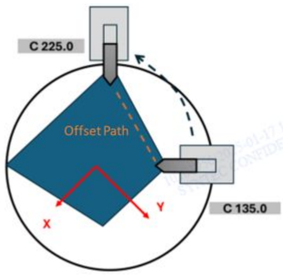
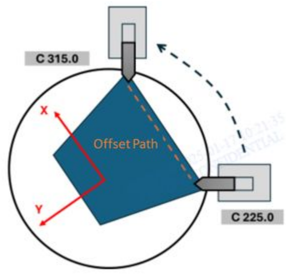
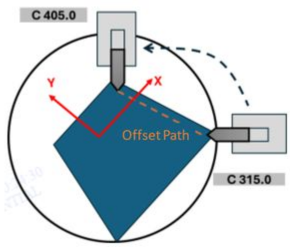
Example Machine - Mix Type	
	
Scenario	**Program**

Type 1	`// Sample Program 1`
	`N10 G55 ;`
	`N20 G90 X0 Y0 Z300.0 B0 C0 ;          // Move to the starting point`
	`N30 G01 G43.4 H01 Z40.0 F500.0 ;      // Enable RTCP`
	`N40 G41.2 D01;                         // Enable 3D tool compensation`
	`N50 X50.0 Y50.0 Z20.0 B30.0 C45.0 ;`
	`N60 X-50.0 C135.0 ;`
	`N70 X-100.0 Y-100.0 C225.0 ;`
	`N80 X100.0 C315.0 ;`
	`N90 X50.0 Y50.0 C405.0 ;`
	`N100 X0 Y0 Z40.0 B0 C360.0 ;`
	`N110 G40 ;                             // Disable 3D tool compensation`
	`N120 G49 ;                             // Disable RTCP`
	`N130 G00 Z300.0 ;                      // Return to the starting point`
	`N140 M30;`

Scenario	Program
Type 2	(see code below)

```
// Sample Program 2
N10 G55 ;
N20 G90 X0 Y0 Z300.0 B0 C0 ;        // Move to the starting point
N30 G01 G43.5 H01 Z40.0 F500.0 ;    // Enable RTCP
N40 G41.6 D01;                      // Enable 3D tool compensation
N50 X50.0 Y50.0 Z20.0 I35.355 J35.355 K86.603 ;
N60 X-50.0 I-35.355 J35.355 K86.603 ;
N70 X-100.0 Y-100.0 I-35.355 J-35.355 K86.603 ;
N80 X100.0 I35.355 J-35.355 K86.603 ;
N90 X50.0 Y50.0 I35.355 J35.355 K86.603 ;
N100 X0 Y0 Z40.0 K1.0 ;
N110 G40 ;                          // Disable 3D tool compensation
N120 G49 ;                          // Disable RTCP
N130 G00 Z300.0 ;                   // Return to the starting point
N140 M30;
```

6.5.1 Action Decomposition

N50 → N60	N60 → N70	N70 → N80	N80 → N90
			

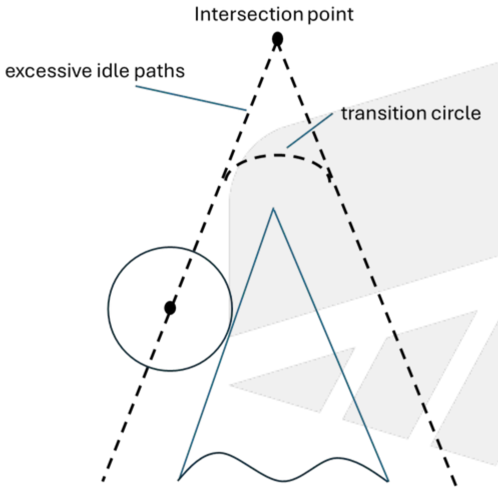
6.6 6.6 Related Parameter Description

6.6.1 Pr3991 The minimum angle of the Intersection of equidistant paths

Version	No	Descriptions	Range	Unit	Default	Effective
V10	Pr3991	The minimum angle of the Intersection of equidistant paths	[0~1500]	0.1deg	300	Reset

V12	P20131	N1~N4					

- **Supported Controller Software Version：10.120.27, and above**
- When using 3-dimensional cutter radius compensation (G41.2/G42.2, G41.6/G42.6) and the Intersection of equidistant paths (G451) for handling tool center path of an **outside corner**, if the **workpiece corner angle** becomes very pointed, G451 can result in excessive idle paths. Therefore, by adjusting the parameter setting of Pr3991, the system switches automatically from G451 (intersection) to G450 (transition circle) when the **workpiece corner angle** is smaller than the setting value.

