5-Axis Function Application Manual.

匯出日期：2021-08-24
修改日期：2020-09-25

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:

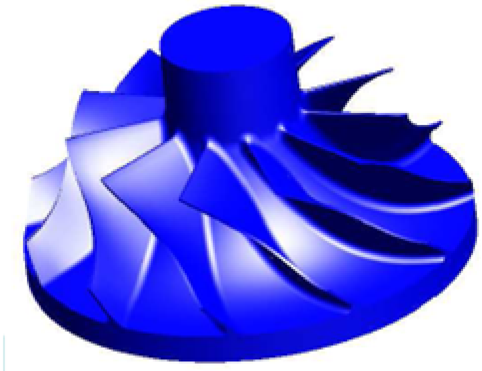


Fig.1

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).

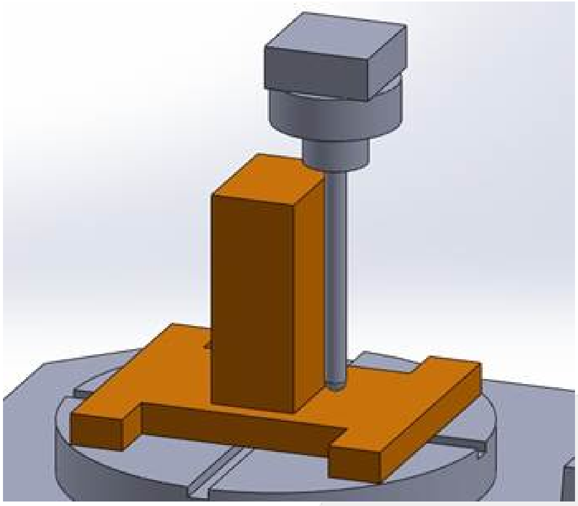


Fig.2

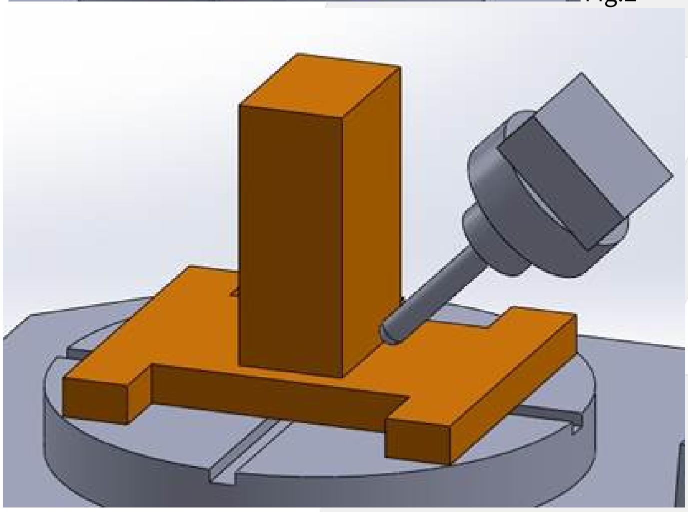


Fig.3

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

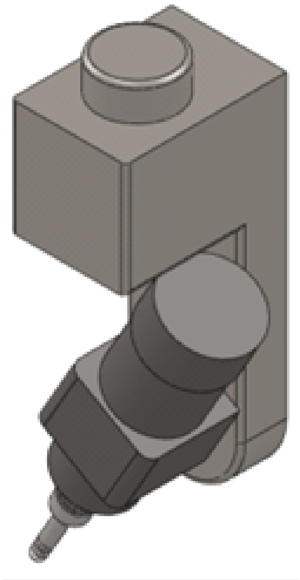
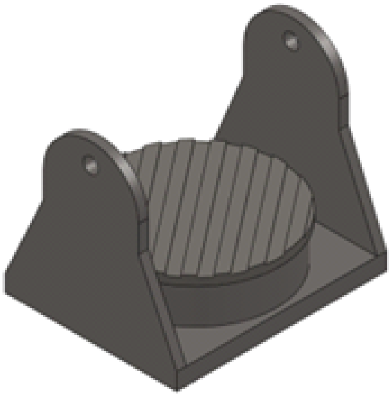
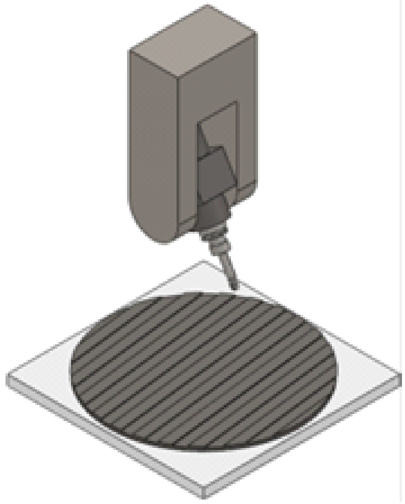
		
Spindle Type	Table Type	Mix Type

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

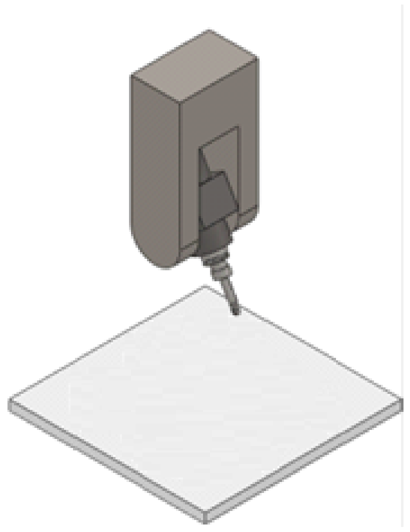
	
Single Spindle Type	Single Table Type

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.

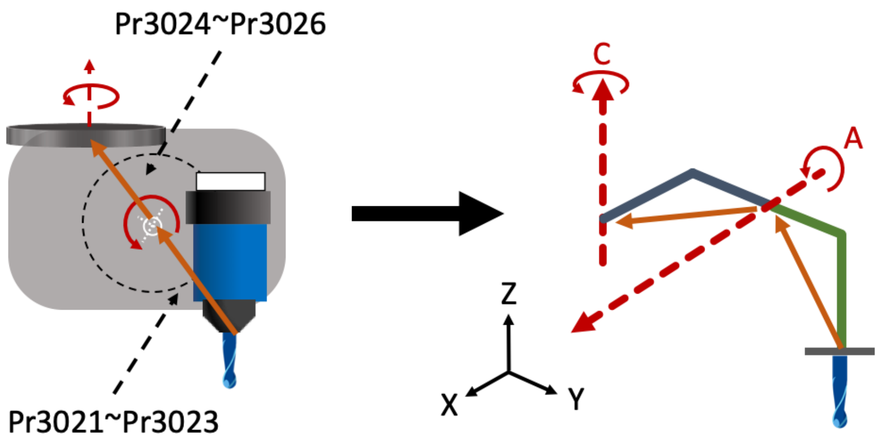


Fig.6

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.

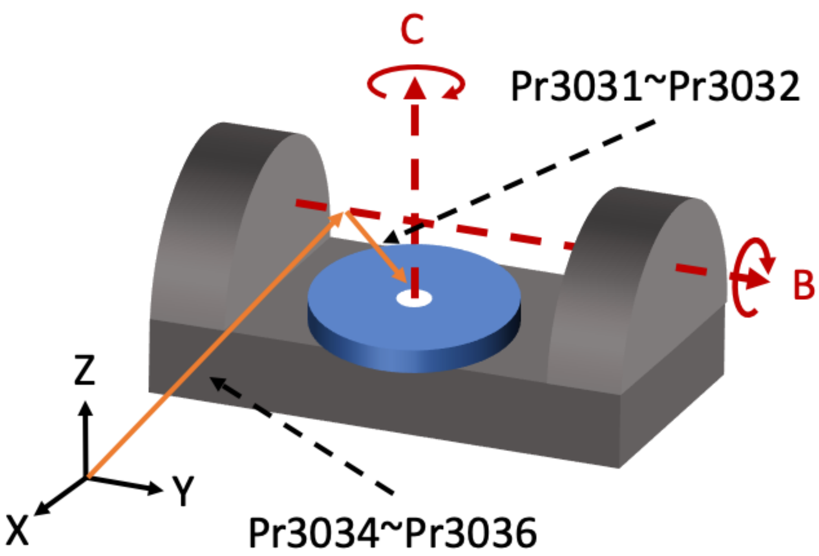


Fig.7

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.

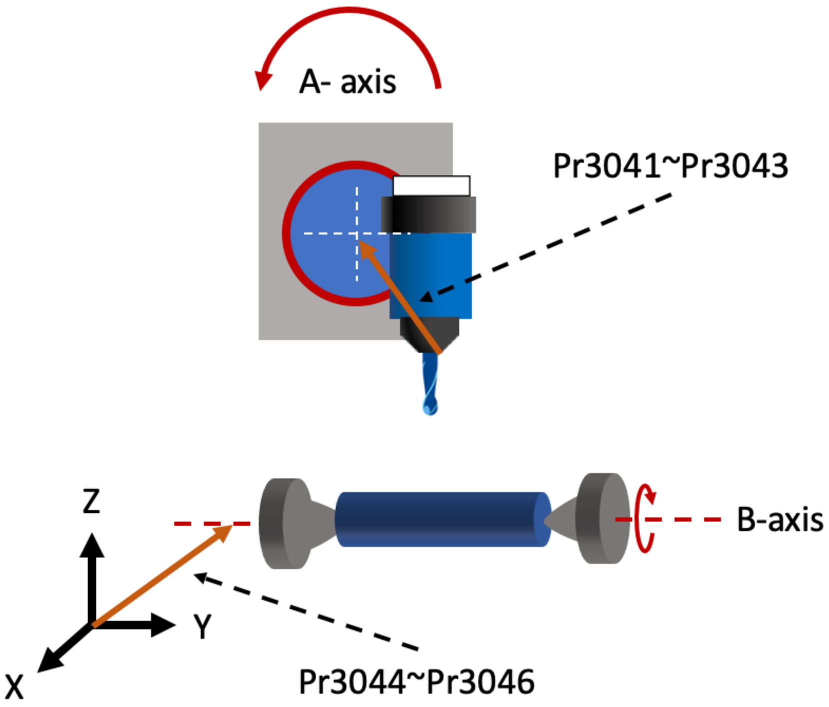


Fig.8

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.

No	Descriptions	Range	Unit	Default	Effective
3001	*1st Organization for five axis machine	[0,5]	-	0	Restart

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

No	Descriptions	Range	Unit	Default	Effective
3002	1st Direction of Tool	[0,3]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3003	1st Incline Angle of Direction of Tool(RA)	[0,360000]	BLU	0	Reset

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.

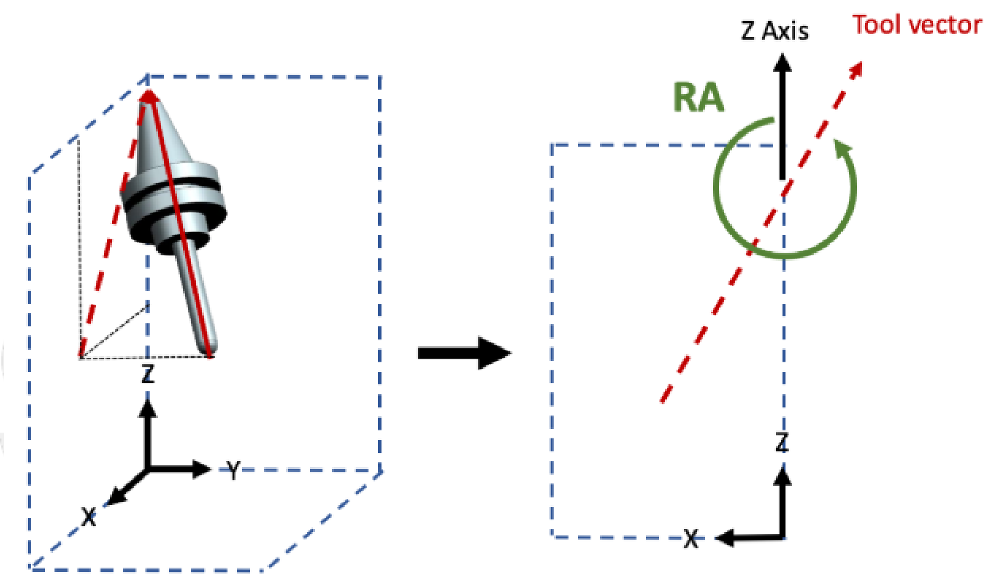


Figure 9

No	Descriptions	Range	Unit	Default	Effective
3004	1st Incline Angle of Direction of Tool(RB)	[0,360000]	BLU	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3005	1st first rotation axis	[0,3]	-	0	Reset
3006	1st second rotation axis	[0,3]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3007	1st rotation direction of first rotation axis	[0,2]	-	0	Reset
3008	1st rotation direction of second rotation axis	[0,2]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3009	1st starting point of rotation angle of first rotation axis	[0,360000]	BLU	0	Reset
3010	1st terminal point of rotation angle of first rotation axis	[0,360000]	BLU	0	Reset

3011	1st starting point of rotation angle of second rotation axis	[0,360000]	BLU	0	Reset
3012	1st terminal point of rotation angle of second rotation axis	[0,360000]	BLU	0	Reset

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.

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.

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.

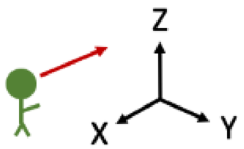


figure 13

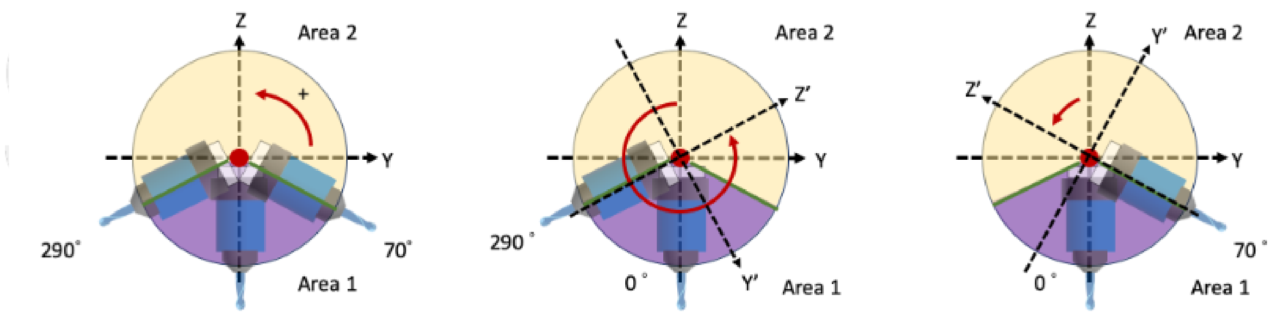


figure 14

No	Descriptions	Range	Unit	Default	Effective

3013	1st Tool Holder Offset	[0,999999999]	BLU	0	Reset

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.

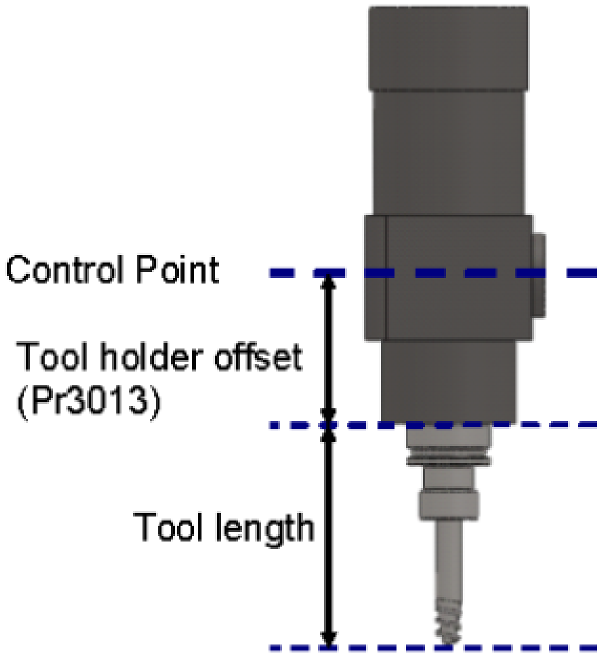


Figure 15

No	Descriptions	Range	Unit	Details	Effective
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.	Reset

No	Descriptions	Range	Unit	Default	Effective
3015	1st A-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset

3016	1st B-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset
3017	1st C-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset
3018	1st A-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset
3019	1st B-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset
3020	1st C-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset

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.

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

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

Figure 16

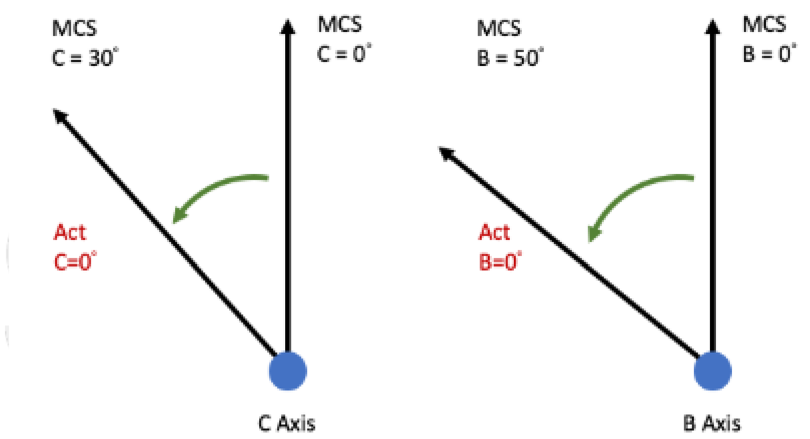


Figure 17.

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

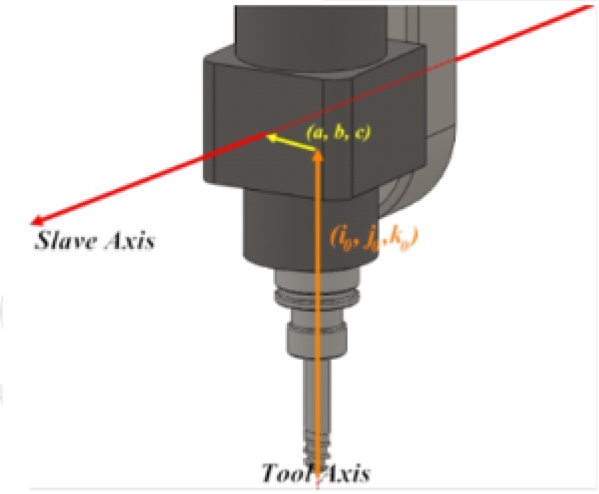


Figure 19

No	Descriptions	Range	Unit	Default	Effective

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

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

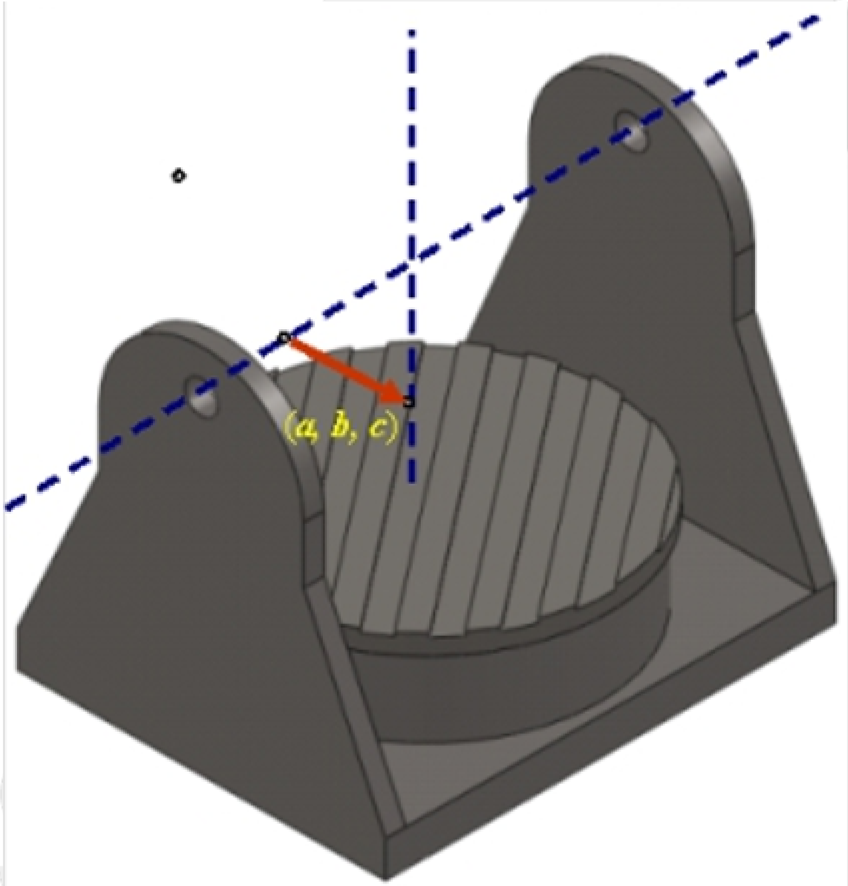


Figure 21

No	Descriptions	Range	Unit	Default	Effective
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

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)

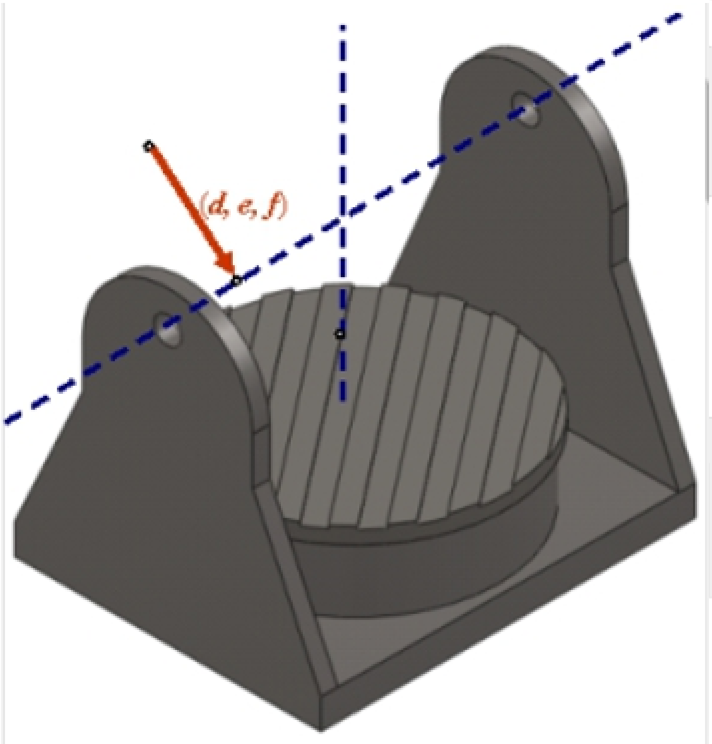


Figure 22

No	Descriptions	Range	Unit	Default	Effective

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

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

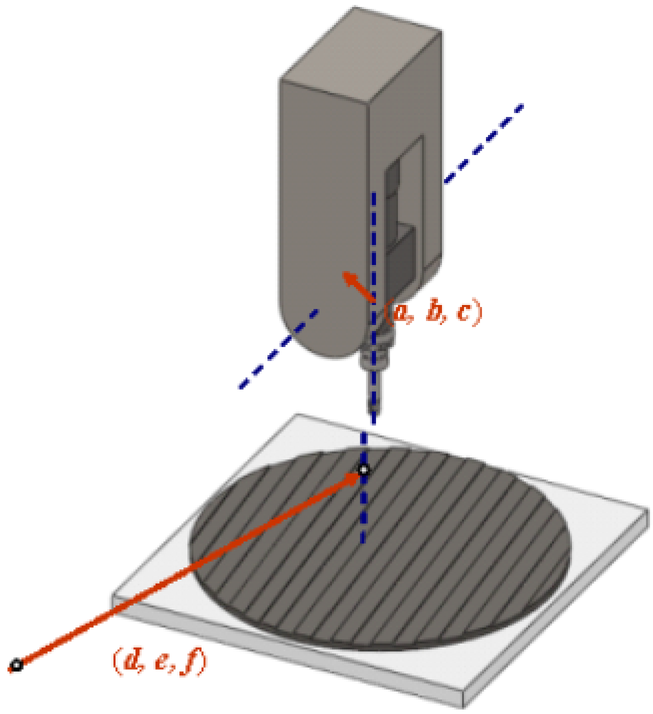


Figure 24

No	Descriptions	Range	Unit	Default	Effective
3051	Enable smooth RTCP function (0:No;1:Yes)	[0,1]	-	0	Reset

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.

No	Descriptions	Range	Unit	Default	Effective
3052	First rotation axis smoothness tolerance (0.001deg)	[1, 360000]	0.001 deg	500	Reset
3053	Second rotation axis smoothness tolerance (0.001deg)	[1, 360000]	0.001 deg	500	Reset

No	Descriptions	Range	Unit	Default	Effective

3054	1st RTCP interpolation mode (0: Five axis simultaneously; 1: Tool vector)	[0,1]	-	0	Reset

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.

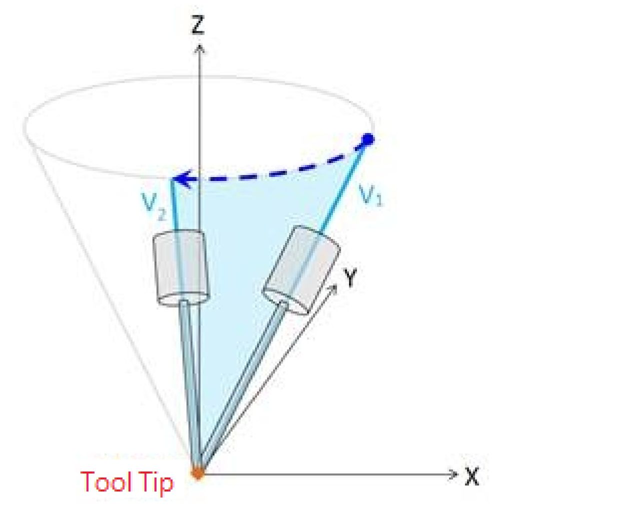
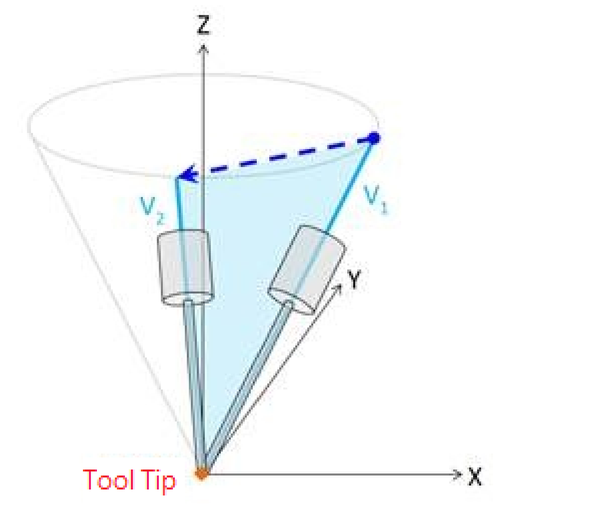
Note

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.

Example 1

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

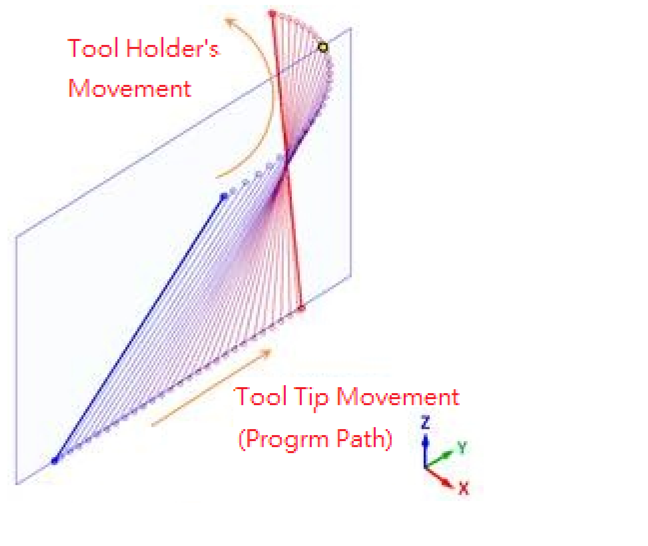
Five Axis Simultaneous	**Tool Vector**
Tool posture changes during movement	**Tool posture stays on the same plane**

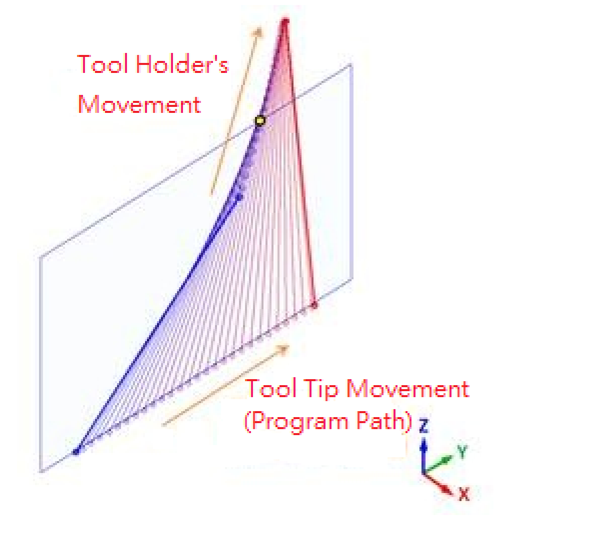
	
Because there is no command for B axis, so B axis stays still; the tool posture is slightly off from the plane composed of V_1 and V_2.	To stay on the plane composed of V_1 and V_2 during movement, B axis rotates without command and eventually returns to its original angle.

ⓘ 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 Vector**
Tool posture changes during movement	**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.

No	Descriptions	Range	Unit	Default	Effective
3055	Basic position of table coordinate system	[0,1]	-	0	Reset

Value

0: RTCP start position

1: zero position of workpiece coordinate

Description

- Table coordinate system：
  Table coordinate system is a workpiece coordinate system exclusive to RTCP and only exist during RTCP is enabled.
  When RTCP is enabled, table coordinate will be established on the table.
  During RTCP is enabled, the commands in NC program will be interpreted as the commands of table coordinate.
- Rotation of the table coordinate system：
  When RTCP is enabled, zero position of rotary axis in table coordinate will be fixed according to Pr3055.

Pr3055 = 0 :

Zero position of rotary axis in table coordinate will be fixed on "current machine coordinate of rotary axis".

Pr3055 = 1 :

Zero position of rotary axis in table coordinate will be fixed on "zero position of rotary axis in workpiece coordinate", regardless of current angle of rotary axis.

During RTCP is enabled, the table coordinate will rotate along with the table.

- Difference in application:

If Pr3055 is 0, user should locate rotary axis to the specific angle before enabling RTCP.

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.

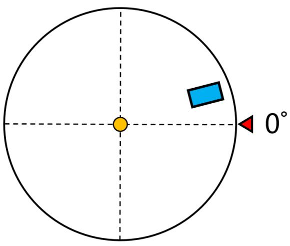
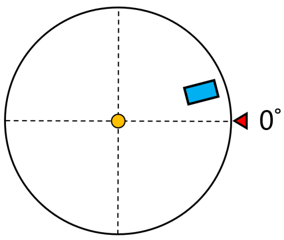
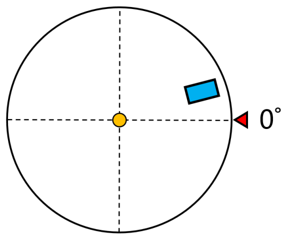
Notice

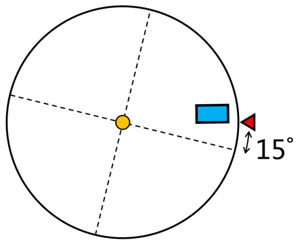
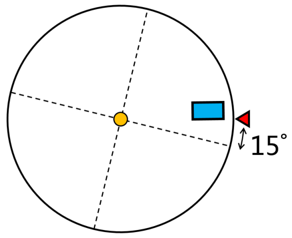
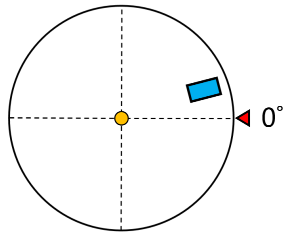
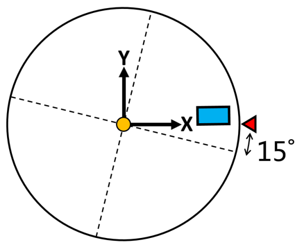
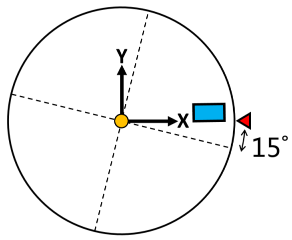
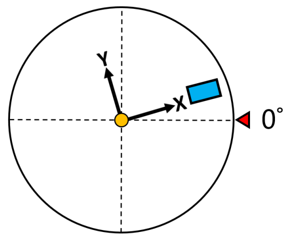
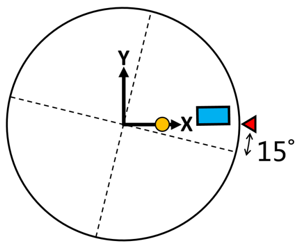
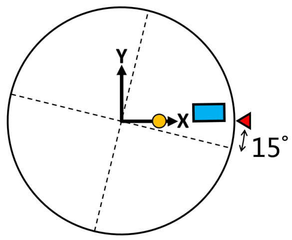
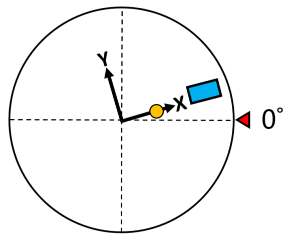
- If the parameter is modified when RTCP is enabled, it will take effect when RTCP being enabled next time.
- This parameter is only valid for the rotary axes on the table side.

Example

- The purpose of following examples is making the table coordinate to align to the angle of the workpiece (that is C15. in machine coordinate).
- If Pr3055 is 0, like example A, user should locate C axis to C15. in advance before enabling RTCP.
  If Pr3055 is 1, like example B and C, user only need to fill C15. in G54 offset.

> **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	无	C+15.	C+15.
NC Command	(locate C axis, then enable RTCP) G90 G49 G54 N1 C15. // C positioning N2 G43.4 H1 Z0. // RTCP ON N3 X10. Y0. Z0. // tool tip moves	(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			

<table>
<tr>
<td>N1</td>
<td>C15. // C positioning
</td>
<td>C0. // C positioning
</td>
<td>// do nothing
</td>
</tr>
<tr>
<td>N2</td>
<td>G43.4 H1 Z0. // RTCP ON
</td>
<td>G43.4 H1 Z0. // RTCP ON
</td>
<td>G43.4 H1 Z0. // RTCP ON
</td>
</tr>
<tr>
<td>N3</td>
<td>X10. Y0. Z0. // tool tip moves
</td>
<td>X10. Y0. Z0. // tool tip moves
</td>
<td>X10. Y0. Z0. // tool tip moves
</td>
</tr>
<tr>
<td>Note</td>
<td>The machine coordinate of C axis is 15. when RTCP is enabled.

Therefore, the zero position of C axis in table coordinate is at C15. in machine coordinate.</td>
<td colspan="2">Regardless of the angle of C axis when RTCP is enabled, the zero position of C axis in table coordinate is the same as zero position of C axis in workpiece coordinate.</td>
</tr>
</table>

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.

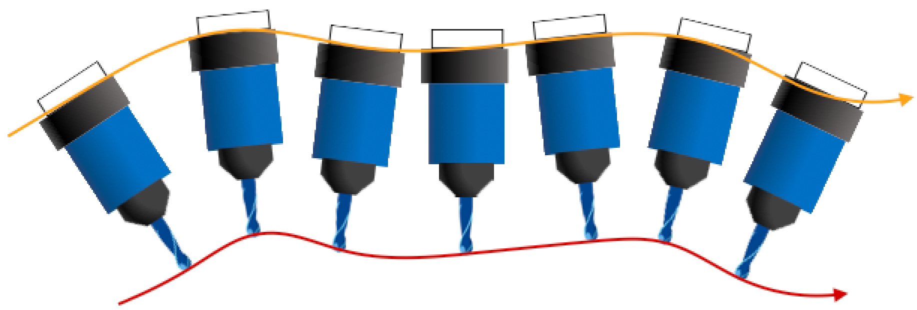


Fig.25

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.

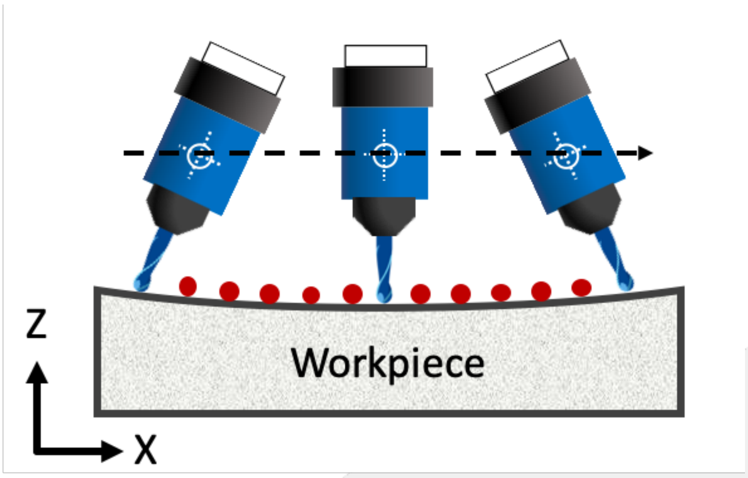


Fig.26

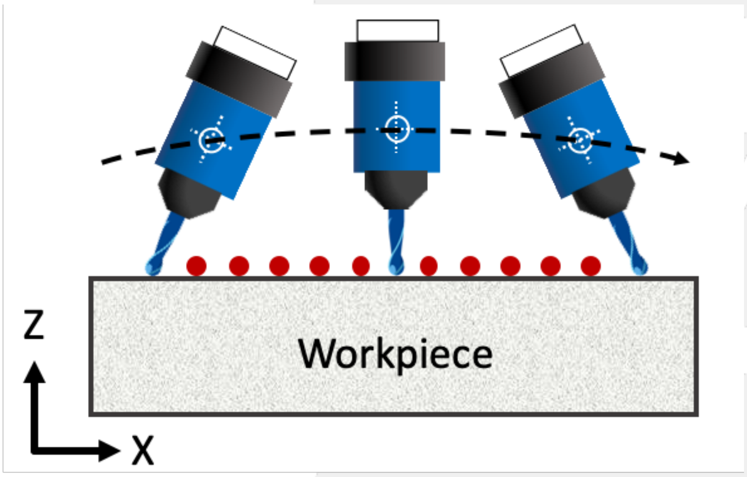


Fig.27

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)
```

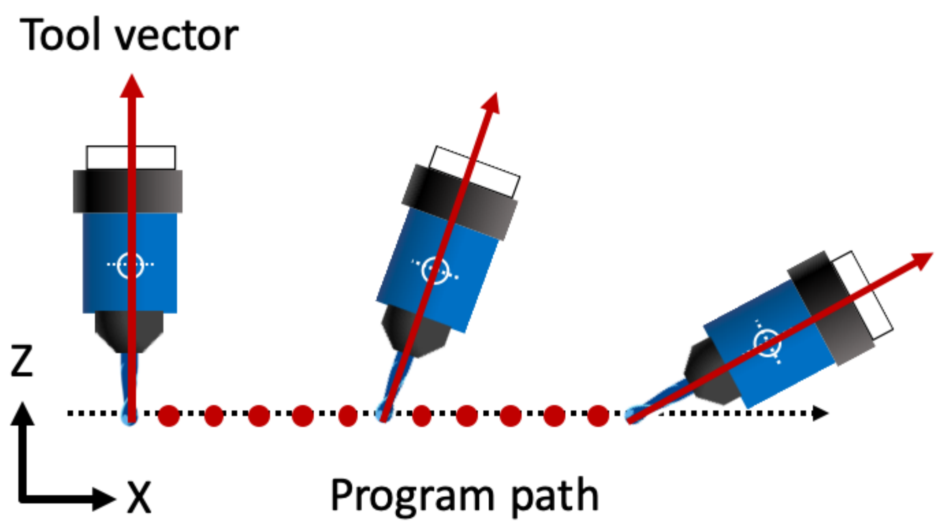


Fig.28

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.

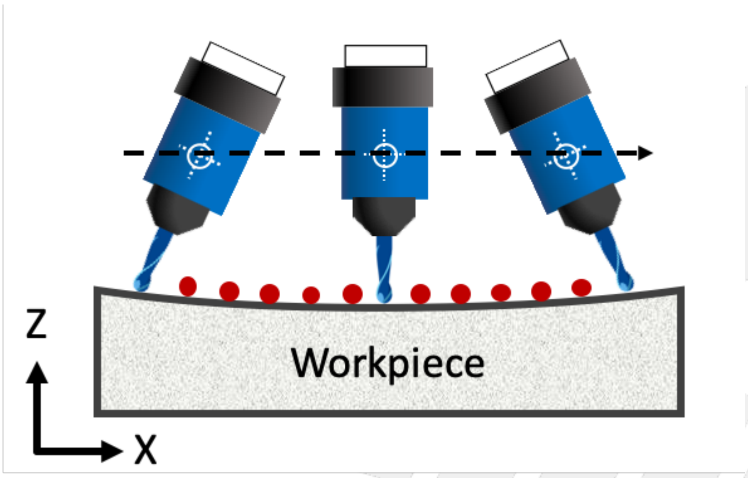


Fig.29

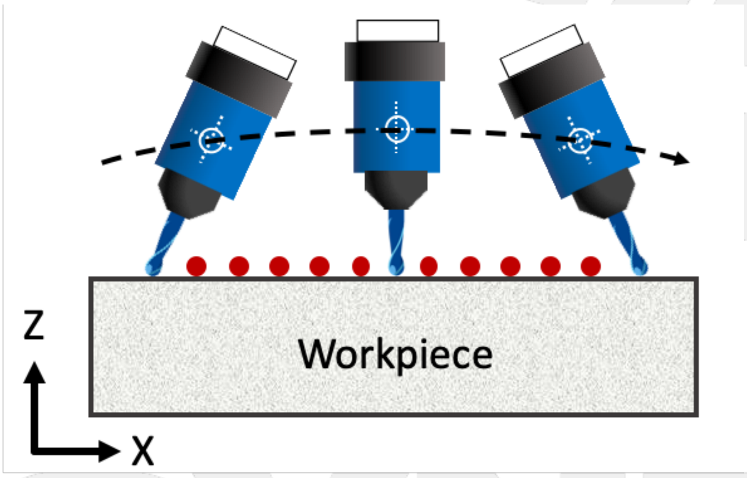


Fig.30

3 3. Extended Functions of RTCP

Extended Functions of RTCP will be introduced in this chapter.

3.1 3.1 Smooth Tool Center Point (STCP)

STCP (Smooth Tool Center Point) is the smoothing function of tool center point control.

When STCP is enabled, in addition to RTCP function, the tool orientation and the path of tool center point will also be smoothed to make the cutting process and workpiece surface smoother.

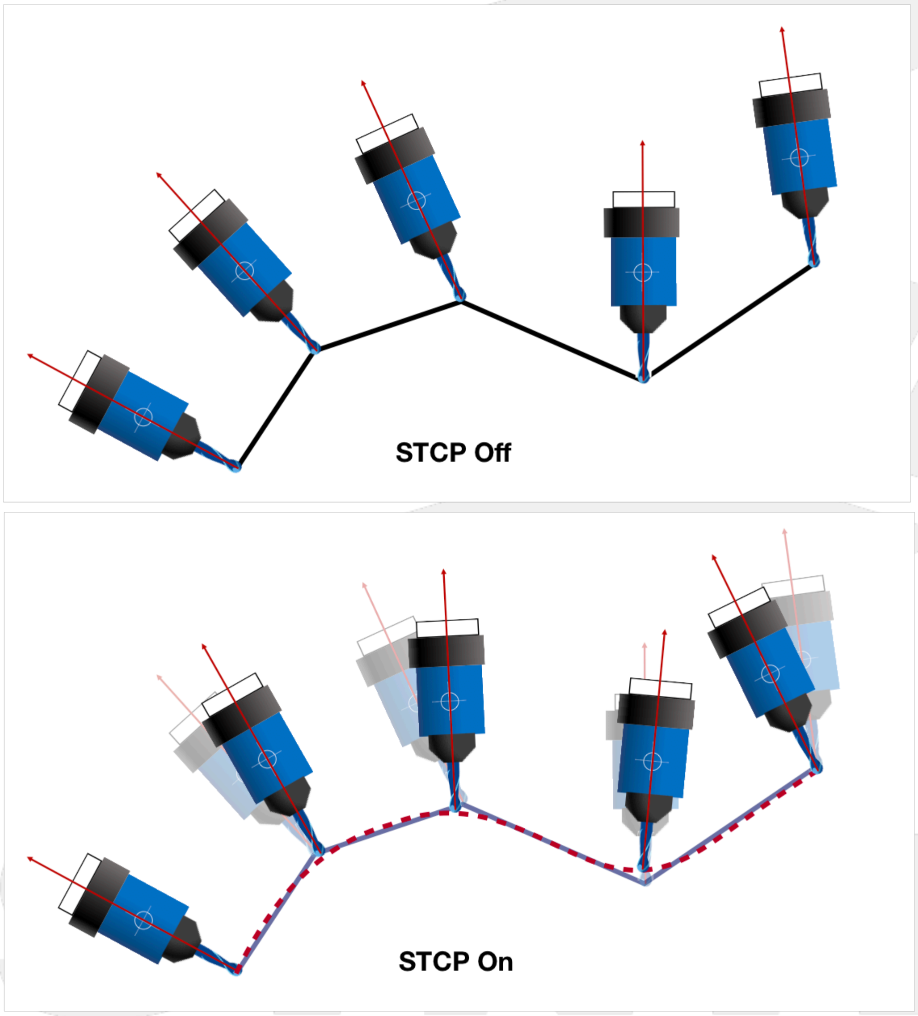


Fig.31

3.1.1 Command Format