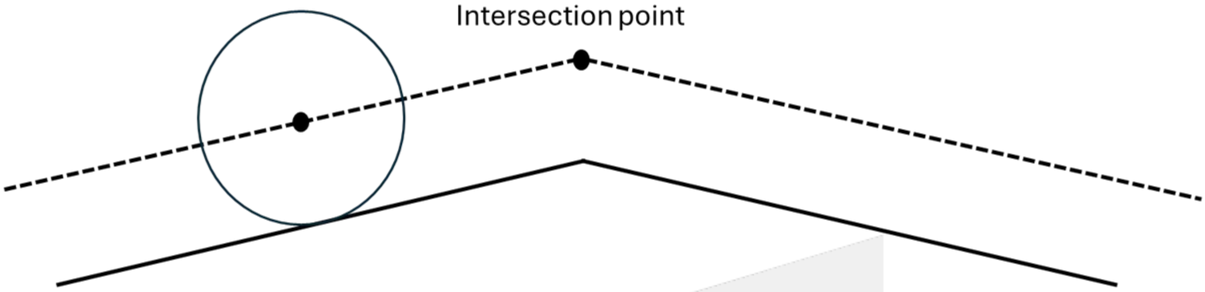


6.6.2 Pr3992 The maximum angle of the transition arc path

Version	No		Descriptions	Range	Unit	Default	Effective
V10	Pr3992		The maximum angle of the circular transition path	[0~1500]	0.1deg	1500	Reset
V12	P20132	N1~N4					

- **Supported Controller Software Version：10.120.27, and above**
- When using 3-dimensional cutter radius compensation (G41.2/G42.2, G41.6/G42.6) and the circular transition path method (G450) for handling tool center path of an **outside corner**, if the **workpiece corner angle** is very flat, the response with G450 (transition circle) and G451 (intersection) becomes increasingly similar. Therefore, by adjusting the parameter setting of Pr3992, the system switches automatically from G450 (transition circle,) to G451 (intersection) when the **workpiece corner angle** is larger than the setting

value.



6.7 6.7 Alarm Specifications

Alarm ID	COR-601 BGND-601	Alarm title	【RTCP is not properly enabled during the execution of 3D tool radius compensation】
Description	Executing 3-dimensional cutter compensation without correctly enabling RTCP.		
Possible Cause	1. Before using G41.2 or G42.2 functions, RTCP was not enabled with G43.4. 2. Before using G41.6 or G42.6 functions, RTCP was not enabled with G43.5. 3. During the execution of 3D tool radius compensation, switching RTCP mode from G43.4 to G43.5, or vice versa. 4. During the execution of 3D tool radius compensation, RTCP was turned off.		
Solution	During the execution of 3D tool radius compensation, use the corresponding RTCP commands correctly.		

Alarm ID	COR-602 BGND-602	Alarm title	【Unsupported command used when 3D tool radius compensation is enabled】
Description	Using unsupported commands when enabling 3D tool radius compensation.		

Possible Cause	1. The following commands are present between the blocks where 3D tool radius compensation is enabled: a. Arc interpolation (G02、G03、G02.4、G03.4)。 b. Polar Interpolation (G12.1、G13.1)。 c. polar coordinate command (G16)。 d. Reference point returning (G28、G29、G30)。 e. High-Speed Positioning(G28.1)。 f. Skip command(G31、G31.10、G31.11)。 g. Screw cutting(G33)。 h. Tangential control, left compensation(G41.1、G42.1)。 i. Mechanical coordinate orientation(G53)。 j. Tapping mode(G63)。 k. Feed per revolution(G95)。 l. Equal surface cutting speed(G96)。 2. Modifying the offset values corresponding to the tool number currently in use while 3D tool radius compensation is enabled.
Solution	1. For arc interpolation, use alternative commands such as G00 or G01. 2. Please remove unsupported commands described above, or disable 3D tool radius compensation before executing those commands. 3. Please wait until 3D tool radius compensation is turned off before modifying the offset values for the tool number in use.

Alarm ID	**COR-603** **BGND-603**	**Alarm title**	**【When 3D tool compensation is enabled, changing the coordinate system is not supported】**
Description	Changing the coordinate system while the 3D tool radius compensation function is enabled.		
Possible Cause	While 3D tool radius compensation is enabled, the following commands are used: 1. Local Coordinate System Setup (G52) 2. Workpiece Coordinate System Setup (G54-G59.9) 3. Enable/Disable Workpiece Coordinate Rotation Functionality (G54.4) 4. Tool orientation reference coordinate system setting (G43.4 Q1) 5. Coordinate Rotation (G68) 6. Tilted Working Plane Machining (G68.2、G68.3) 7. Absolute Zero Point Coordinate Setup (G92、G92.1)		
Solution	Before changing the coordinate system, use G40 to deactivate the 3D tool radius compensation function.		