```
G43.4 H_ [L_]；
G49；
```

G43.4：enable RTCP;
H ：tool compensation number;
L1：enable STCP;
L0：disable STCP;
G49：disable RTCP, which disable STCP at the same time;

In short, the L argument added behind G43.4 decides whether to enable STCP function, L1: enable; L0: disable. When Pr3051 is set as 1, the L argument can be omitted and STCP will still be enabled.

3.1.2 Application Limitations

1. For STCP, RTCP and HPCC options are all required; after version 10.116.38D, 10.116.54B, 10.118.0A (included), RTCP and STCP options are all required.
2. Only the commands for 5 related axis are allowed under STCP mode, or alarm【COR-107 The format of G5.1/G05 is incorrect】will be triggered.
3. Enable HPCC function with G05 P10000 under STCP mode will trigger alarm【COR-140 G05 HPCC cannot apply under RTCP mode】.

3.1.3 Notifications

1. The word HPCC will be shown on monitor page when STCP function is enabled.
2. Pr3051~Pr3053 are only visible when STCP and RTCP options are both enabled.

3.1.4 Related Parameters

No.	Description	Range	Unit	Initial Value	Take Effect
407	HPCC smoothing tolerance	[2~20]	um	3	Reset
3051	enable smooth RTCP function	[0,1]	-	0	Reset
3052	first rotation axis smooth tolerance	[1,360000]	0.001 deg	500	Reset
3053	second rotation axis smooth tolerance	[1,360000]	0.001 deg	500	Reset

3.2 3.2 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).

3.2.1 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.

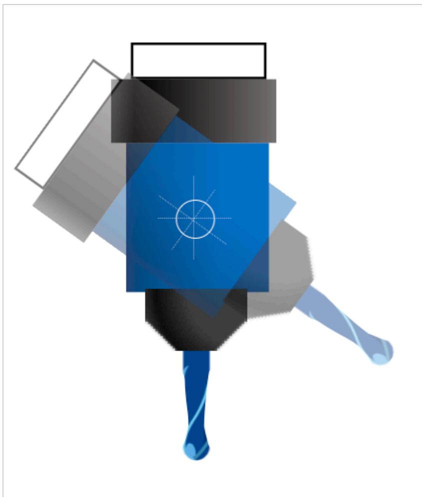
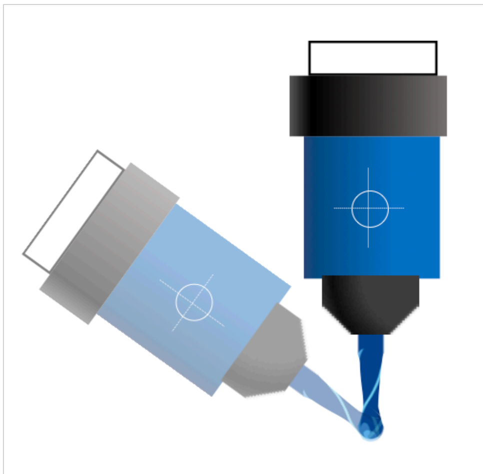
	
R519 = 0	R519 = 1

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

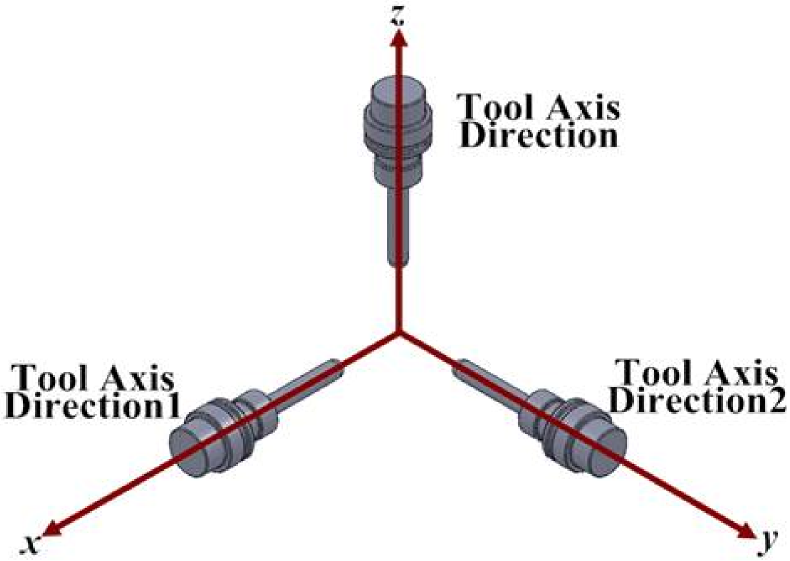


Fig.33

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.

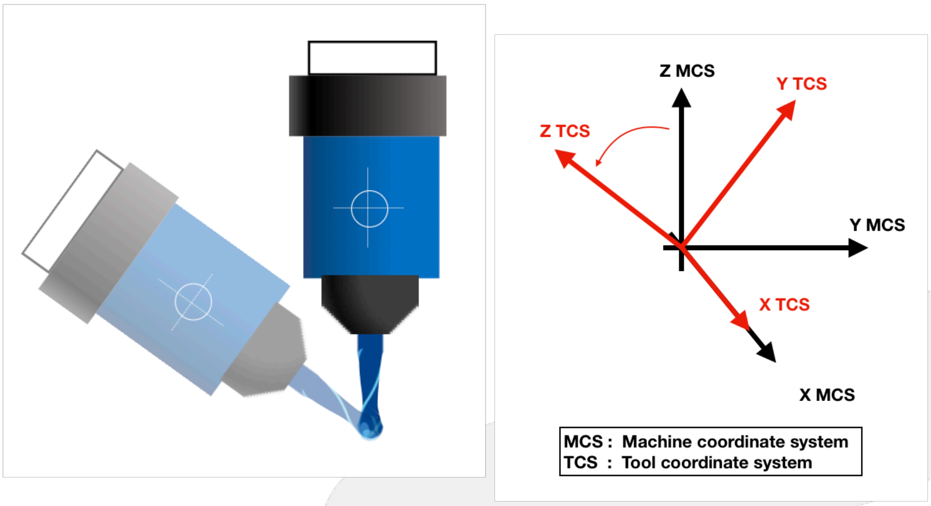


Fig.34

3.2.2 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.

3.2.3 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.

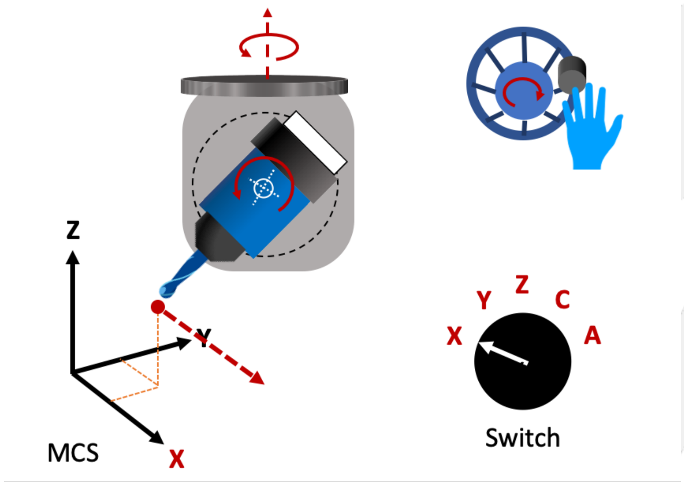


Fig.35

R518 = 1

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

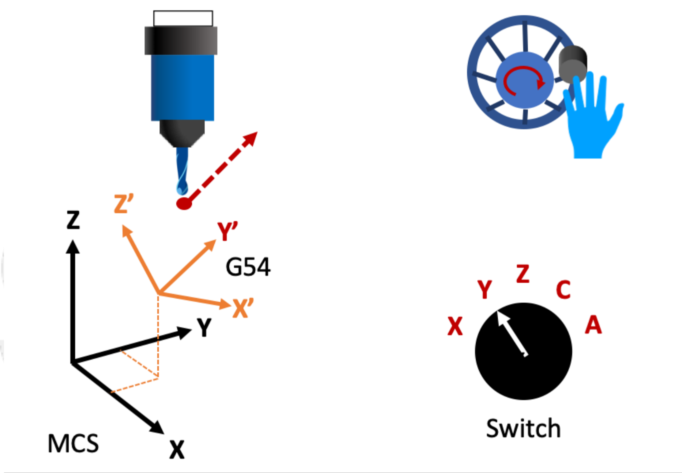


Fig.36

R518 = 2

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

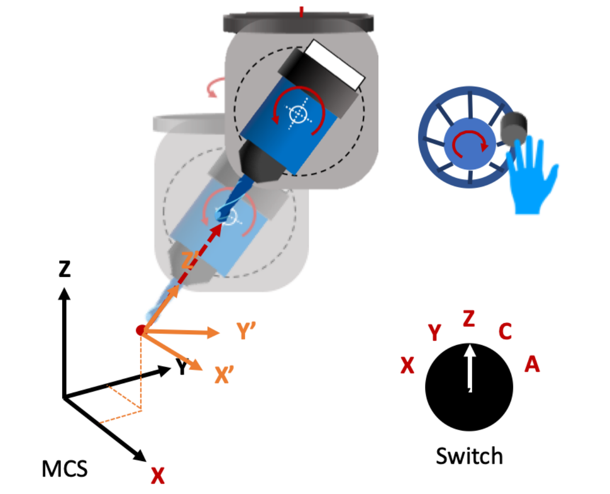


Fig.37

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 3.3 G68.2 Tilted Working Plane Teach Function

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

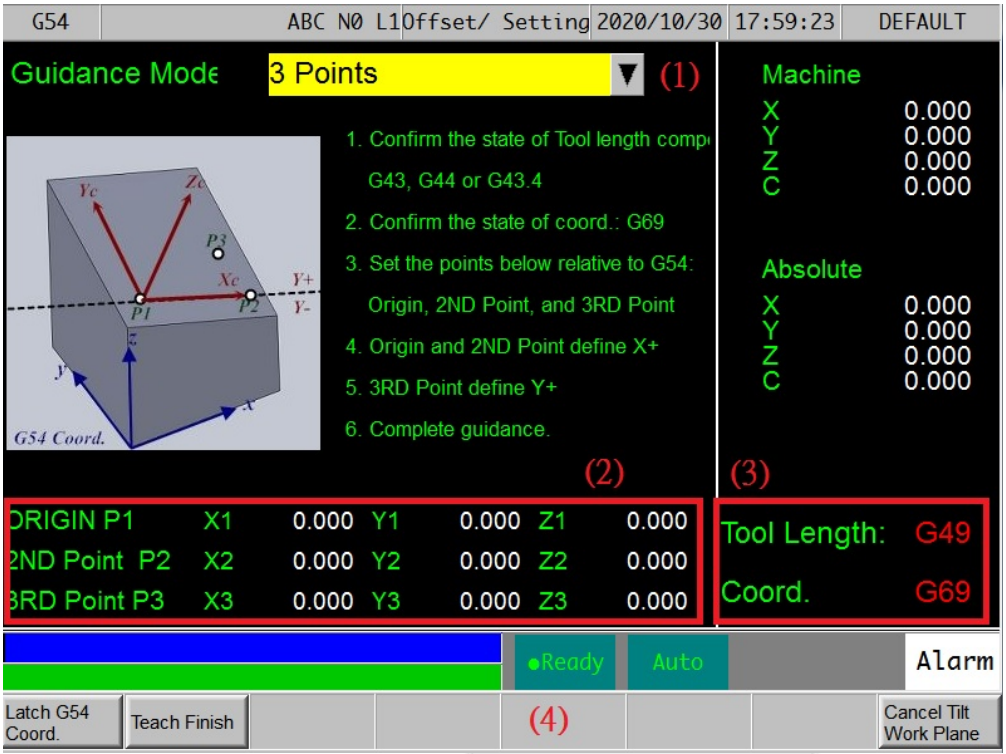


Fig.38

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.

3.4 3.4 Teach Modes Description

3.4.1 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.

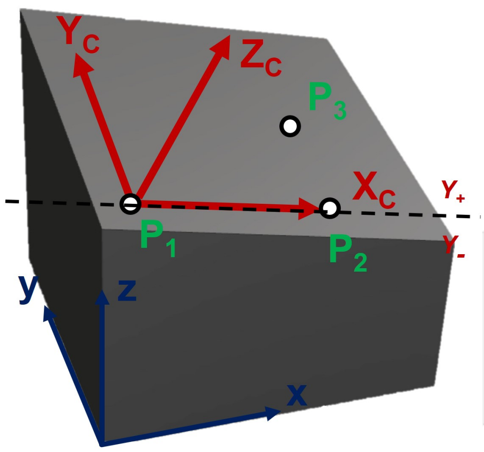


Fig.39

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.

3.4.2 Tool Direction

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

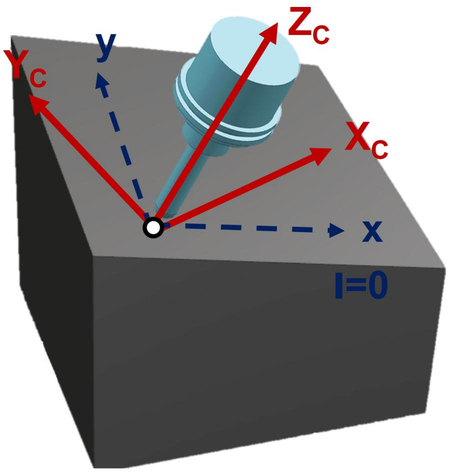


Fig.40

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.

3.4.3 Euler Angle

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

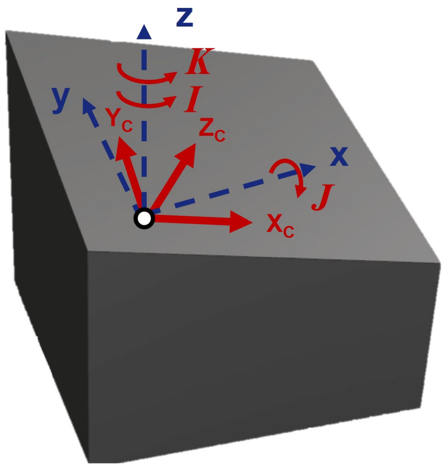


Fig.41

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 4.1 G68.2 Tilted Working Plane Machining (Euler Angle) for the definition of Euler angle.

3.4.4 2 Vectors

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

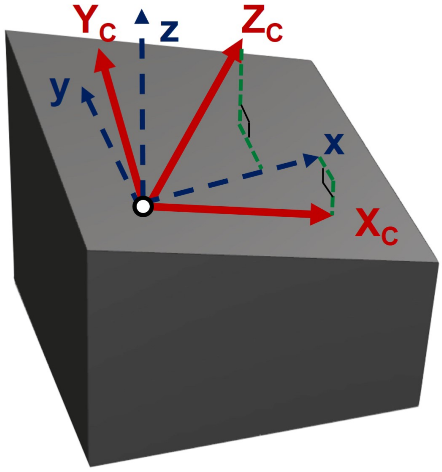


Fig.42

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. Tilted Working Plane Machining

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

4.1 4.1 G68.2 Tilted Working Plane Machining (Euler Angle)

4.1.1 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.

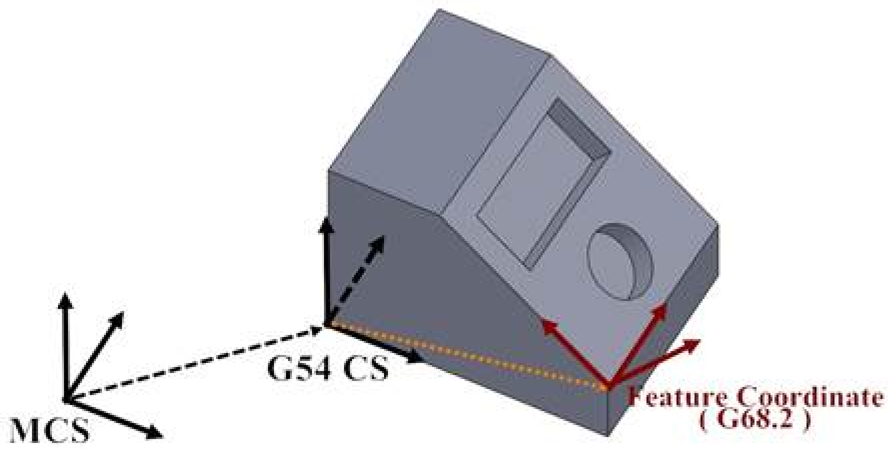


Fig.43

4.1.2 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.

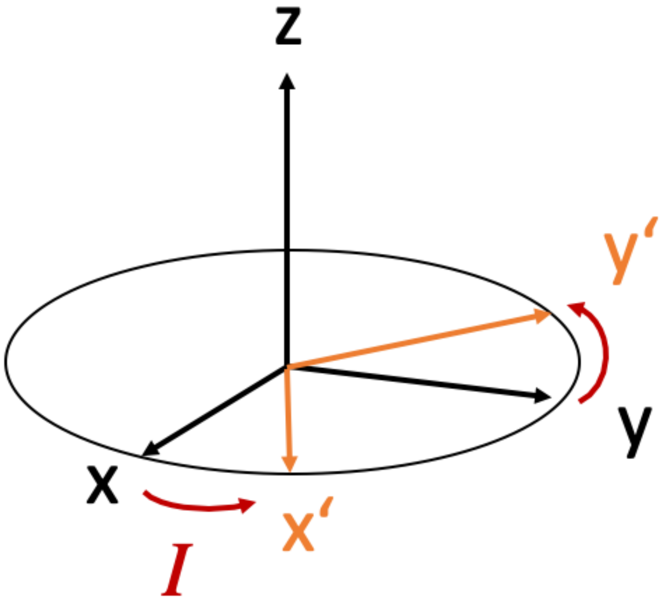


Fig.44

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.

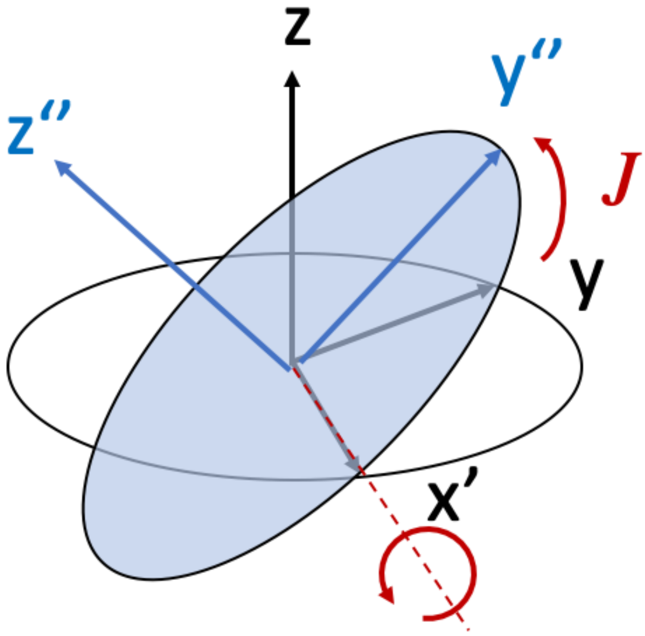


Fig.45

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.

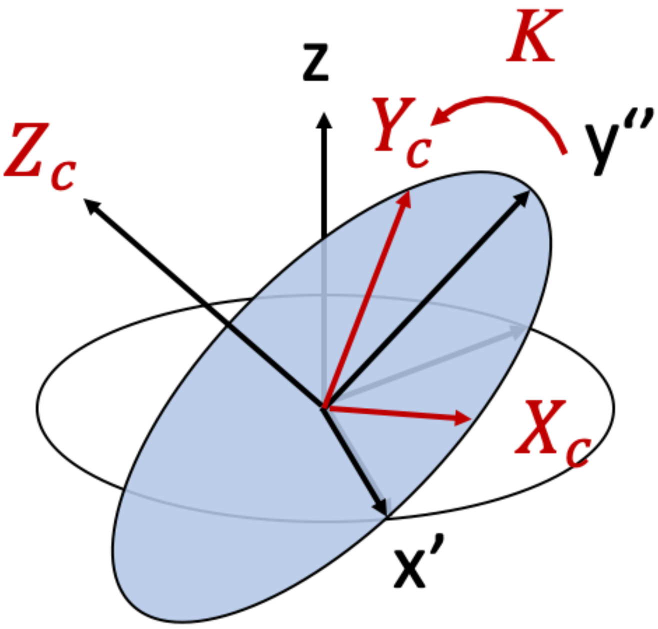


Fig.46

4.1.3 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;

4.1.4 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.

4.1.5 Related Parameters

No	Descriptions	Range	Unit	Details	Effective
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.	Reset

4.2 4.2 G53.1 Tool Alignment Function for Tilted Working Plane

4.2.1 **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.

4.2.2 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.

4.2.3 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；
```