Alarm ID	**COR-604** **BGND-604**	**Alarm title**	**【When 3D tool radius compensation is enabled, changing the machining plane is not allowed】**

Description	Changing the machining plane while 3D tool radius compensation is enabled will cause the controller to be unable to calculate the correct offset path.
Possible Cause	While 3D tool radius compensation is enabled, changing the machining plane with G17, G18, or G19.
Solution	Before changing the machining plane, first deactivate 3D tool radius compensation, and then reactivate it after the machining plane change is complete.

Alarm ID	**COR-605** **BGND-605**	**Alarm title**	**【Tool radius compensation mode conflict】**
Description	Changing the tool radius compensation mode is prohibited while 3D tool radius compensation is enabled.		
Possible Cause	1. Using G41.2 or G42.2 3D tool radius compensation commands while G41 or G42 cutter radius compensation is enabled. 2. Using G41 or G42 cutter radius compensation commands while G41.2 or G42.2 3D tool radius compensation is enabled.		
Solution	Before switching the cutter radius compensation mode, use G40 to cancel the current cutter radius compensation mode, then enable the new cutter radius compensation mode.		

Alarm ID	**COR-606** **BGND-606**	**Alarm title**	**【While coordinate transformation is enabled, the use of 3D tool radius compensation is prohibited】**
Description	While coordinate transformation is enabled, the use of 3D tool radius compensation is prohibited		
Possible Cause	When the following coordinate transformation functions are enabled, 3D tool radius compensation is activated: 1. Activate/Deactivate Polar Coordinate Interpolation (G12.1、G13.1) 2. Polar Coordinates Command Mode (G16) 3. Tool attitude reference coordinate system(Pr3057 - Tool attitude reference coordinate system = 1) 4. Coordinate Rotation(G68) 5. Tilted Working Plane Machining (G68.2、G68.3)		
Solution	1. Remove 3D tool radius compensation commands. 2. Disable those above unsupported coordinate settings for 3D tool radius compensation.		

<table>
<tr><td>Alarm ID</td><td>COR-607
BGND-607</td><td>Alarm title</td><td>【This block cannot calculate the 3D tool radius compensation amount】</td></tr>
<tr><td>Description</td><td colspan="3">This block cannot calculate the 3D tool radius compensation amount</td></tr>
<tr><td>Possible Cause</td><td colspan="3">1. If the tool orientation and path are coplanar, the 3D tool radius compensation amount cannot be calculated.</td></tr>
<tr><td>Solution</td><td colspan="3">1. Please modify the machining program so that the tool orientation is not coplanar with the path</td></tr>
<tr><td>Alarm ID</td><td>COR-608
BGND-608</td><td>Alarm title</td><td>【Incorrect tool number specified while 3D tool compensation is enabled】</td></tr>
<tr><td>Description</td><td colspan="3">After 3D tool radius compensation is enabled, an incorrect cutter radius number is specified.</td></tr>
<tr><td>Possible Cause</td><td colspan="3">After 3D tool radius compensation is enabled, the following situations occur:
1. Cutter number D is not specified.
2. After specifying cutter number D, another cutter number D is specified again.
3. The specified cutter number has invalid offset values or wear values set.</td></tr>
<tr><td>Solution</td><td colspan="3">1. Please specify cutter number D before the first movement block after G41.2, G42.2, G41.6, or G42.6.
2. Please ensure there is only one cutter number D specified within the 3D tool radius compensation activation period.
3. Please correctly set the compensation and wear values for the cutter, ensuring they comply with the following principles:
a. The compensation value cannot be less than 0.
b. If the compensation value is greater than 0, the sum of the compensation value and wear value cannot be less than 0.</td></tr>
<tr><td>Alarm ID</td><td>COR-609
BGND-609</td><td>Alarm title</td><td>【There are too many ineffective blocks between the compensable blocks】</td></tr>
<tr><td>Description</td><td colspan="3">The system cannot retain excessive invalid compensation blocks inserted between contour blocks when 3D tool radius compensation is activated.</td></tr>
</table>

Possible Cause	1. When 3D tool radius compensation is enabled, too many invalid compensation blocks are inserted between contour blocks, exceeding the system's capacity limit. 2. The following conditions define an "invalid compensation block": a. G00/G01 blocks that only specify movements for the A, B, or C rotational axes. b. G00/G01 blocks where the tool center point coordinates remain unchanged from the previous tool center point coordinates, meaning there is no movement of the tool center point. c. The G00/G01 block's tool center point coordinates cause the tool center point to move in a direction parallel to the tool axis, indicating that the block is used for the tool to enter or exit the workpiece machining area in a parallel manner d. Any G-codes other than G00/G01. e. M-codes, H-codes, T-codes, etc. f. G-code macro
Solution	Please reduce the number of invalid compensation blocks between contour blocks.

Alarm ID	**COR-610** **BGND-610**	**Alarm title**	**【Corner type change by tool rotation is disallowed】**
Description	In 3D tool radius compensation mode, tool rotation causes a change in the corner type.		
Possible Cause	• Variations in tool rotation angles between blocks can lead to the system misjudging the corner type, increasing the risk of cutting errors.		
Solution	Modify the tool rotation block to avoid misinterpretation of corner types.		

Alarm ID	**COR-611** **BGND-611**	**Alarm title**	**【The system does not support excessive invalid compensation blocks inserted between effective compensation blocks during 3D tool radius compensation mode.】**
Description	The system is unable to retain excess invalid compensation blocks inserted between effective compensation blocks when 3D tool radius compensation is activated		
Possible Cause	• When 3D tool radius compensation is enabled, the system cannot retain excessive invalid compensation blocks inserted between effective compensation blocks. • A block is considered an "invalid compensation block" if it meets the following conditions: a. The G00/G01 block only involves movement of the A, B, or C rotary axes. b. G00/G01 blocks where the tool center point coordinates remain unchanged from the previous tool center point coordinates, meaning there is no movement of the tool center point. c. The G00/G01 block's tool center point coordinates cause the tool center point to move in a direction parallel to the tool axis, indicating that the block is used for the tool to enter or exit the workpiece machining area in a parallel manner d. Auxiliary G codes are not supported by the 3D tool radius compensation. e. M-codes, H-codes, T-codes, etc.		

7 7. G54 Offset of Rotary Axis and 5-Axis Function

Effect on RTCP function due to G54 offset of rotary axis will be introduced in this chapter.

For 3-axis machines, G54 offset of rotary axis is a simple function to change the origin of program coordinate.

For 5-axis machines, G54 offset of rotary axis will affect the performance of five axis functions.

7.1 Rotary axis on spindle side

- In any circumstances, G54 offset of rotary axis on spindle side should be 0.
- If the offset of rotary axis on spindle side is necessary, please refer to Cross head 5-axis machine.

7.2 Rotary axis on table side

- RTCP function
    - **Pr3055 = 0 :**

        The calculation of tool tip position is based on the setting of mechanical chain and "the angle of rotary axis when RTCP is enabled".

        G54 offset of rotary axis will not affect the calculation of tool tip position.

    - **Pr3055 = 1 :**

        The calculation of tool tip position is based on the setting of mechanical chain and "G54 offset of rotary axis".

        User can apply the same NC program in different area by changing G54 offset.

        Please refers to the manual of Pr3055 for details.

- Tilted working plane functions (G68.2, G68.3, ... + G53.1, G53.3, G53.6, ...)
    - Pr3055 will not affect the calculation of tool tip position.
    - The calculation of tool tip position is based on the setting of mechanical chain and "G54 offset of rotary axis".
      User can apply the same NC program in different area by changing G54 offset.

8 8. Related Alarms

8.1 COR Alarm

Alarm ID	**COR–070** **BGND-070**	**Alarm title**	**Invalid G Code**
Description	Enter incorrect G code to controller.		
Reason	Program error.		
Solution	Enter the valid G-code.		

Alarm ID	**COR-100** **BGND-100**	**Alarm Title**	**Unsupported G code command or option software is not activated**
Description	Different controllers will have correspond G code, but not all G code can use.		
Reason	1. This controller type may not support this G code command. 2. This controller type will not support serial bus spindle (C-Type) to use lathe G21, G33, G34, G78 commands. 3. This controller type will not support serial bus spindle (A-Type) to use lathe G32, G73, G76, G92 commands. 4. This controller type can support this G code command, but the option software function has not been purchased, which makes the G code unusable. 5. Loader path and Wood auxiliary path only support part of G codes: G00, G01, G02, G03, G04, G04.1, G09, G10, G17, G18, G19, G22, G23, G31, G52, G53, G54, G55, G56, G57, G58, G59, G59.x, G90, G91, G92. 6. The setting of Pr3802 is incorrect. This controller type does not support the G62 command. 7. Synchronized positioning axis only support part of G codes: G00, G53		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Wood auxiliary path to do process operation. 6. Set Pr3802 to 0. 7. Do not use synchronized positioning axis to do process operation.		

Alarm ID	**COR-107** **BGND-107**	**Alarm Title**	**G5.1/G05 command format error**
Description	The G5.1 and G05 commands are in the wrong format.		
Possible Cause	1. The format of the G5.1 path smoothing command in the NC program is incorrect. 2. The G05 high-precision cutting mode command format in the NC program is incorrect.		