4.2.4 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.
```

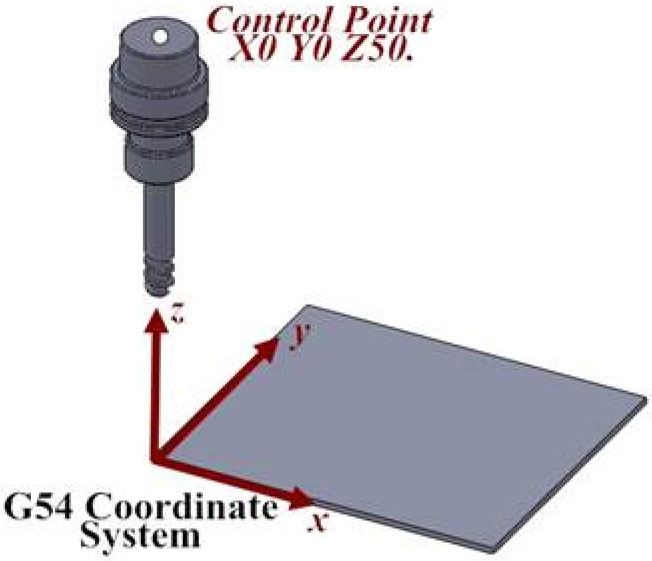


Fig.47

```
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.
```

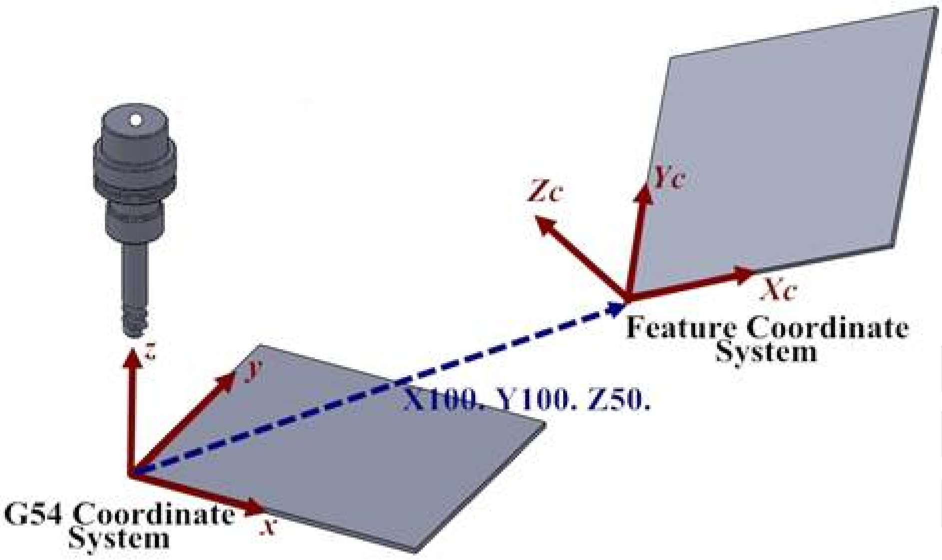


Fig.48

```
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.
```

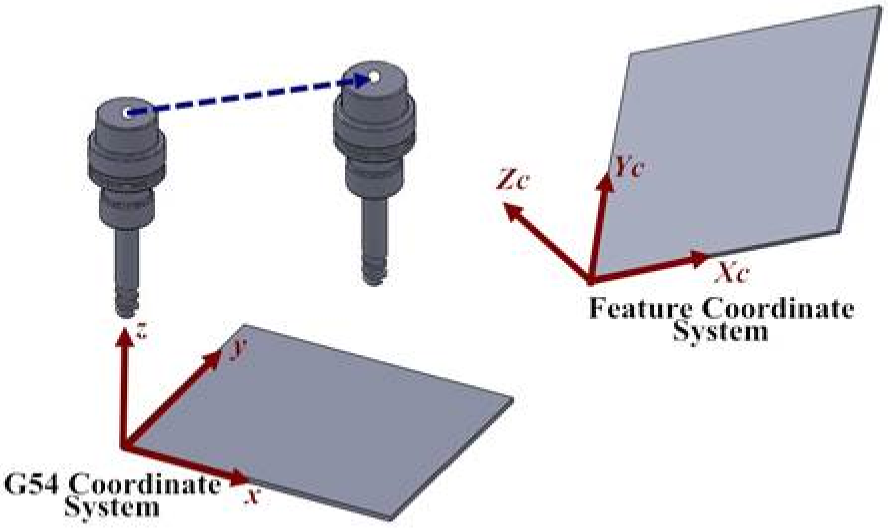


Fig.49

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

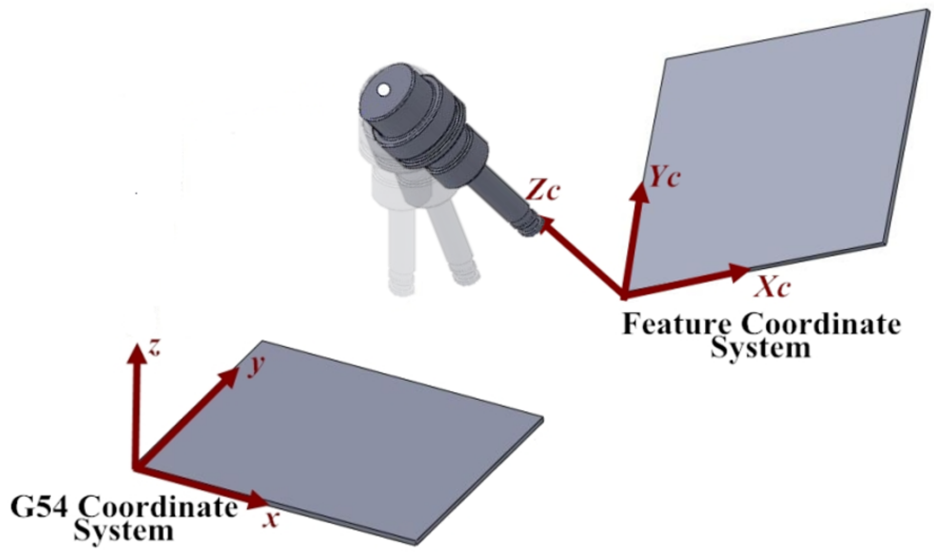


Fig.50

```
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.
```

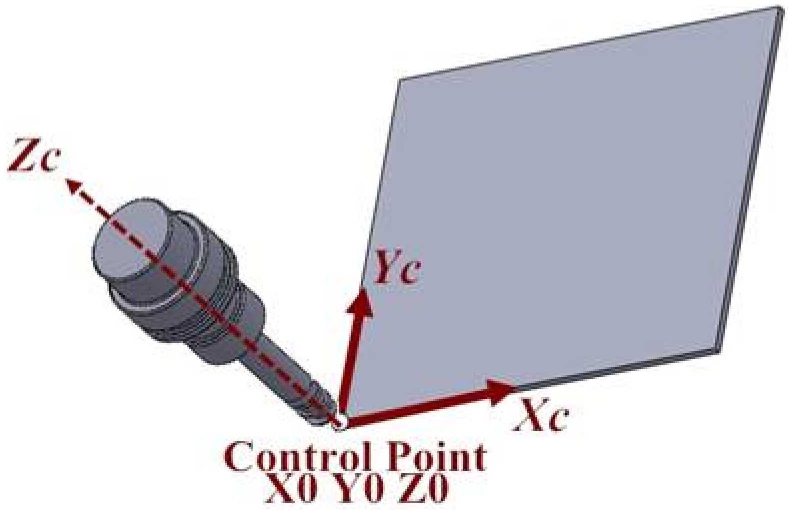


Fig.51

```
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.
```

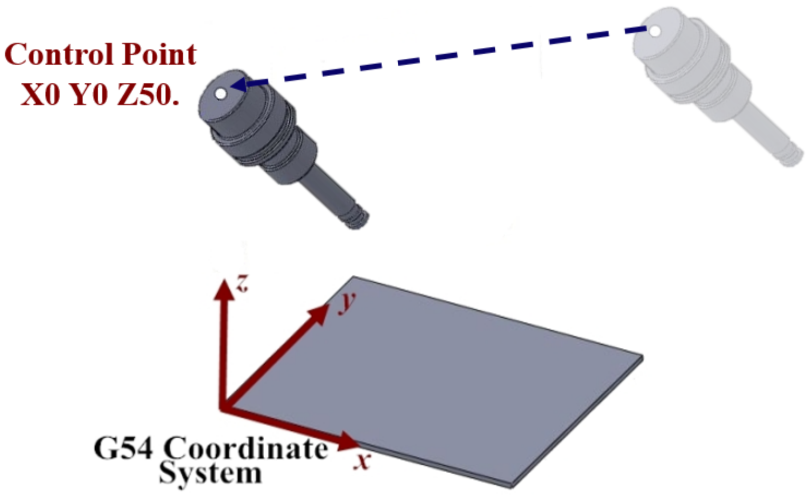


Fig.52

4.3 4.3 G53.3 Tool Alignment Function for Tilted Working Plane (5-Axis simultaneous motion)

4.3.1 **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.

4.3.2 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.

4.3.3 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

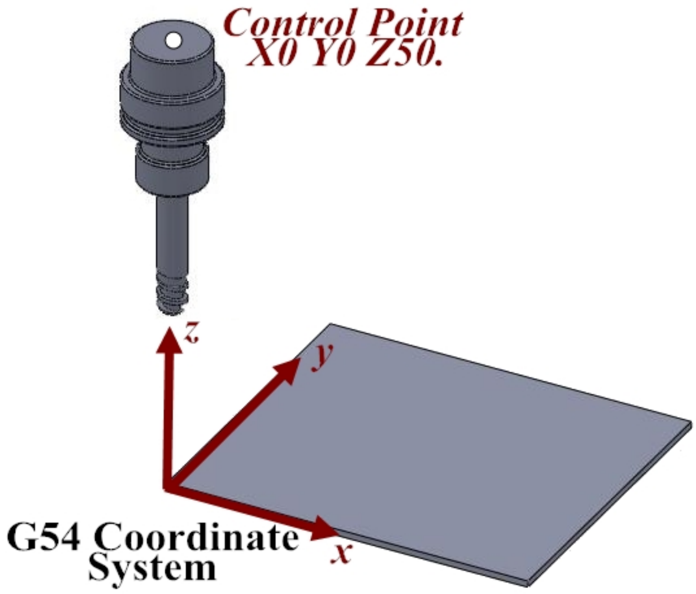
4.3.4 Program Example

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

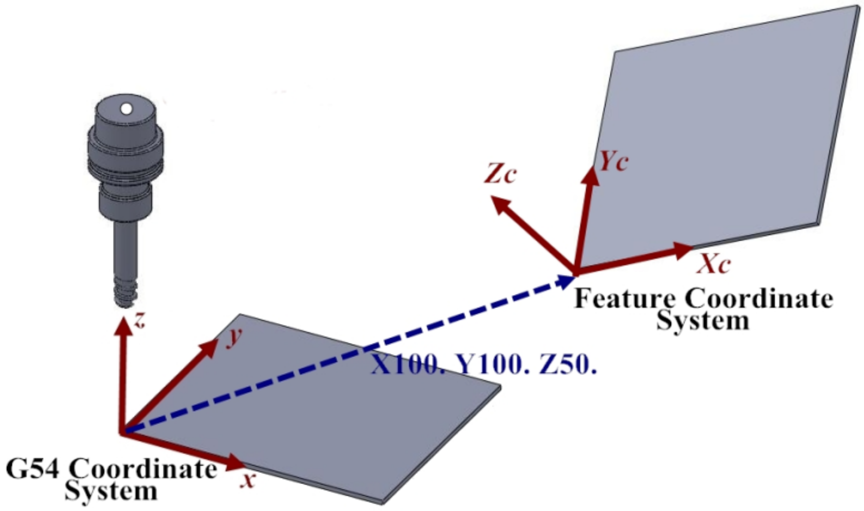
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
N3 G01 X0 Y0 Z50. F1000;
N4 G53.3 X0 Y0 Z0 H1;

... //  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:

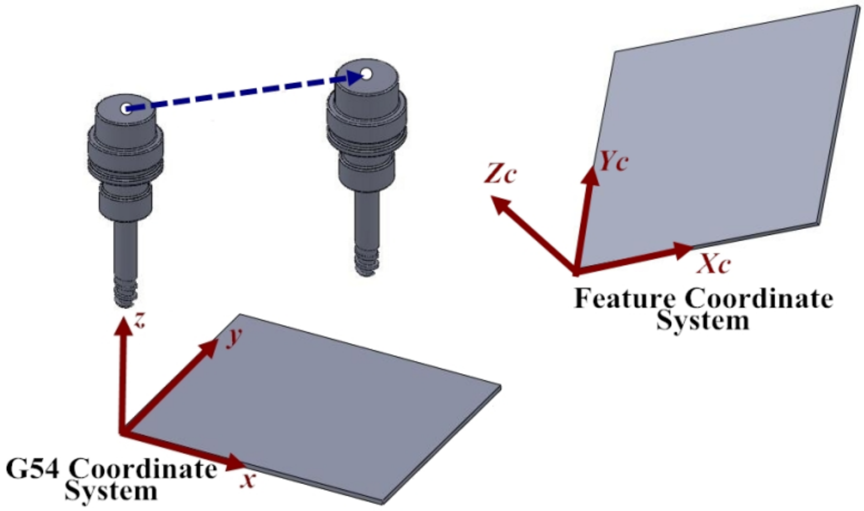
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of G54 coordinate by G01 in speed of 1000 mm/min.
```



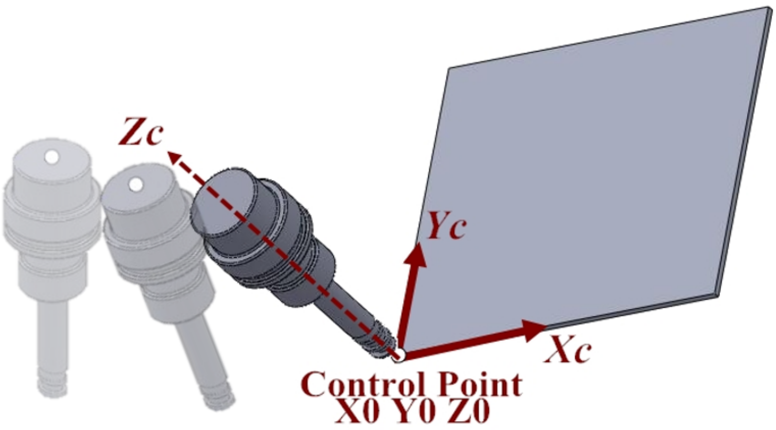
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
// Specify X100. Y100. Z50. relative to the origin of G54 coordinate as the origin of Tilted Working Plane, and the Euler angles are I30. J15. K20.
// The program coordinate will transform to Tilted Working Plane after G68.2 is executed.
```



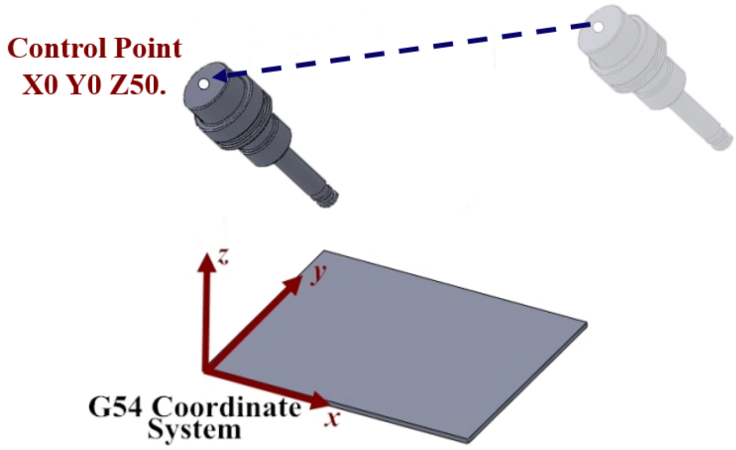

```
N3 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of Tilted Working Plane by G01 in speed of 1000 mm/min, but the tool direction remains the same.
```



```
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.
```



4.4 4.4 G53.6 Tool Alignment Function for Tilted Working Plane (TCP/ Rotation Center)

4.4.1 **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.

4.4.2 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.

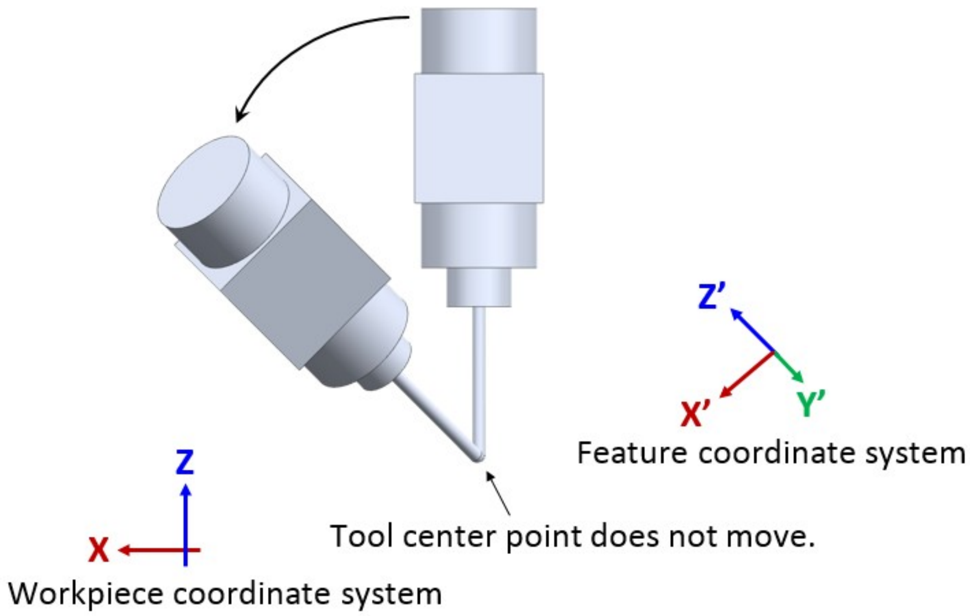


Fig.52

- **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.

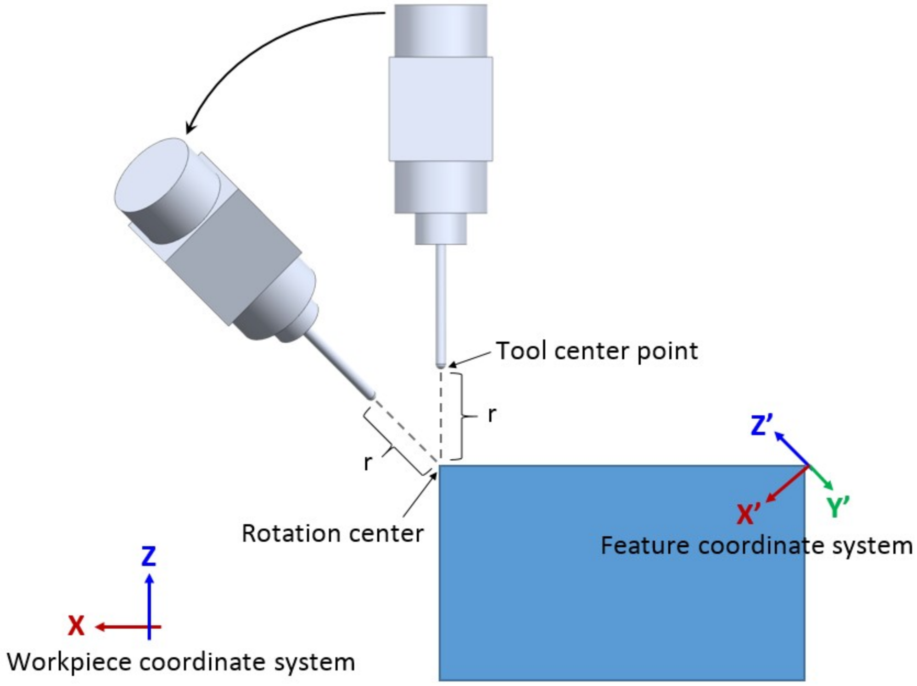


Fig.53

4.4.3 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

4.4.4 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.
```