Solution	Confirm the following command formats are correct not have these error:
	1. G5.1
	  a. Q argument: None, more than 2, or less than 0.
	  b. E argument: None or less than 0.
	2. G05
	  a. System issue alarm when using G05 in following cases for each version:
	    i. G05 P argument is not 10000 nor 0.
	    ii. G05 E argument is not positive.
	  b. Activate command G05 P10000 X0 Y0 Z0 α_ β_ in 10.116.36 or above versions:
	    i. More than 5 axial directions are assigned.
	    ii. The geometry axis argument not 0.
	    iii. The rotary axis argument is configured to 0.
	    iv. The axial direction of geometry axis is configured but this of rotary axis is not.
	    v. The axis of the rotation axis is not set when the axis of the geometry axis is not set.
	    vi. More than 2 axial directions of rotation axes are configured.
	    vii. Any axial arguments is negative.
	  c. In the version before 10.116.16B, there is the 4th axis command in addition to the block movement commands of X, Y, or Z axes after G05 is executed.		

Alarm ID	**COR-118** **BGND-118**	**Alarm Title**	**Prohibit G53 commands in tool tip control mode**
Description	G53 command cannot be used in the tool point control mode.		
Possible Cause	1. The NC programming error. 2. The machine type is the tool point control mode.		

Alarm ID	COR-118 BGND-118	Alarm Title	Prohibit G53 commands in tool tip control mode
Solution	1. Please check the NC program, make sure that the G53 command is not within the validity range of G43.4 or G43.5. 2. Please check the NC program, make sure that the G53 command is not within the validity of G12.1. 3. If the machine configuration used is the tool point control mode, the G53 command cannot be used.		

Alarm ID	COR-140 BGND-140	Alarm Title	Invalid high-precision contour control mode using
Description	1. G05 command (high-precision contour control mode) is used in the RTCP/STCP mode. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		
Possible Cause	1. G05 command (high-precision contour control mode) is used in the RTCP/STCP mode. 2. When the high-precision contour control mode is enabled during processing, use single block stop C40. 3. When the STCP mode is enabled during processing, use single block stop C40.		

Solution	1. Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-precision contour control mode.
	If (1), remove the G05 command in the RTCP/STCP mode.
	If (2), turn off the RTCP/STCP mode before turning on the G05 high-precision contour control mode.
	2. When the high-precision contour control mode is enabled during processing, do not use the single block stop C40 at the same time.
	3. When the STCP mode is enabled during processing, do not use the single block stop C40 at the same time.		

Alarm ID	**COR-141** **BGND-141**	**Alarm Title**	**G68.3 command format error**

Description	[command format]
	G68.3 X_ Y_ Z_ R_; // The origin and z-axis rotation angle in the characteristic coordinate system.
	G68.3 P1 X_ Y_ Z_; // The origin of the characteristic coordinate system, and the coordinate system is determined with the tool rotation angle.		

Possible Cause	G68.3 command format, X, Y and Z are all exist or non-exist at the same time.		
Solution	Check if G68.3 command format is correct.		

Alarm ID	**COR-151** **BGND-151**	**Alarm Title**	**1^{st} rotation axis entering illegal range**
Description	1^{st} rotation axis entering illegal range.		
Possible Cause	1. Pr3007, Pr3009, or Pr3010 configuration error. 2. The angle of 1^{st} rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3009 and Pr3010 are configured correctly. The determination of such two configurations is related to Pr3007. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

Alarm ID	**COR-152** **BGND-152**	**Alarm Title**	**2^{nd} rotation axis entering illegal range**
Description	2^{nd} rotation axis entering illegal range		
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2^{nd} rotation axis is incorrect in the executed 5-axis NC program.		
Solution	1. Check if Pr3011 and Pr3012 are configured correctly. The determination of such two configurations is related to Pr3008. In case of the alarm, please re-confirm these 3 configurations. 2. Check the NC program.		

Alarm ID	**COR-153** **BGND-153**	**Alarm Title**	**Tool direction unknown**
Description	Tool direction unknown.		
Possible Cause	5-axis configurations and machine mechanism is incompatible.		
Solution	The tool cannot reach the destination. It may be caused by the incompatible 5-axis configurations and machine mechanism. Please check all 5-axis configurations.		

Alarm ID	**COR-154** **BGND-154**	**Alarm Title**	**No 5-axis function**
Description	No 5-axis function.		
Possible Cause	Pr3001 is not configured when executing G53.1 tool alignment command.		
Solution	Check if Pr3001 is configured to 0. If yes, configure the other non-zero values based on the 5-axis mechanism type and reboot.		

Alarm ID	**COR-155** **BGND-155**	**Alarm Title**	**5-axis tool direction error**
Description	5-axis tool direction error.		
Possible Cause	5-axis tool direction (Pr3002) or the 1^{st} and 2^{nd} rotation axis (Pr3005 and Pr3006) configuration error.		
Solution	Check if the Pr3002 is configured correctly, or if the Pr3005 or Pr3006 is configured correctly. The alarm will be triggered in case the 2^{nd} rotation axis is parallel to the Spindle in the Spindle type, or the 1^{st} rotation axis is parallel to the Spindle in the workbench type.		