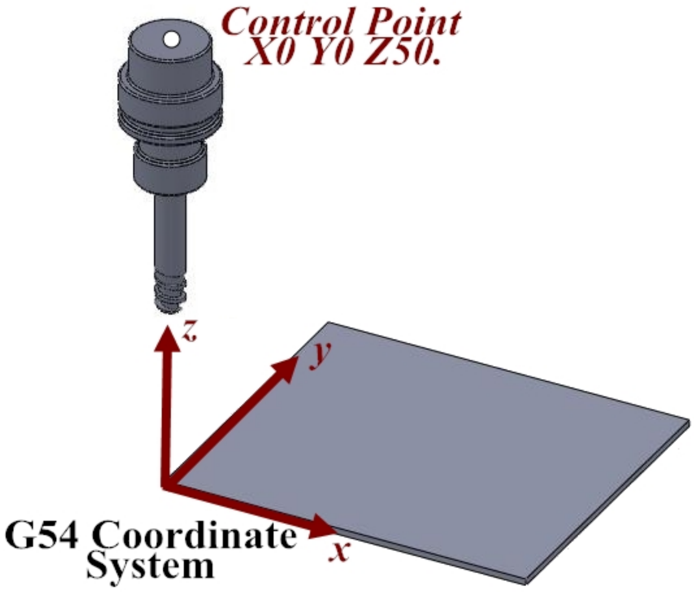


Fig.54

```
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.
```

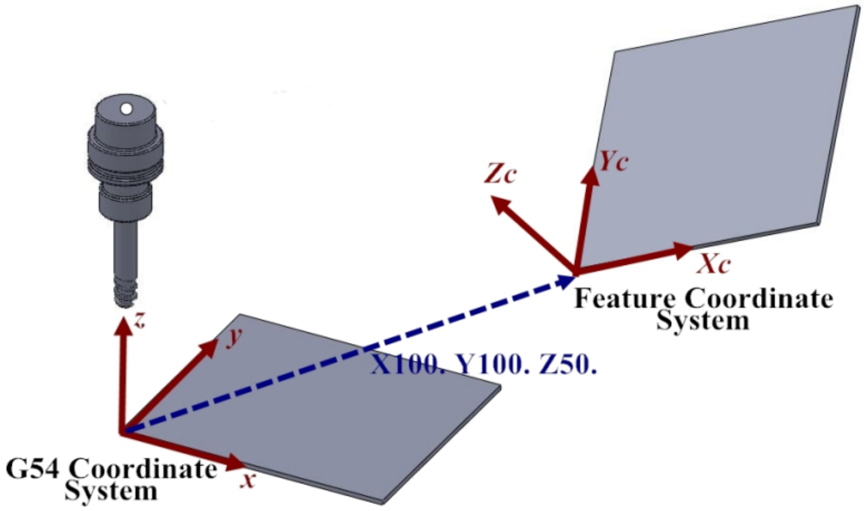


Fig.55

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

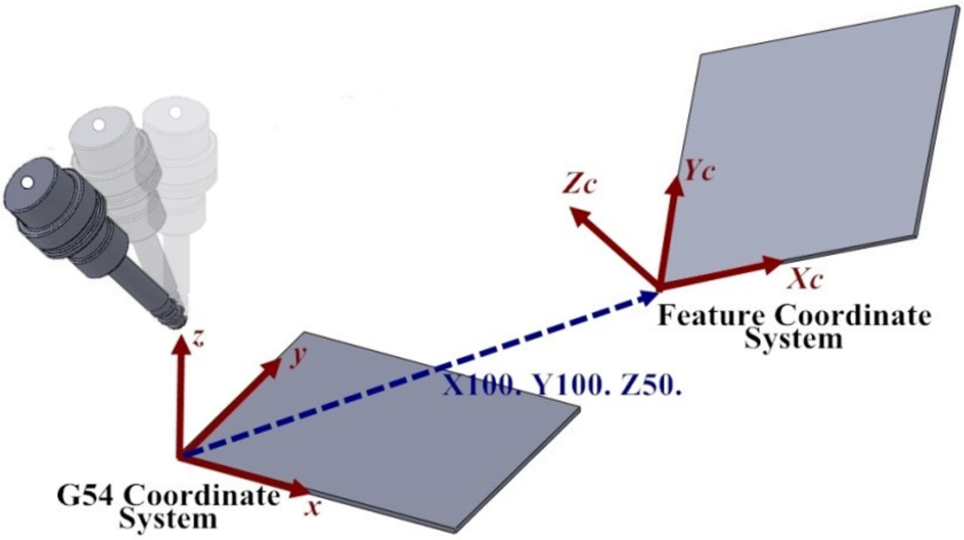


Fig.56

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

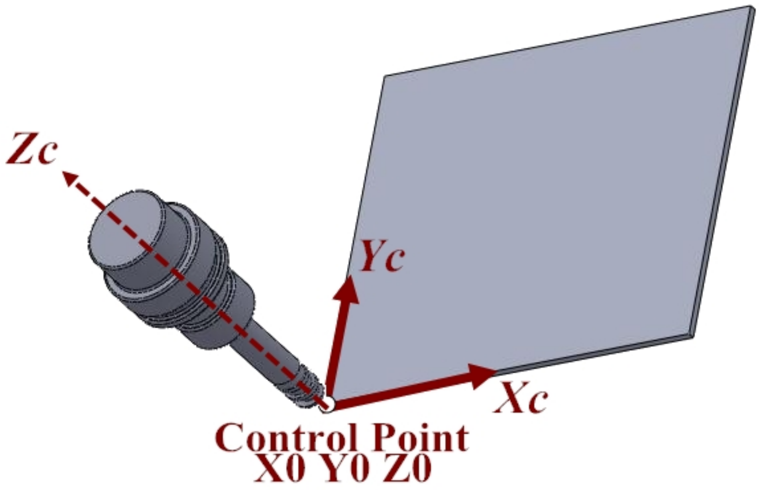


Fig.57

```
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.
```

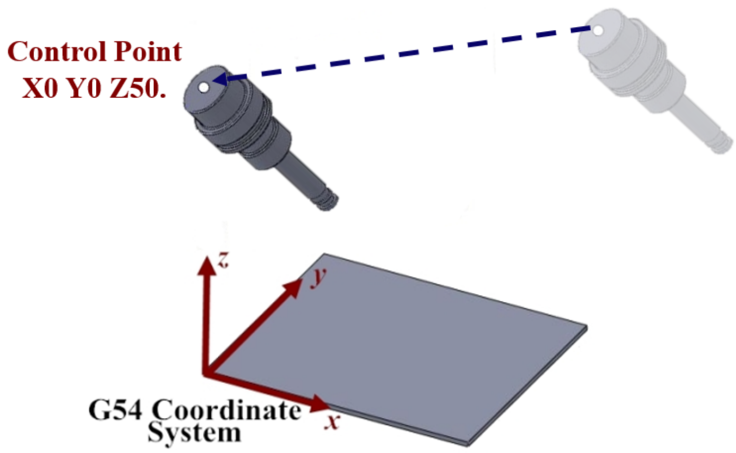
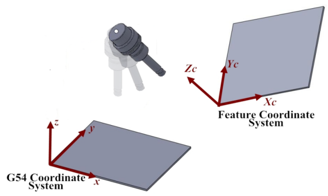
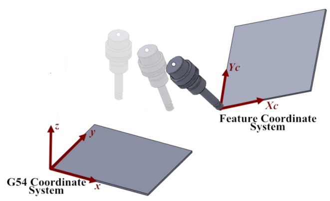
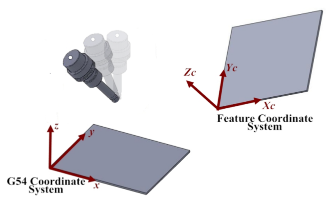


Fig.58

4.5 4.5 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
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

4.6 4.6 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.

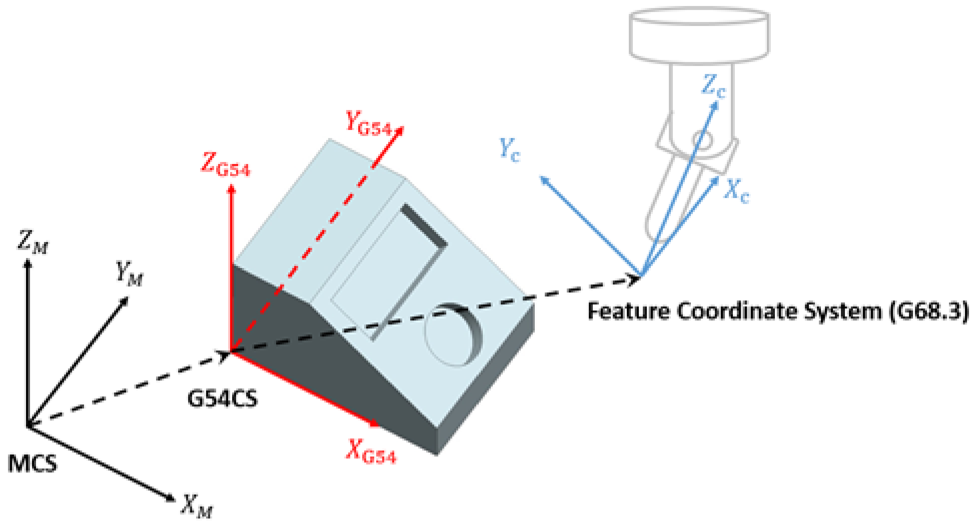


Fig.57

4.6.1 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***

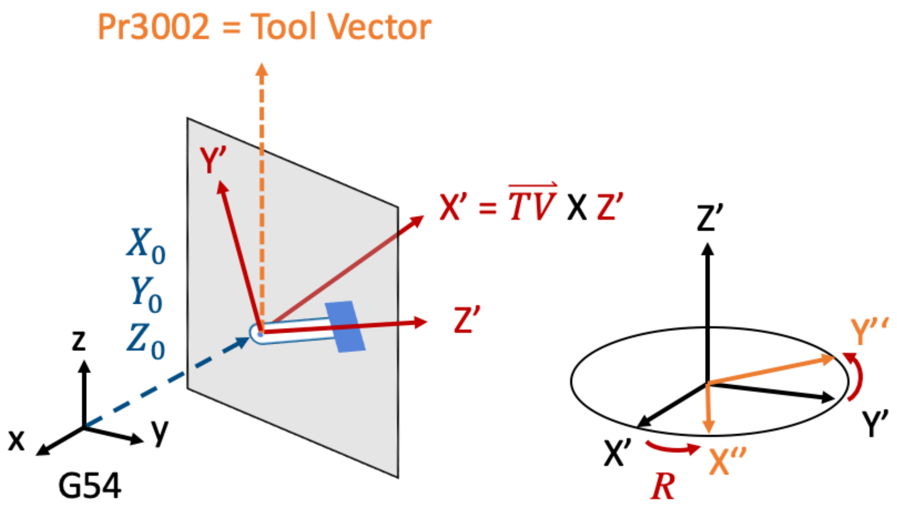


Fig.58

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*

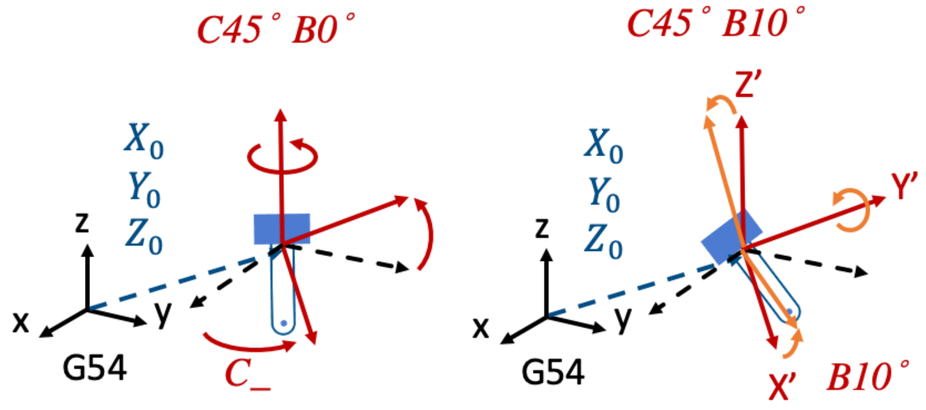


Fig.59

4.6.2 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. G43 should be executed after G68.3.
4. G68.3 can NOT be executed while RTCP (G43.4/G43.5) is enabled.
5. When G68.3 P1 is executed, R argument will be ignored if provided.
6. G68.3 can be executed for multiple times and each setting is relative to G54 coordinate.
7. G68.3 is only for 5-axis machines with option-13 enabled at the same time.

4.6.3 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.

4.6.4 Program Example:

4.6.5 (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;
```

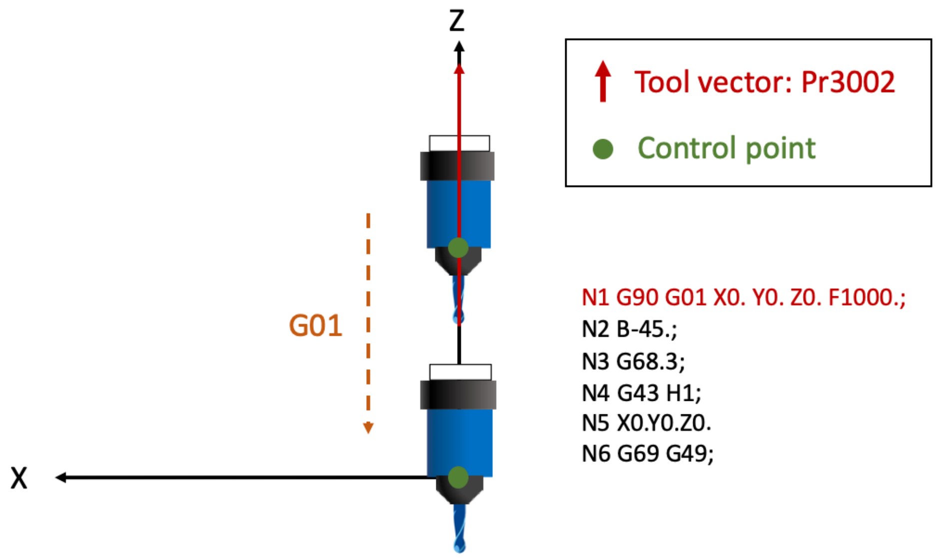


Fig.60

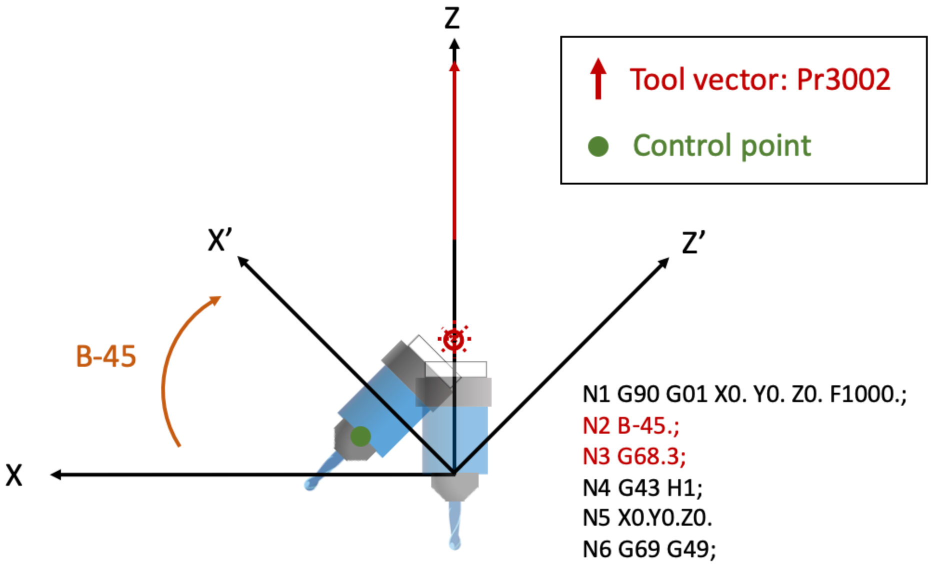

Z
Tool vector: Pr3002
Control point
X'
Z'
B-45
X
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;
N4 G43 H1;
N5 X0.Y0.Z0.
N6 G69 G49;

Fig.61

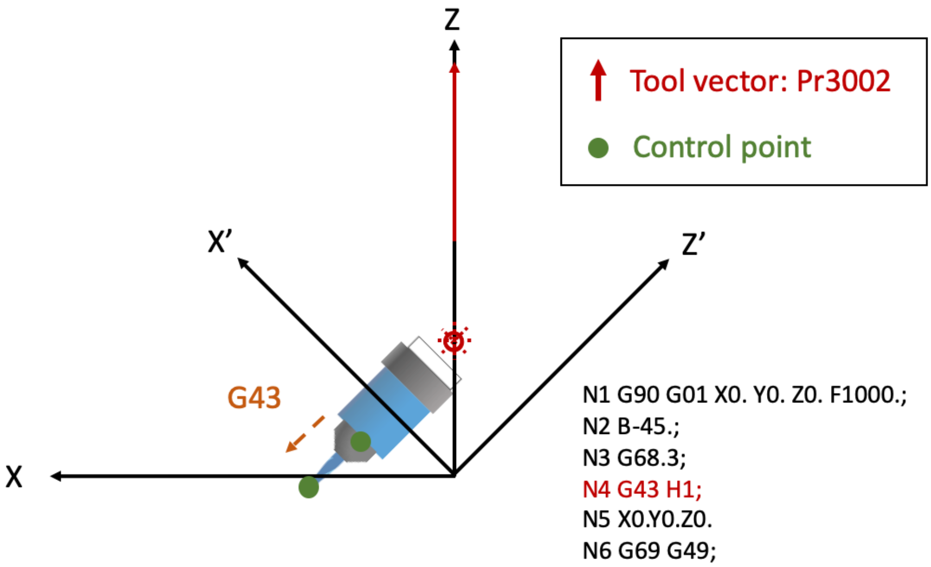

Z
Tool vector: Pr3002
Control point
X'
Z'
G43
X
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;
N4 G43 H1;
N5 X0.Y0.Z0.
N6 G69 G49;

Fig.62

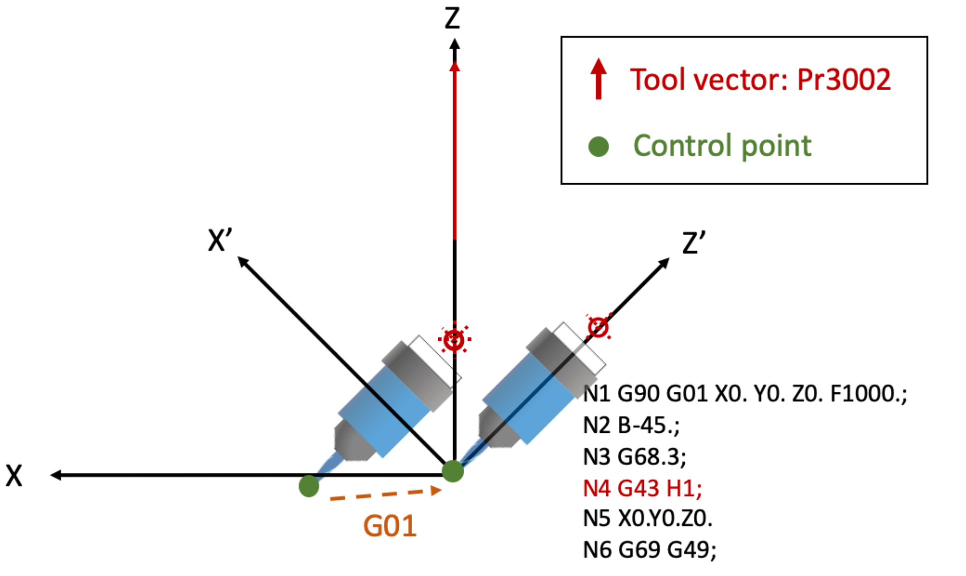
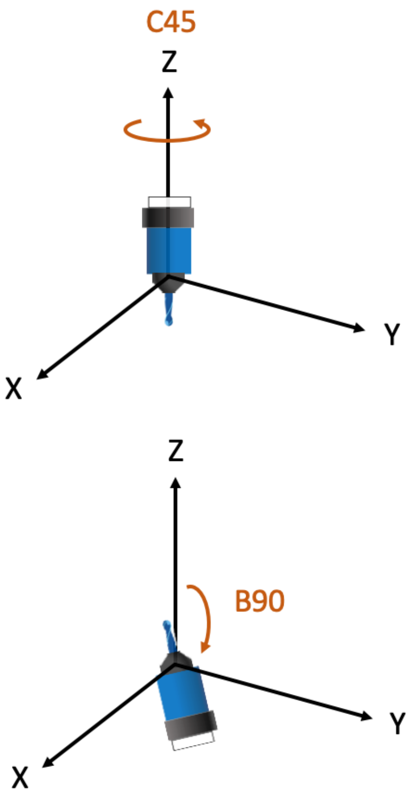


Fig.63

4.6.6 (2)

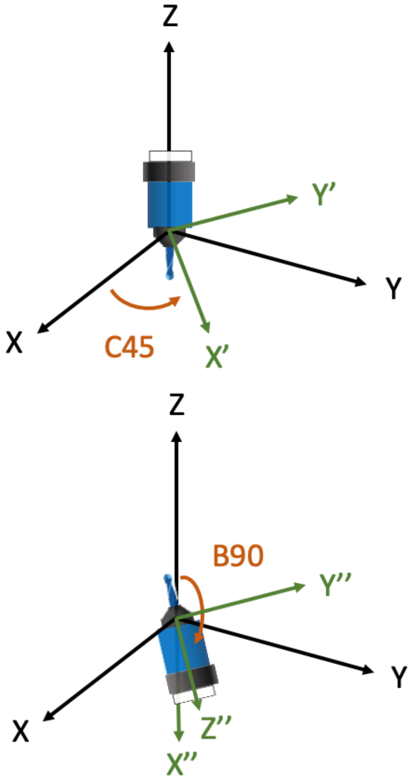
NC program and Fig.64~66 are used to explain the motions within Tilted Working Plane when G68.3 is enabled.

```
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;
```



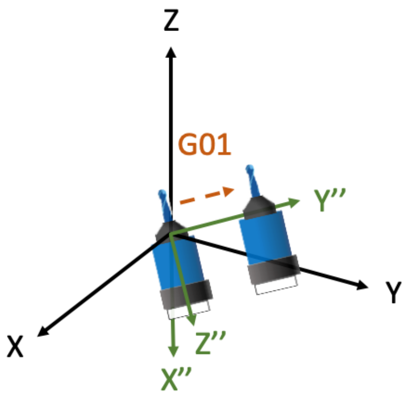

```
N1 G54 G90 G00 B0. C0.;
N2 C45.;
N3 B90.;
N4 G68.3 P1 X0. Y0. Z0.;
N5 G01 Y10.;
N6 G69;
N7 G00 X0. Y0. Z0. B0. C0.;
N8 M30;
```

Fig.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

4.6.7 (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;
```

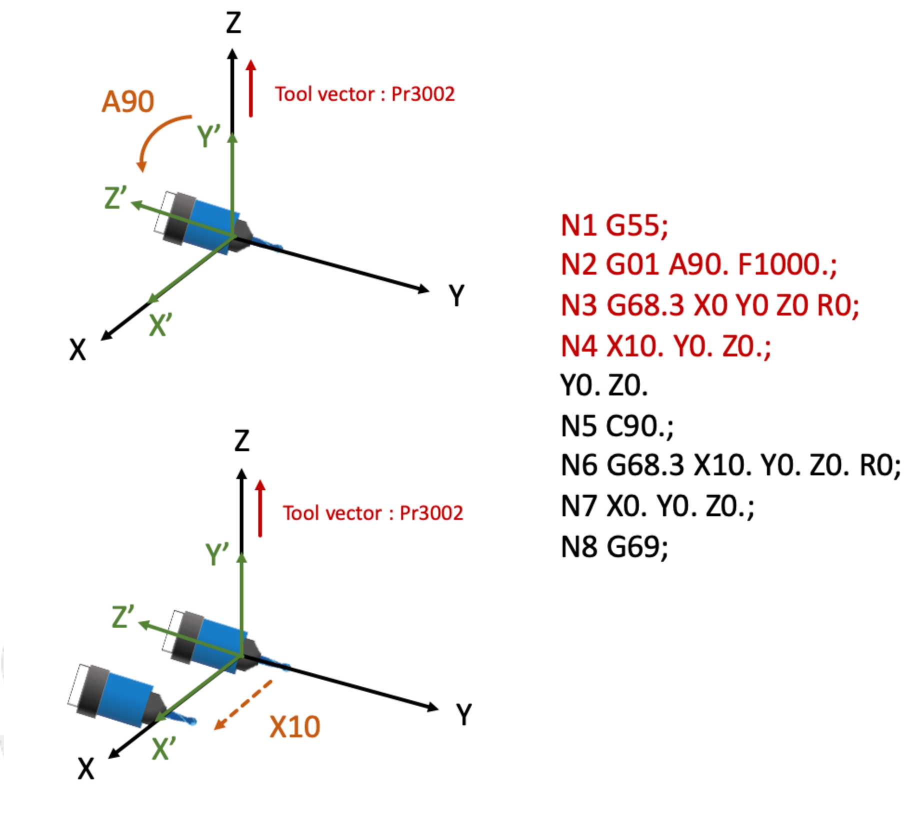
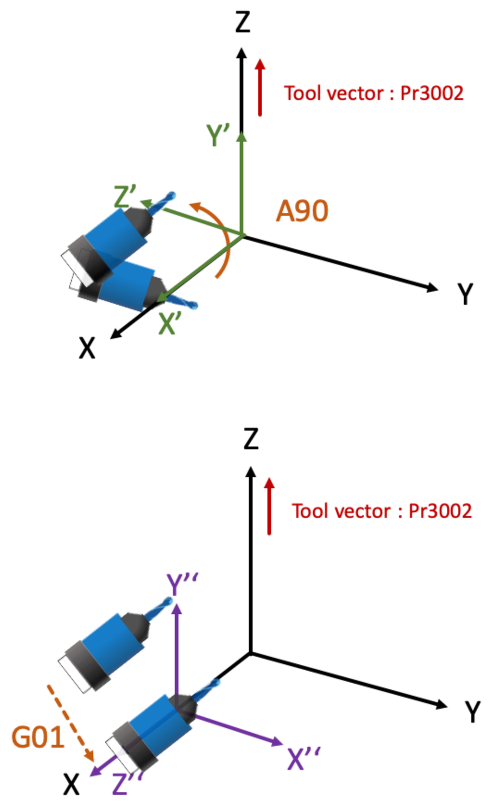


Fig.67




```
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

4.7 4.7 Program Example of Tilted Working Plane Machining

4.7.1 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.

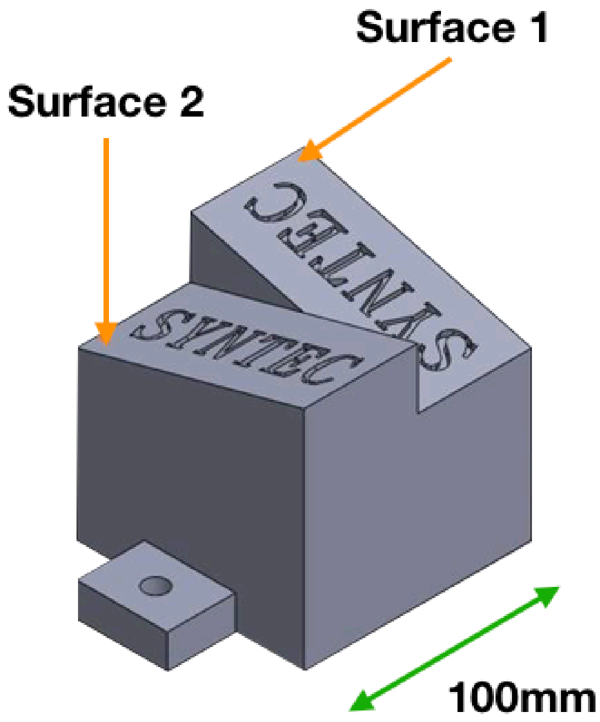


Fig.69

4.7.2 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
.........
```