Alarm ID	**COR-156** **BGND-156**	**Alarm Title**	**5-axis axial direction error**
Description	5-axis axial direction error.		
Possible Cause	Incorrect configurations are mapped to the axial direction parameters of 5 axis.		

Alarm ID	**COR-156** **BGND-156**	**Alarm Title**	**5-axis axial direction error**
Solution	Check if each axial direction is configured completely (Pr21~), if Pr3005, Pr3006, Pr3007 and Pr3008 is configured correctly, or if the axis name (Pr321~) is mapped to Pr3005 and Pr3006.		

Alarm ID	**COR-157** **BGND-157**	**Alarm Title**	**Incompatible direction of 5-axis tool direction and this of rotation axis**
Description	Incompatible direction of 5-axis tool direction and this of rotation axis.		
Possible Cause	1. The 2nd axial direction and the tool direction are the same in the Spindle type. 2. The 1st axial direction and the tool direction are the same in the table type. 3. The 1st axial direction and the tool direction are the same in the hybrid type.		
Solution	Check if the tool direction and the rotation axial direction are the same depended on the used 5-axis mechanism type.		

P.S. Valid version of COR-157 : before 10.118.41M, 10.118.47 (included).

Alarm ID	**COR-158** **BGND-158**	**Alarm Title**	**Prohibit the 1st and 2nd rotary axis commands in the G43.5 mode**
Description	Since the G43.5 mode specifies the tool attitude based on the tool vector I, J and K, it shall not be executed for the 1st and 2nd rotation axis commands which can also specify the tool attitude.		
Possible Cause	Programming error.		
Solution	Check the NC program to ensure the movement commands of the 1st and 2nd rotation axis are over the valid range in the G43.5 mode.		

Alarm ID	**COR-159** **BGND-159**	**Alarm Title**	**Illegal tool vector**
Description	In NC program, a movement block assigns an incorrect tool vector.		
Possible Cause	Programming error. i.e. Execute G01 X_ Y_ Z_ I0 J0 K0 in the G43.5 mode, and the I0 J0 K0 refers to the 0 vector, 0 vector is illegal.		

Alarm ID	COR-159 BGND-159	Alarm Title	Illegal tool vector
Solution	Check the NC program to ensure the assigned tool vector is correct in the block indicated by the alarm line number		

Alarm ID	COR-160 BGND-160	Alarm Title	5-axis mechanism chain switched when the 5-axis function is ON
Description	In NC program, the command G10 L5000 [P_] is executed to switch the 5-axis mechanism when turning on the 5-axis function (RTCP or the tool vector alignment on the characteristic coordinate system is completed).		
Possible Cause	Programming error. i.e. Execute G10 L5000 [P_] in the G43.4 mode.		
Solution	Check the NC program to ensure the 5-axis function is not turned on in the block indicated by the alarm line number.		

Alarm ID	COR-161 BGND-161	Alarm Title	Selected 5-axis mechanism chain is not ON
Description	In NC program, the 5-axis mechanism chain parameters assigned by the command G10 L5000 [P_] is not ON.		
Possible Cause	Programming error. i.e. The assigned 5-axis mechanism chain is not configured correctly when executing a command G10 L5000 [P_].		
Solution	Check the 5-axis mechanism chain parameters to ensure the assigned 5-axis mechanism chain function is turned on correctly: 1. 1st set: Pr3001 2. 2nd set: Pr3101 3. 3rd set: Pr5501 4. 4th set: Pr5601		

Alarm ID	COR-162 BGND-162	Alarm Title	4-axis RTCP configuration error
Description	The controller will issue the alarm in case the 4-axis RTCP configuration is incorrect.		

Alarm ID	**COR-162** **BGND-162**	**Alarm Title**	**4-axis RTCP configuration error**
Possible Cause	The 5-axis mechanism parameters, Pr3001, Pr3101, Pr5501 and Pr5601 are configured to 1~3 in case the specific 4-axis tool tip control function (option-29) is turned on but the tool tip control function (option-12) is not.		
Solution	1. Configure the 5-axis mechanism parameters to 4 or 5 correctly. 2. Turn on the tool tip control function (option-12).		

Alarm ID	**COR-163** **BGND-163**	**Alarm Title**	**Multi-kinematic chain command Q Argument setting error.**
Description	Command G10 L5000P_ Q_, Q argument range error.		
Possible Cause	Command G10 L5000P_ Q_, Q argument range error.		
Solution	While using G10 L5000P_ Q_, check Q argument to be within 0~4, and is a integer.		

Alarm ID	**COR-164** **BGND-164**	**Alarm Title**	**Multi-kinematic chain command related 5-Axis mechanism setting error.**
Description	Command G10 L5000P_ Q_ specified the 5-Axis kinematic chain, and the 5-Axis mechanism parameter setting error.		
Possible Cause	While executing G10 L5000P_ Q_, Q argument is given, but the 5-Axis mechanism parameter of the designated 5-Axis kinematic-chain is not a spindle-type 5-Axis machine.		
Solution	Please check the designated 5-Axis kinematic-chain. The 5-Axis mechanism configuration parameter must be a spindle-type 5-Axis machine. 1. The first group : Pr3001 is 1. 2. The second group : Pr3101 is 1. 3. The third group : Pr5501 is 1. 4. The fourth group : Pr5601 is 1.		

Alarm ID	**COR-165** **BGND-165**	**Alarm Title**	**Multi-kinematic chain command not illegal.**
Description	Command G10 L5000P_ Q_ is used for switching 5-Axis kinematic chain, and only provides partial 5-Axis mechanism function command.		

Alarm ID	**COR-165** **BGND-165**	**Alarm Title**	**Multi-kinematic chain command not illegal.**
Possible Cause	G10 L5000 P_ Q_ command, the Q argument is set to 2~4 (not the first group of sub-kinematic chain), and only supports the following 5-Axis machine function command. 1. RTCP: G43.4. 2. RTCP: G43.5. 3. Tilted working plane : G68.2 + Tool alignment functions. 4. Tilted working plane : G68.3. Notice: Tool alignment functions include G53.1, G53.3, G53.6, ...		
Solution	When using the G10 L5000 P_ Q_ command to switch multi-kinematic chains, please use the supported 5-Axis machine function command.		

Alarm ID	**COR-166** **BGND-166**	**Alarm Title**	**Characteristic Coordinate System Option not supported**
Description	Option13 (Characteristic Coordinate System Option) was not active, therefore the CNC couldn't execute relevant commands.		
Possible Cause	One or more commands below were given while the Option13 (Characteristic Coordinate System Option) was inactive: 1. G68.2, G68.3 2. G53.1, G53.3, G53.6 3. Other commands that are relevant to Option13 (Characteristic Coordinate System Option)		
Solution	1. Activate Option13 (Characteristic Coordinate System Option) 2. Avoid using the commands listed above		

Alarm ID	**COR-305** **BGND-305**	**Alarm title**	**【Relative position input method is forbidden in current mode】**
Description	The mode in which the system is currently running cannot be used with the G91 incremental command.		
Possible Cause	G43.5 cannot be used with the G91 incremental command function: G43.5 determines the tool attitude through the tool vectors I, J, and K. The tool attitude is expressed only in absolute quantities.		

Alarm ID	COR-305 BGND-305	Alarm title	【Relative position input method is forbidden in current mode】
Solution	Check the NC program to confirm that G91 was not executed in G43.5 mode and G43.5 was not executed in G91 mode		

8.2 OP Alarm

Alarm ID	OP-032	Alarm Title	Mechanism type configuration conflicted
Description	Pr3201 Machine Type setting conflicted.		
Possible Cause	Currently only the following mechanism type support five axis function; 1. Pr3201 sets as 0, close lathe feature. (Use general milling interface) 2. Pr3201 sets as 1, Lathe Habit Type C. 3. Pr3201 sets as 2, Lathe Habit Type A. 4. Pr3201 sets as 3, Lathe Habit Type B. Therefore, open both five axis (Pr3001, Pr3101, Pr5501, Pr5601) and other non-lathe/mill machine tool feature's mechanism type (Pr3201) at the same time, and alarm will be issued to inform user. For example: 1. Activate first group of five axis function (Pr3001 isn't 0. First path default to use first five axis kinematic chain) and first path is not lath/mill machine tool attribute. (Pr3201 in first path is not 0~3) 2. Activate second group of five axis function (Pr3101 isn't 0. Second path default use second five axis kinematic chain) and second path is not lath/mill machine tool attribute. (Pr3201 in second path is not 0~3) Besides, when activate option software function Option 29 (Four axis dedicated Rotate Tool Center Point function (4AXRTCP)), however, and option software function Option 12 (Rotate Tool Center Point (RTCP)) and Option 13 (Feature coordinate function) are not activated, but sets five axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 as 1~3, and this alarm will be issued.		

Alarm ID	OP-032	Alarm Title	Mechanism type configuration conflicted
Solution	Only the milling machine supports the 5-axis models. Configure Pr3201 to 0~3, or configure the mapped 5-axis function parameters (Pr3001, Pr3101, Pr5501 and Pr5601) to 0. [Note] In the versions after the 10.116.54G and 10.118.0D, the lathe can enable RTCP and is limited to 200TB-5. 1. Only lathe/mill machine tool supports five axis machine type. Please sets Pr3201 as 0~3, or sets corresponding five axis function parameter as 0. (Pr3001, Pr3101, Pr5501, Pr5601) 2. Active RTCP for Lathe 200TB-5. Supported version: 10.116.54G, 10.118.0D or above versions. 3. Please sets five axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 as 4 or 5, or please open software option function Option 12 (Rotate Tool Center Point (RTCP)) and Option 13 (Feature coordinate function).		

8.3 Macro Alarm

ID	Description	Solution
406	G53.6 needs to be enabled in G40 mode.	Make sure it is in G40 mode before executing G53.6.
407	The selected tool number of G53.6 cannot be 0.	Make sure that H argument exists after G53.6 and be a non-zero number; if not, the tool length compensation should be enabled with non-zero tool number.

9 9. Q&A

1. Why applying positive tool length on 5-axis machines? What is positive tool length?
    - Ans: The definition of tool length compensation is different for 5-axis and 3-axis machines. Tool length compensation for 3-axis machines is often used to deal with the coordinate offset between machine coordinate and workpiece coordinate, it's so called negative tool length because it's normally a negative value. For 5-axis machines, we can't only consider about the movements in XYZ directions during cutting since the rotary axis are also involved, so we need to notice the posture and position of the tool to avoid collision, thus the actual tool length must be provided to the controller. It's always a positive value for actual tool length, thus it's called positive tool length.
2. How to set G54 on 5-axis machines?
    - Ans: It's the same to set X, Y offset of G54 for 5-axis and 3-axis machines, but there is a little different for Z. The actual tool length should be deducted to obtain Z offset of G54 on 5-axis machines. In other words, we measure the tool length with tool tip for 3-axis machines but with spindle nose for 5-axis machines.
3. How to identify the directionality of the offset between rotary axis?
    - Ans: Find out the starting and end point of the offset vector according to the parameter definitions, then use the direction of XYZ to determine the direction of the offset and complete the parameter setting.
4. What is the reasonable resolution for rotary axis?
    - Ans: DD motor or servo motor with gear box might be used for the rotary axis, and there is no rule for the resolution, only the positioning precision is required. The angular error will be enlarged when workpiece is far from the rotation center, for this case the resolution should be increased.
5. How to set the axis type of the rotary axis?
    - Ans: No matter the command is positive or negative, it's alright to set Pr201~ as 1 or 2, the difference is the way to deal with the sign of the commands. When set to 1, the sign will be converted into a corresponding angle between 0~360 degrees, and the controller will automatically use the shortest path to move to the target angle; when set to 2, the sign will also be converted into a corresponding angle, but the positive sign means rotating in positive direction, and the negative sign means the opposite. For special needs, axis type 3, 4, 5 is applicable or it's possible to develop new types. The details of the axis type can be found in parameter manuals.
6. How to check if RTCP or Feature Coordinate is enabled?
    - Ans: F4 Run =>F4 Parameter Set, the state of G43.4 or G49, G68.2 or G69 will be shown on the screen. (For 200MA-5 only)
7. How to execute the backlash compensation of the rotary axis?
    - Ans: To measure or calculate the backlash angle and input to Pr1241~. For example, the backlash value is 0.5 degree, then input 500. Remember to set Pr1221~ to 1.
8. What is static error? How to deal with it?
    - Ans: When RTCP function is enabled, the controller calculates the coordinates according to the values of the related parameters, therefore, the correctness of the parameters related with the mechanism chain will affect the accuracy of theoretical coordinate and machine position (Pr3021~Pr3046). In other words, incorrect parameters will make the tool tip position calculated unable to coincide with the actual tip position, and the deviations are determined by the correctness of the parameters. These errors occur even when the rotary axis is fixed, thus it's called static error.

Normally we compare the tool tip position when the rotary axis is at 0 degree with the position after rotating to a certain angle, when error occurs, we adjust the corresponding parameter to improve the error till it's minimized to a certain range. Take the spindle type with CB axis as example, when B axis is at 0 & 45 degree, the program coordinates of Z axis should be the same, and the difference in machine coordinate should be a theoretical value, which relates to the tool length and angle. When the measured value is different from the theoretical value, the involved factors including tool length, Pr3013 and Pr3021 might be wrong, and need to do some tests to clarify the cause of the problem.

9. What is dynamic error? How to deal with it?

- Ans: In comparison to the static errors, dynamic errors occur during the rotation of the rotary axis, at this time 4 or 5 axis are moving simultaneously. Common reason is the poor compatibility between linear and rotary axis, thus result in the servo lag problem. For the situation, adjust the servo gain value (Pr181~) according to the error or enable SPA function (Pr3808) to improve it.