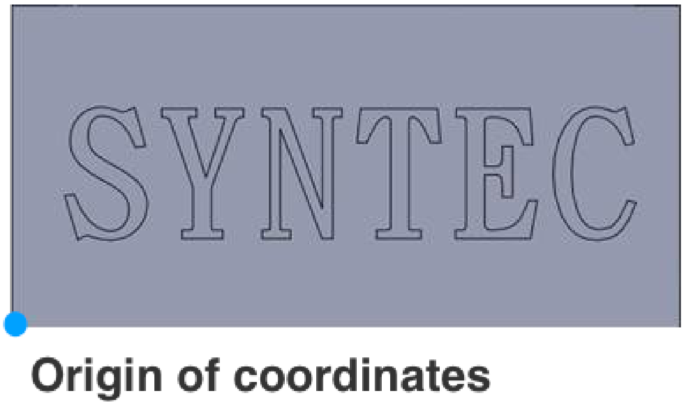


Fig.70

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
.........
```

4.7.3 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.

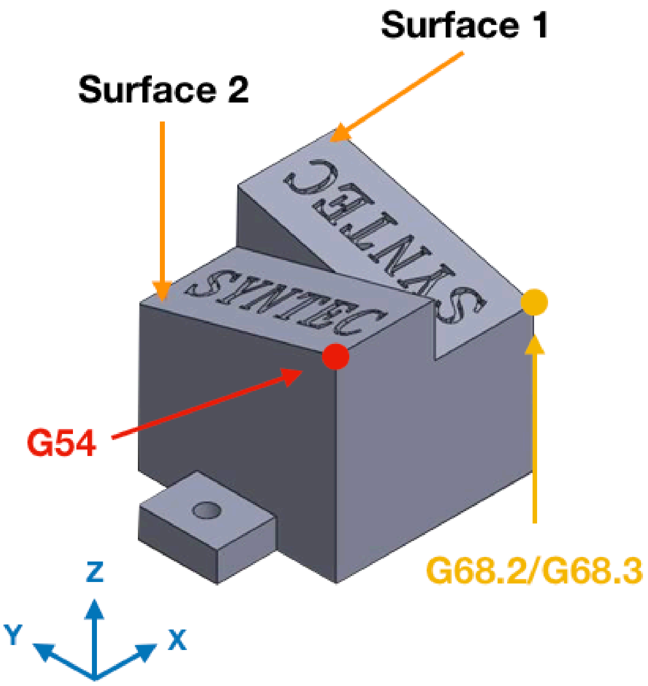


Fig.71

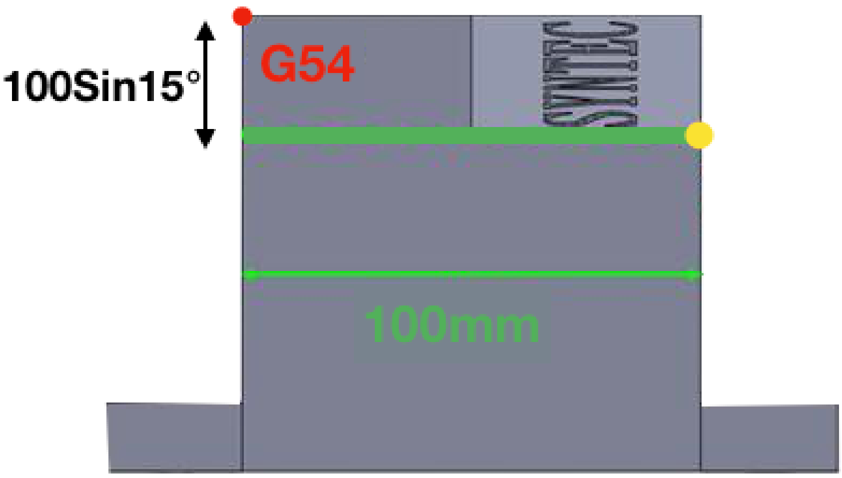


Fig.72

4.7.4 **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.

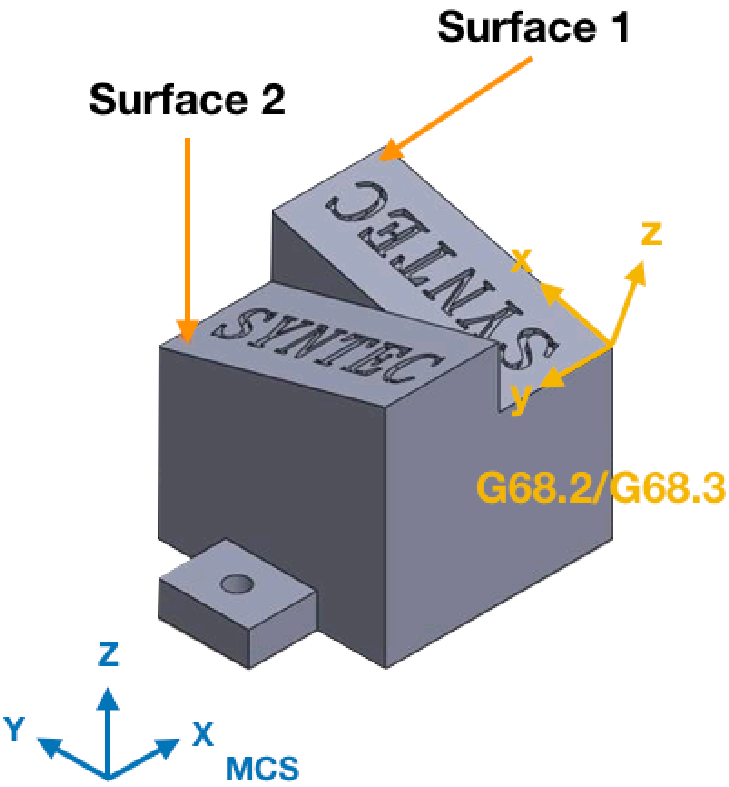


Fig.73

4.7.5 **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.

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.

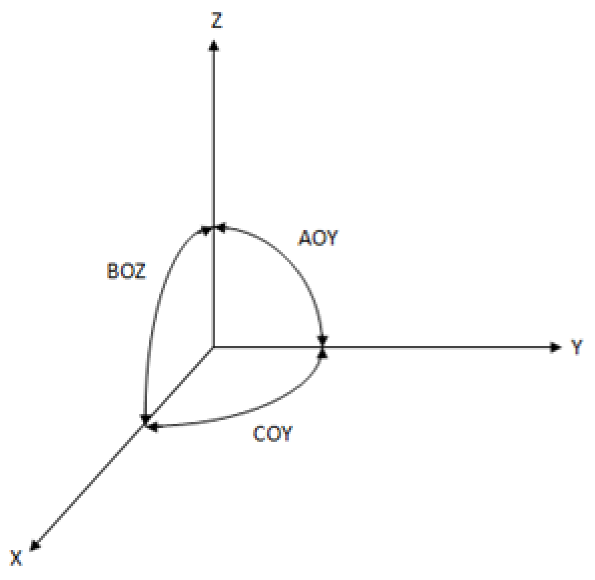


Fig.55

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.

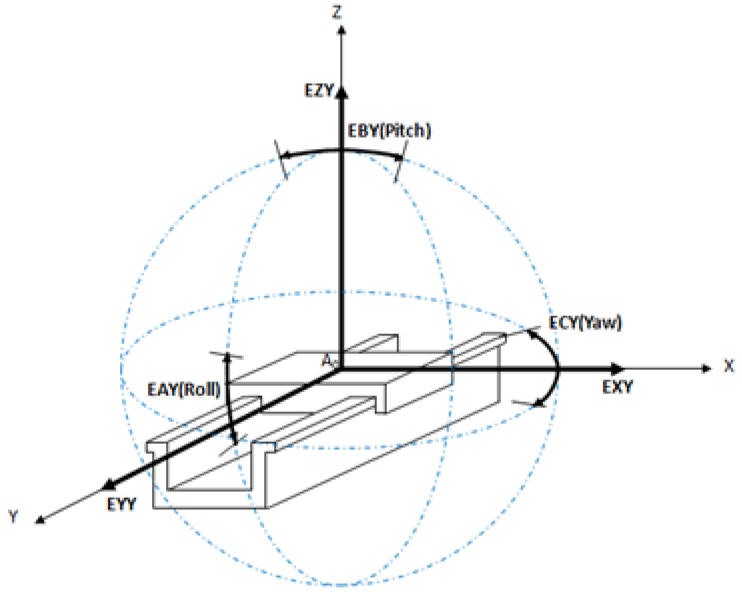


Fig.56

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.

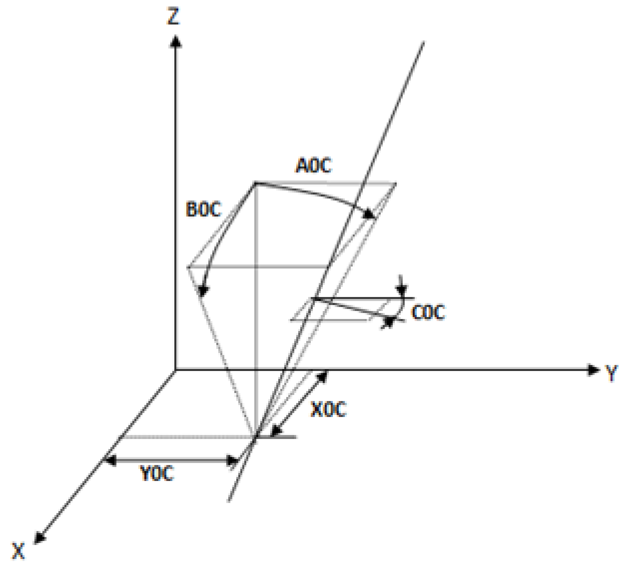


Fig.57

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.

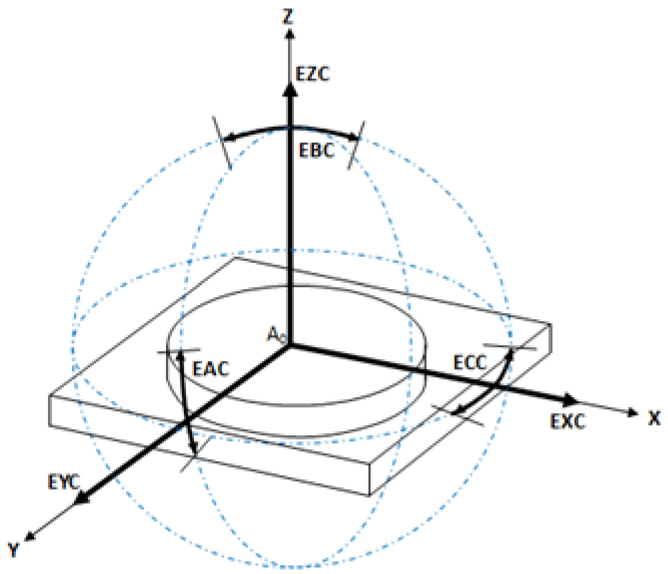


Fig.58

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

30 42	1st y-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
30 43	1st z-component of Offset from tool holder to first rotation axis	[-999999999,999999999]	BLU	0	Reset
Table Type for 4-Axis Machine					
30 34	1st x-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
30 35	1st y-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset
30 36	1st z-component of Offset from machine to first rotation axis	[-999999999,999999999]	BLU	0	Reset

6 6. Related Alarms

6.1 COR Alarm

Alarm ID	**COR–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**	**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 Woodworking label 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.		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Woodworking label path to do process operation. 6. Set Pr3802 to 0.		

Alarm ID	**COR-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.		

Alarm ID	**COR-107**	**Alarm Title**	**G5.1/G05 command format error**

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**	**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.		
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**	**Alarm Title**	**Prohibit G05 in tool tip control mode**
Description	Turn on G05 high-speed high-precision mode in the RTCP/STCP mode.		
Possible Cause	In the RTCP/STCP mode, turn on the G05 high-speed high-precision mode with commands, such as G05 P10000.		

Alarm ID	COR-140	Alarm Title	Prohibit G05 in tool tip control mode

Solution	Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-speed high-precision mode.
	If (1), remove the command to turn on the G05 high-speed high-precision mode in the RTCP/STCP mode.
	If (2), turn off the RTCP/STCP mode before turning on the G05 high-speed high-precision mode.		

Alarm ID	COR-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	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	Alarm Title	2^{nd} rotation axis entering illegal range
Description	2^{nd} rotation axis entering illegal range		

Alarm ID	COR-152	Alarm Title	2nd rotation axis entering illegal range
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2nd 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	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	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	Alarm Title	5-axis tool direction error
Description	5-axis tool direction error.		
Possible Cause	5-axis tool direction (Pr3002) or the 1st and 2nd 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 2nd rotation axis is parallel to the Spindle in the Spindle type, or the 1st rotation axis is parallel to the Spindle in the workbench type.		

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

Alarm ID	COR-162	Alarm Title	4-axis RTCP configuration error
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	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	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	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.		
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, ...		

Alarm ID	**COR-165**	**Alarm Title**	**Multi-kinematic chain command not illegal.**
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**	**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**	**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.		
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		

6.2 OP Alarm

Alarm ID	**OP-032**	**Alarm Title**	**Mechanism type configuration conflicted**
Description	Pr3201 Machine Type setting conflicted.		

<table>
<tr><th>Alarm ID</th><th>OP-032</th><th>Alarm Title</th><th>Mechanism type configuration conflicted</th></tr>
<tr><td>Possible Cause</td><td colspan="3">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.</td></tr>
<tr><td>Solution</td><td colspan="3">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).</td></tr>
</table>

6.3 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.

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. 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.

9 1. Introduction of 5-Axis Machine.

9.1 **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.

9.1.1 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:

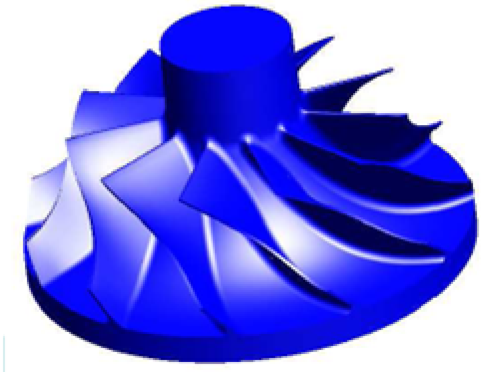


Fig.1

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.

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.

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).

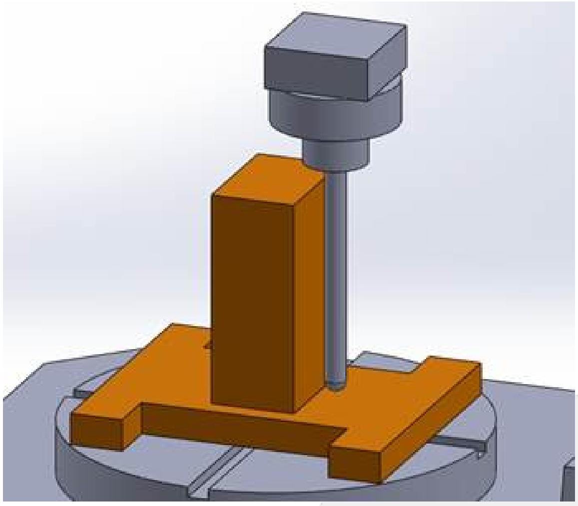


Fig.2

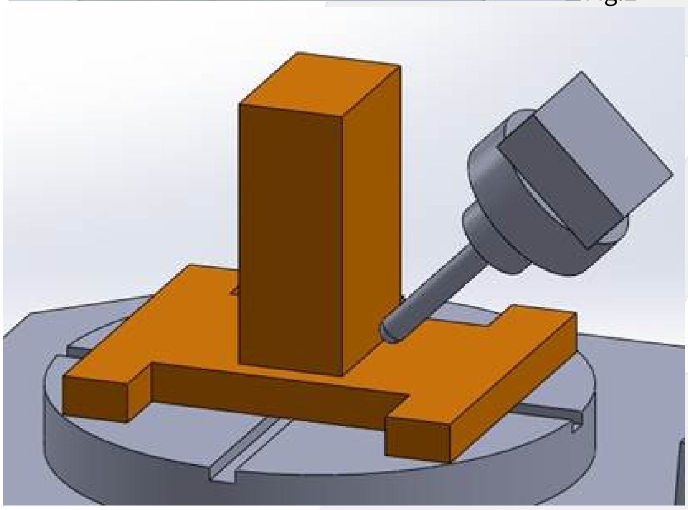


Fig.3

9.1.2 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

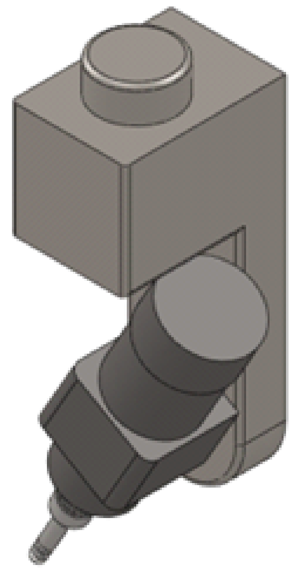
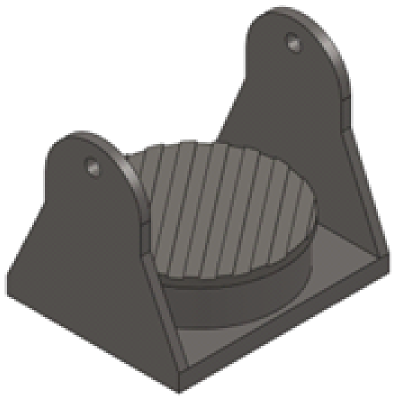
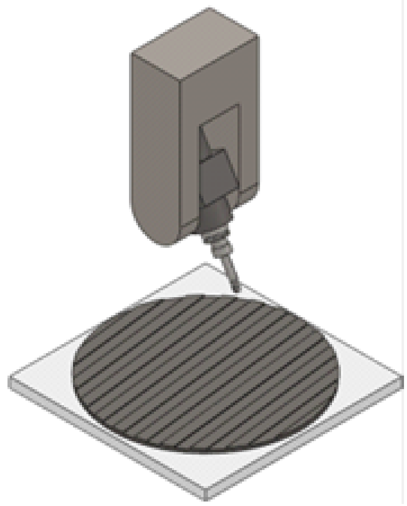
		
Spindle Type	Table Type	Mix Type

Fig.4

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.

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.

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

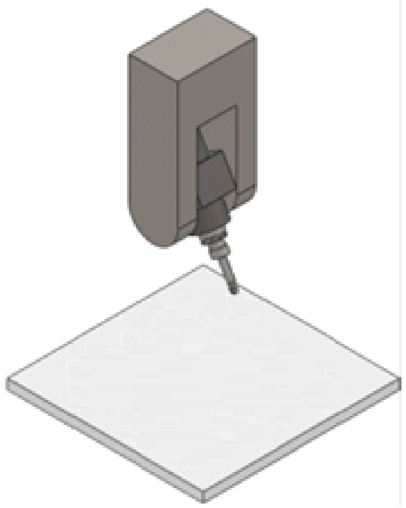
	
Single Spindle Type	Single Table Type

Fig.5

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.

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.

9.1.3 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：

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.

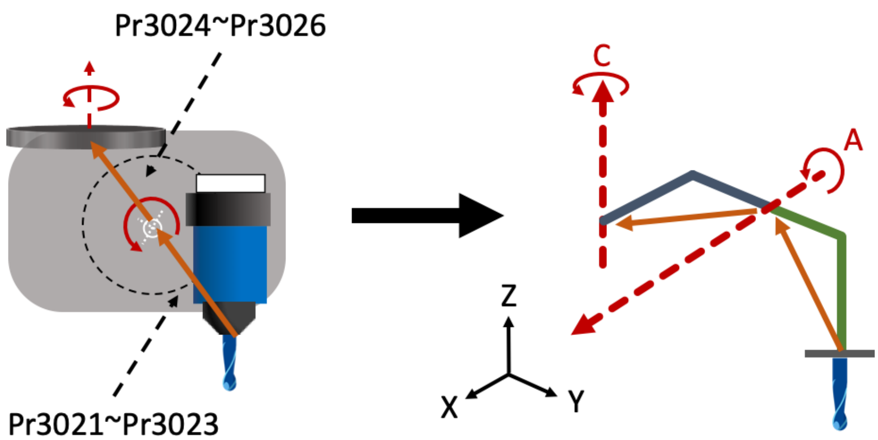


Fig.6

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.

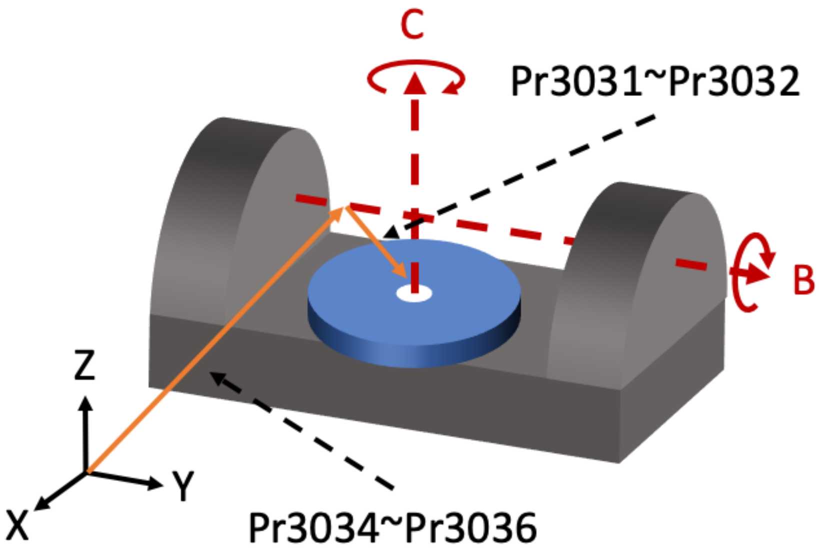


Fig.7

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.

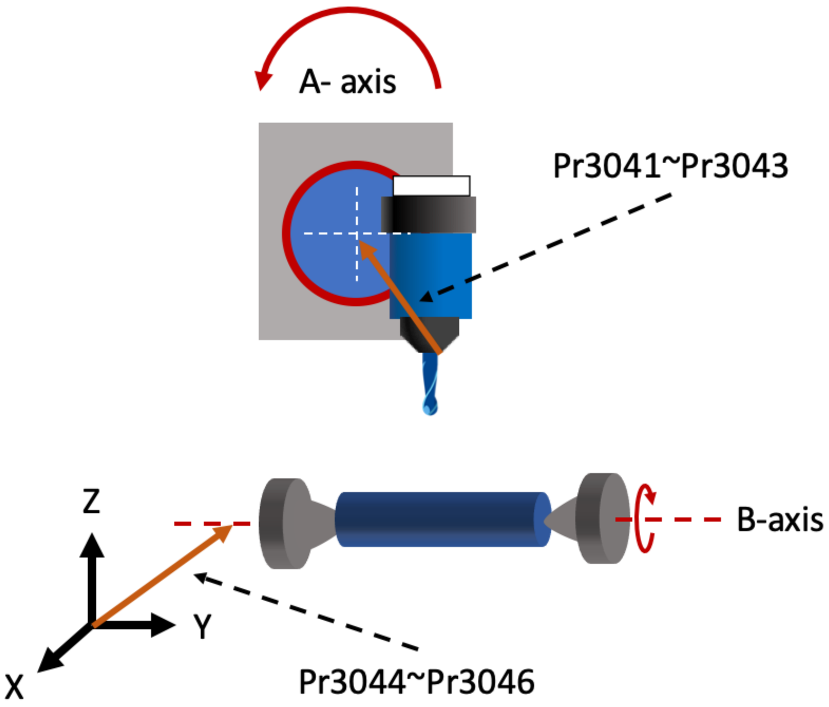


Fig.8

9.1.4 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.

No	Descriptions	Range	Unit	Default	Effective
3001	*1st Organization for five axis machine	[0,5]	-	0	Restart

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

No	Descriptions	Range	Unit	Default	Effective
3002	1st Direction of Tool	[0,3]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3003	1st Incline Angle of Direction of Tool(RA)	[0,360000]	BLU	0	Reset

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.

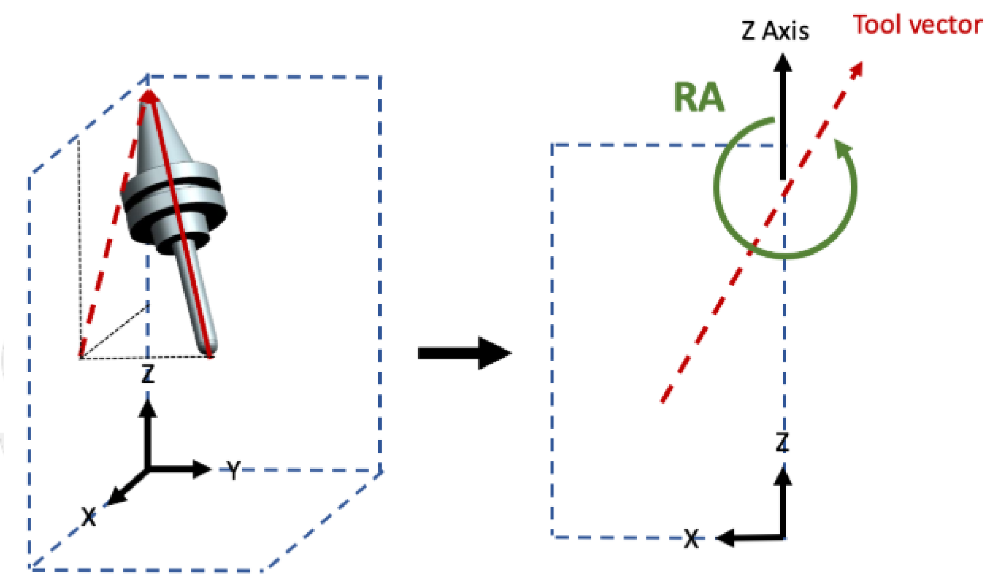


Figure 9

No	Descriptions	Range	Unit	Default	Effective
3004	1st Incline Angle of Direction of Tool(RB)	[0,360000]	BLU	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3005	1st first rotation axis	[0,3]	-	0	Reset
3006	1st second rotation axis	[0,3]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3007	1st rotation direction of first rotation axis	[0,2]	-	0	Reset
3008	1st rotation direction of second rotation axis	[0,2]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3009	1st starting point of rotation angle of first rotation axis	[0,360000]	BLU	0	Reset
3010	1st terminal point of rotation angle of first rotation axis	[0,360000]	BLU	0	Reset

3011	1st starting point of rotation angle of second rotation axis	[0,360000]	BLU	0	Reset
3012	1st terminal point of rotation angle of second rotation axis	[0,360000]	BLU	0	Reset

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.

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.

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.

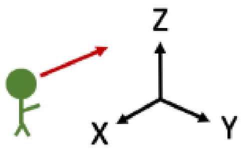


figure 13

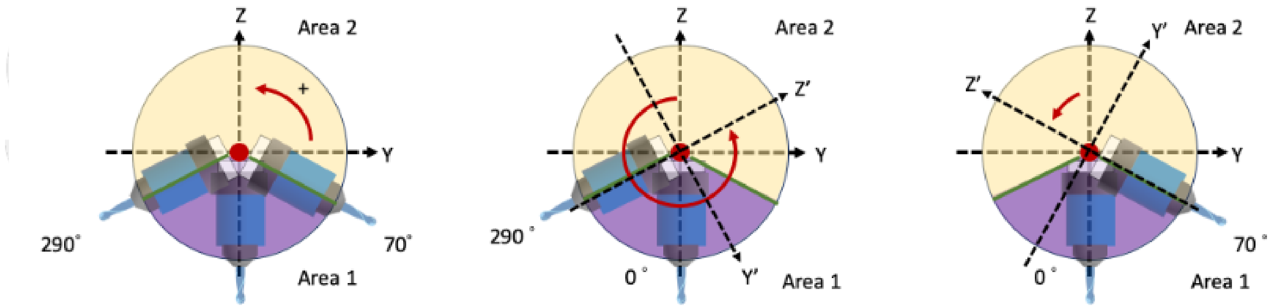


figure 14

No	Descriptions	Range	Unit	Default	Effective

3013	1st Tool Holder Offset	[0,999999999]	BLU	0	Reset

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.

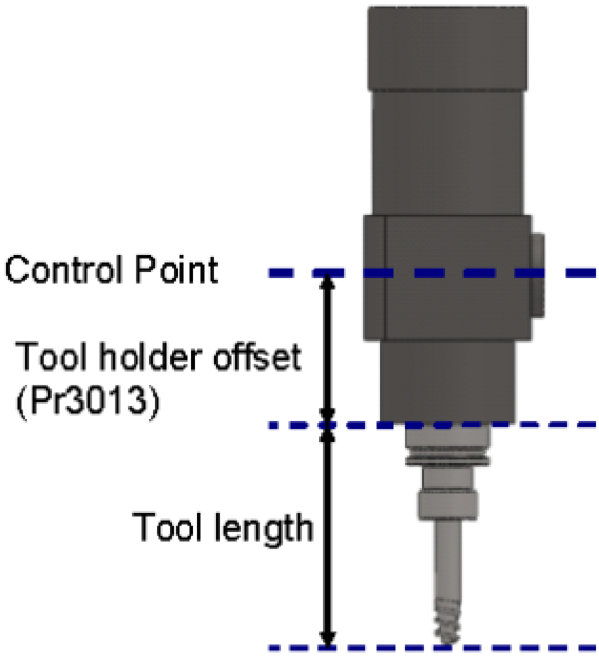


Figure 15

No	Descriptions	Range	Unit	Details	Effective
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.	Reset

No	Descriptions	Range	Unit	Default	Effective
3015	1st A-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset

3016	1st B-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset
3017	1st C-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset
3018	1st A-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset
3019	1st B-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset
3020	1st C-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset

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.

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

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

Figure 16

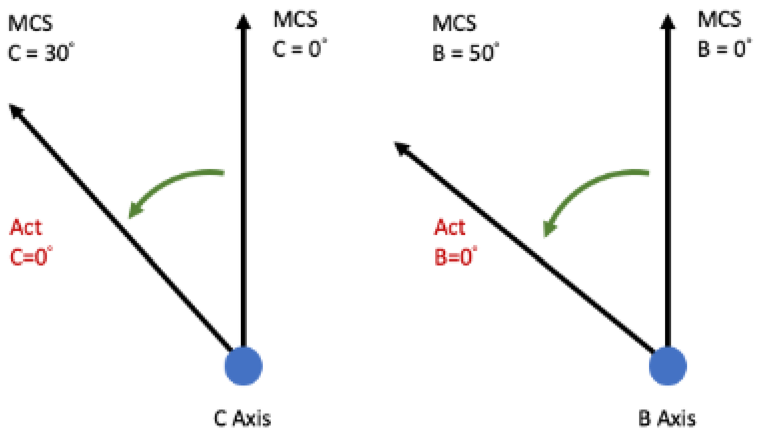


Figure 17.

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

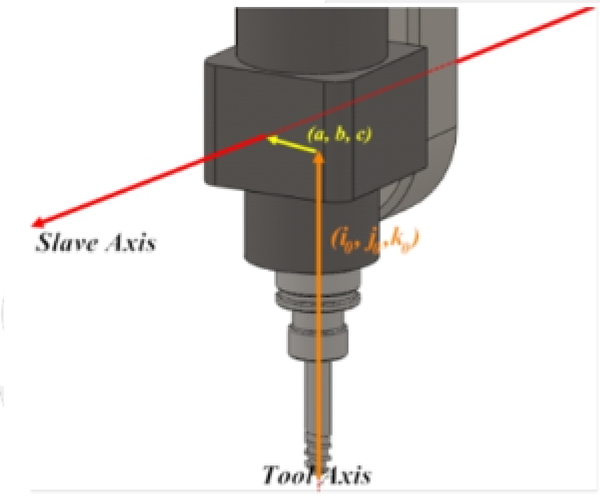


Figure 19

No	Descriptions	Range	Unit	Default	Effective

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

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

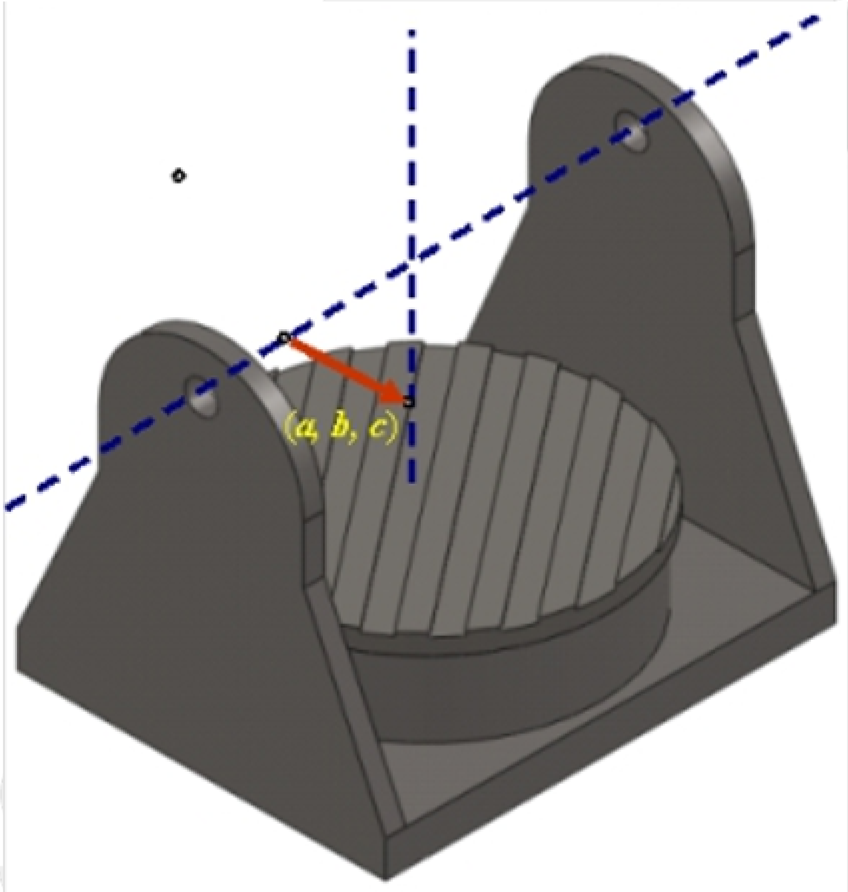


Figure 21

No	Descriptions	Range	Unit	Default	Effective
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

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)

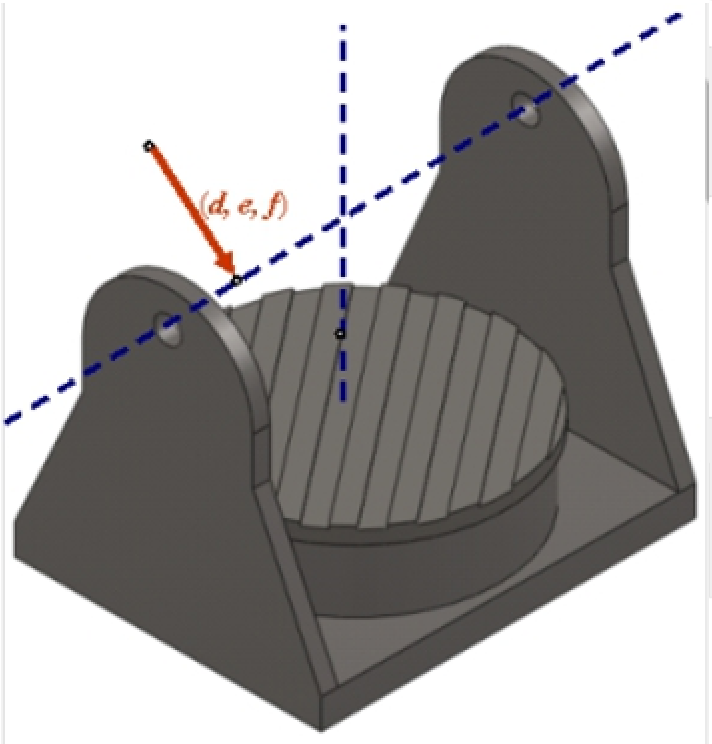


Figure 22

No	Descriptions	Range	Unit	Default	Effective

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

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

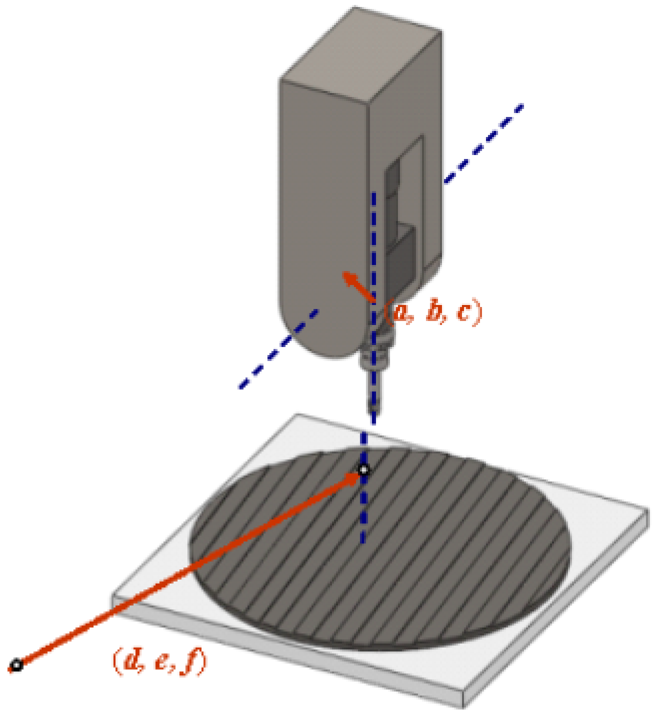


Figure 24

No	Descriptions	Range	Unit	Default	Effective
3051	Enable smooth RTCP function (0:No;1:Yes)	[0,1]	-	0	Reset

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.

No	Descriptions	Range	Unit	Default	Effective
3052	First rotation axis smoothness tolerance (0.001deg)	[1, 360000]	0.001 deg	500	Reset
3053	Second rotation axis smoothness tolerance (0.001deg)	[1, 360000]	0.001 deg	500	Reset

No	Descriptions	Range	Unit	Default	Effective

3054	1st RTCP interpolation mode (0: Five axis simultaneously; 1: Tool vector)	[0,1]	-	0	Reset

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.

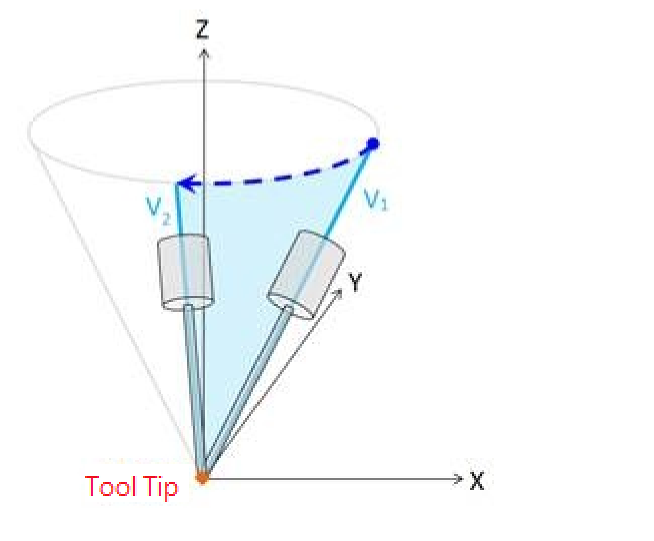
Note

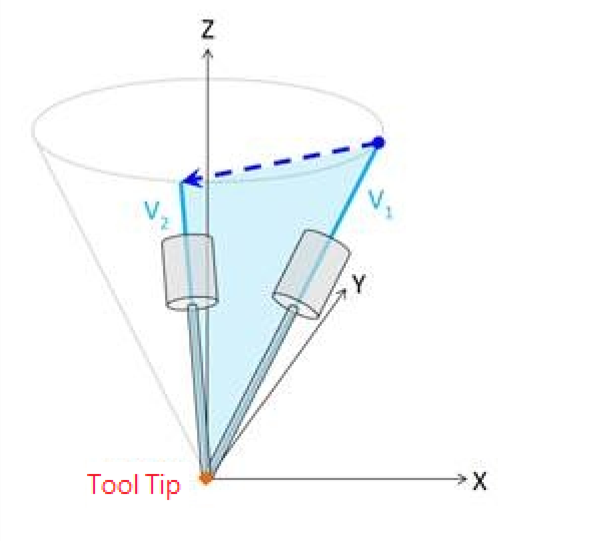
1. Tool vector interpolation mode is only valid for cutting G codes, such as G01, G02, G03, G02.4, G03.4. Axial movement generated from other G codes, such as G00, will not be affected by this parameter.
2. Under Pr3054 = 1, NC blocks will be interpreted as tool vectors.
   Therefore, the axes might 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.

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 Vector**
Tool posture changes during movement	**Tool posture stays on the same plane**



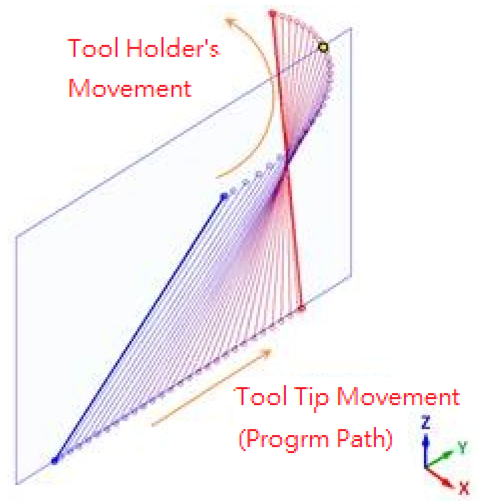


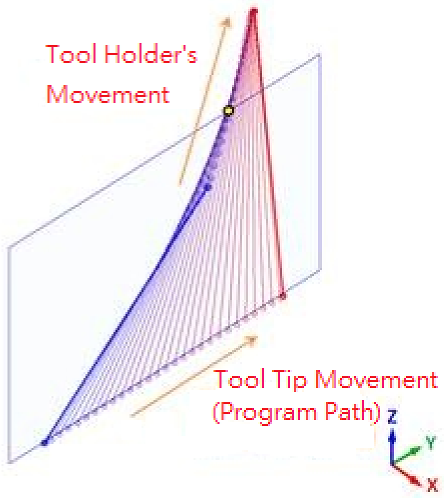
Because there is no command for B axis, so B axis stays still; the tool posture is slightly off from the plane composed of V_1 and V_2.	To stay on the plane composed of V_1 and V_2 during movement, B axis rotates without command and eventually returns to its original angle.

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.

No	Descriptions	Range	Unit	Default	Effective
3055	Basic position of table coordinate system	[0,1]	-	0	Reset

Value

0: RTCP start position

1: zero position of workpiece coordinate

Description

- Table coordinate system：
  Table coordinate system is a workpiece coordinate system exclusive to RTCP and only exist during RTCP is enabled.
  When RTCP is enabled, table coordinate will be established on the table.
  During RTCP is enabled, the commands in NC program will be interpreted as the commands of table coordinate.
- Rotation of the table coordinate system：
  When RTCP is enabled, zero position of rotary axis in table coordinate will be fixed according to Pr3055.

Pr3055 = 0 :

Zero position of rotary axis in table coordinate will be fixed on "current machine coordinate of rotary axis".

Pr3055 = 1 :

Zero position of rotary axis in table coordinate will be fixed on "zero position of rotary axis in workpiece coordinate", regardless of current angle of rotary axis.

During RTCP is enabled, the table coordinate will rotate along with the table.

- Difference in application:

If Pr3055 is 0, user should locate rotary axis to the specific angle before enabling RTCP.

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.

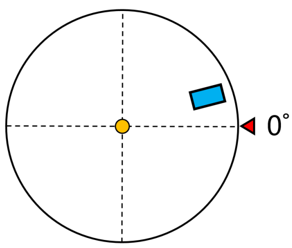
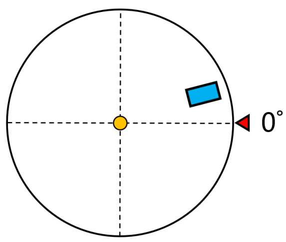
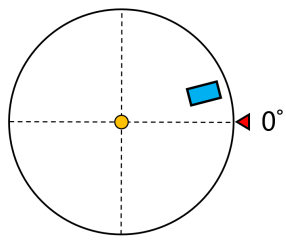
Notice

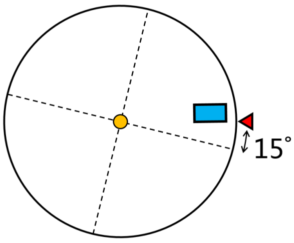
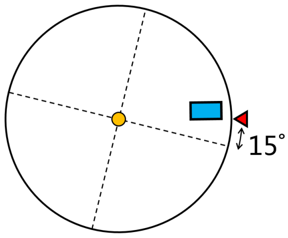
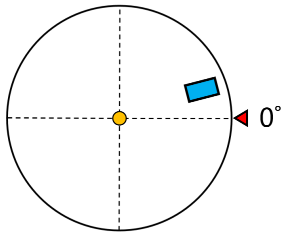
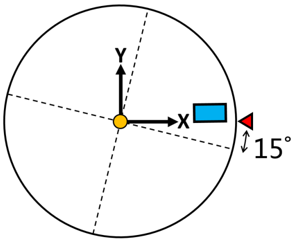
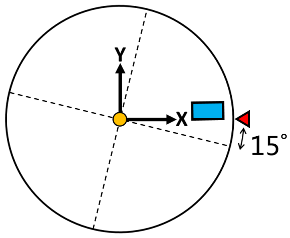
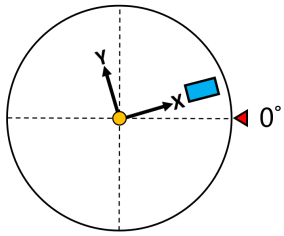
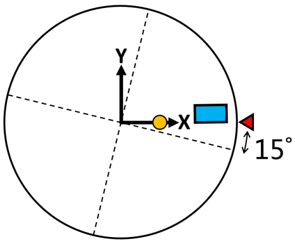
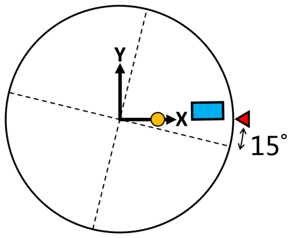
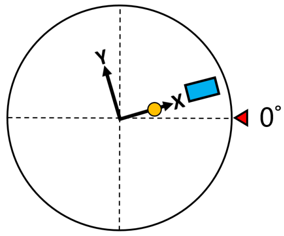
- If the parameter is modified when RTCP is enabled, it will take effect when RTCP being enabled next time.
- This parameter is only valid for the rotary axes on the table side.

Example

- The purpose of following examples is making the table coordinate to align to the angle of the workpiece (that is C15. in machine coordinate).
- If Pr3055 is 0, like example A, user should locate C axis to C15. in advance before enabling RTCP.
  If Pr3055 is 1, like example B and C, user only need to fill C15. in G54 offset.

> **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	无	C+15.	C+15.
NC Command	(locate C axis, then enable RTCP) G90 G49 G54 N1 C15. // C positioning N2 G43.4 H1 Z0. // RTCP ON N3 X10. Y0. Z0. // tool tip moves	(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	C15. // C positioning 	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 
Note	The machine coordinate of C axis is 15. when RTCP is enabled. Therefore, the zero position of C axis in table coordinate is at C15. in machine coordinate.	Regardless of the angle of C axis when RTCP is enabled, the zero position of C axis in table coordinate is the same as zero position of C axis in workpiece coordinate.	

9.2 1.1 5-Axis Machine Features.

9.2.1 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:

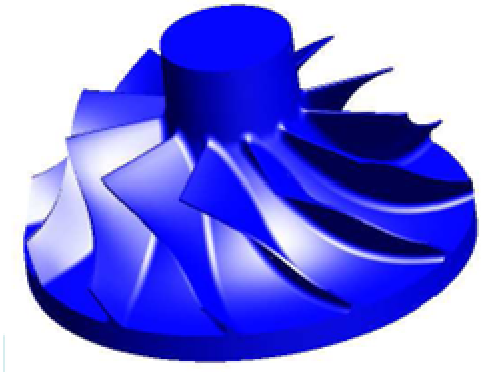


Fig.1

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.

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.

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).

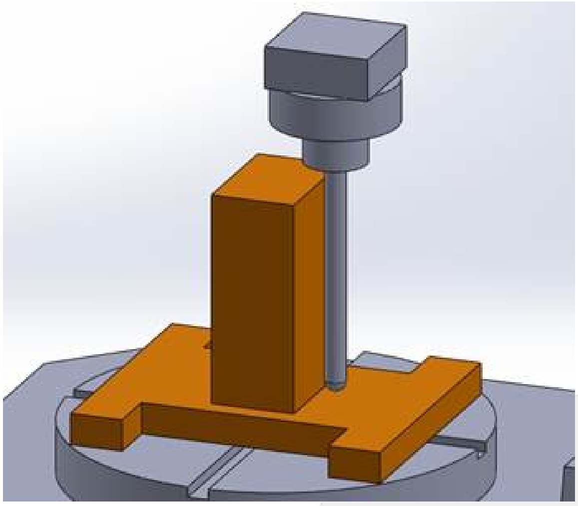


Fig.2

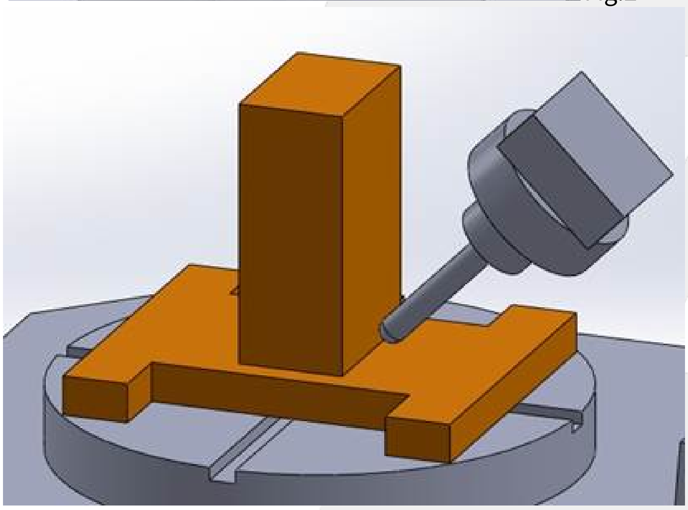


Fig.3

9.3 1.2 Machine Type.

9.3.1 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

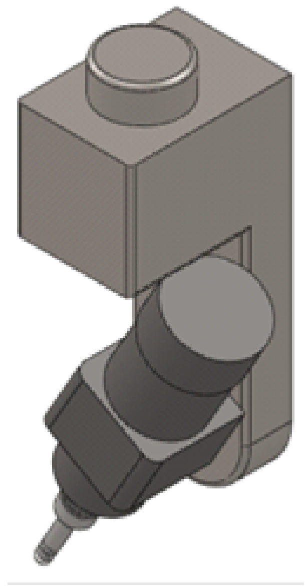
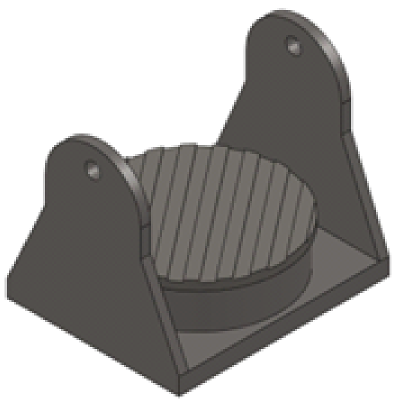
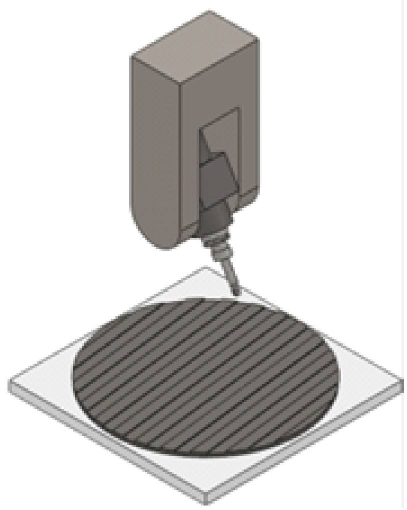
		
Spindle Type	Table Type	Mix Type

Fig.4

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.

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.

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

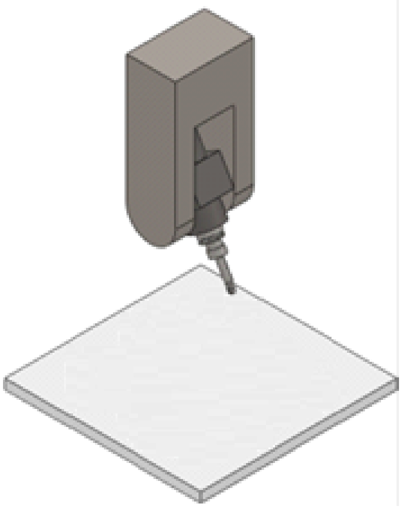
	
Single Spindle Type	Single Table Type

Fig.5

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.

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.

9.4 1.3 Definitions of Rotary Axis.

9.4.1 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：

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.

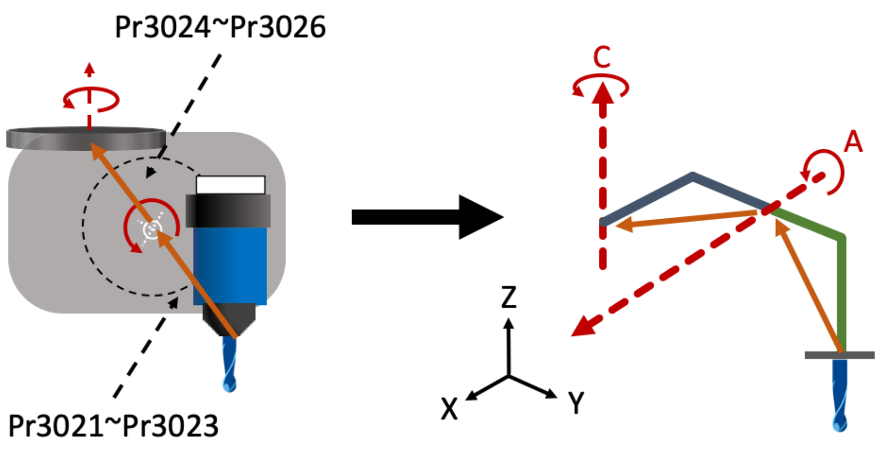


Fig.6

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.

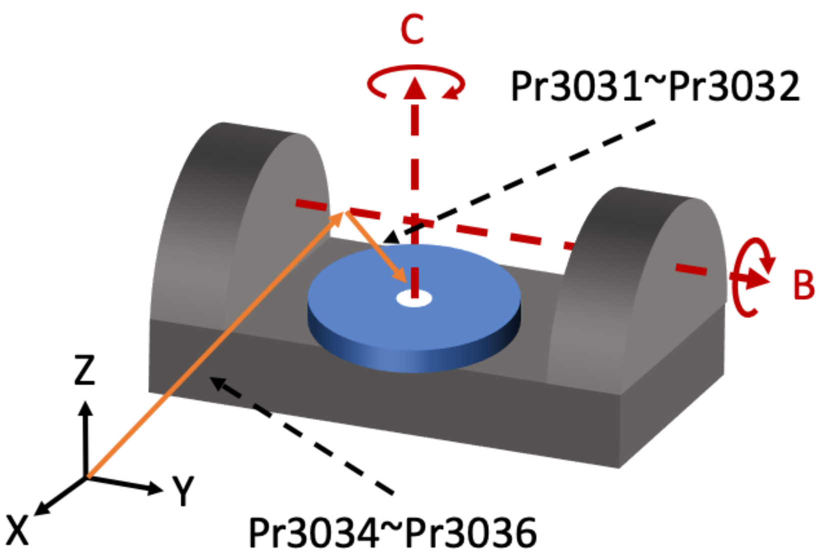


Fig.7

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.

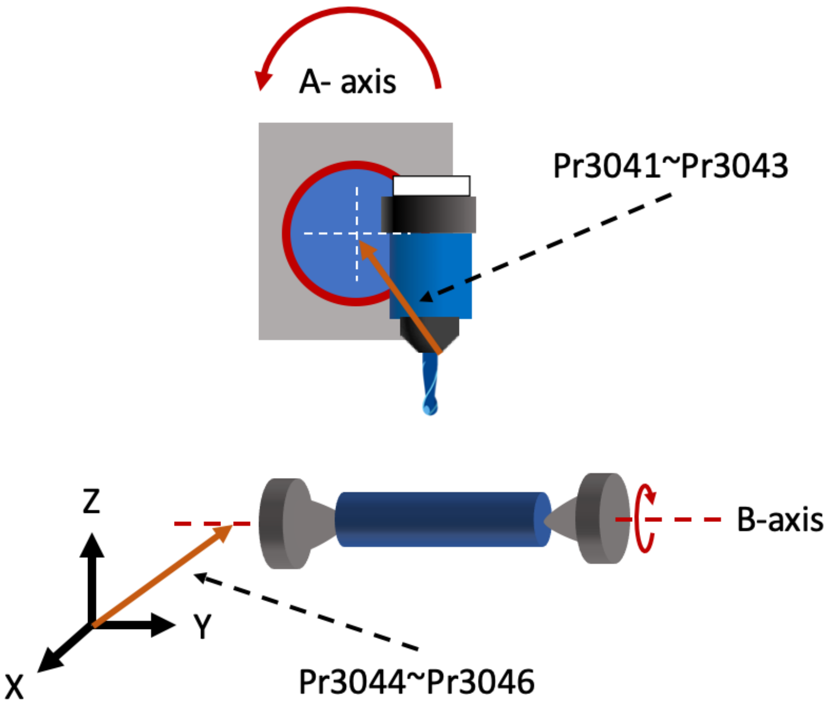


Fig.8

9.5 1.4 Parameter Descriptions.

9.5.1 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.

No	Descriptions	Range	Unit	Default	Effective
3001	*1st Organization for five axis machine	[0,5]	-	0	Restart

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

No	Descriptions	Range	Unit	Default	Effective

3002	1st Direction of Tool	[0,3]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3003	1st Incline Angle of Direction of Tool(RA)	[0,360000]	BLU	0	Reset

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.

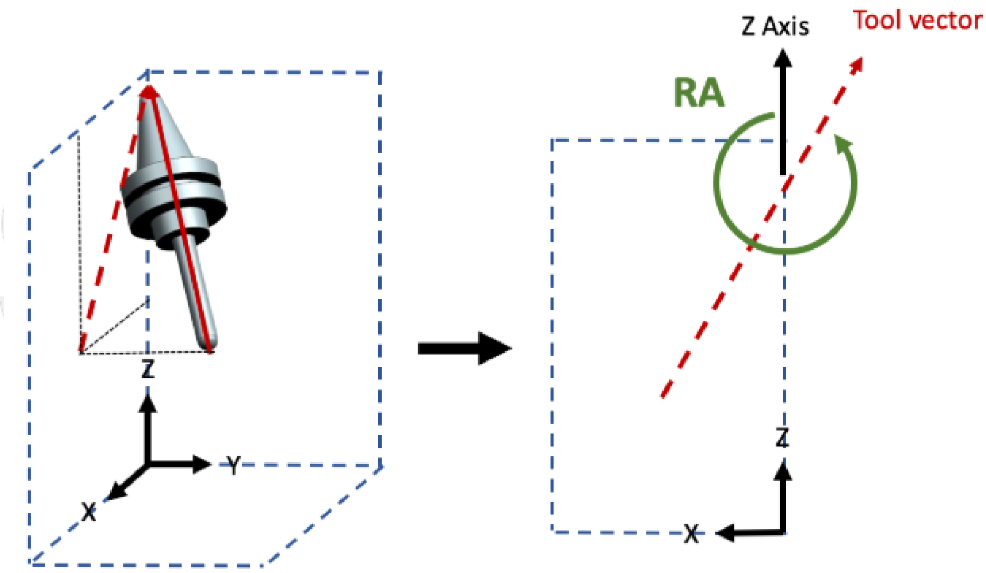


Figure 9

No	Descriptions	Range	Unit	Default	Effective
3004	1st Incline Angle of Direction of Tool(RB)	[0,360000]	BLU	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3005	1st first rotation axis	[0,3]	-	0	Reset
3006	1st second rotation axis	[0,3]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3007	1st rotation direction of first rotation axis	[0,2]	-	0	Reset
3008	1st rotation direction of second rotation axis	[0,2]	-	0	Reset

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

No	Descriptions	Range	Unit	Default	Effective
3009	1st starting point of rotation angle of first rotation axis	[0,360000]	BLU	0	Reset
3010	1st terminal point of rotation angle of first rotation axis	[0,360000]	BLU	0	Reset

3011	1st starting point of rotation angle of second rotation axis	[0,360000]	BLU	0	Reset
3012	1st terminal point of rotation angle of second rotation axis	[0,360000]	BLU	0	Reset

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.

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.

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.

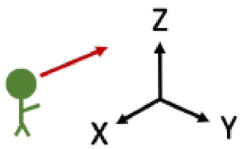


figure 13

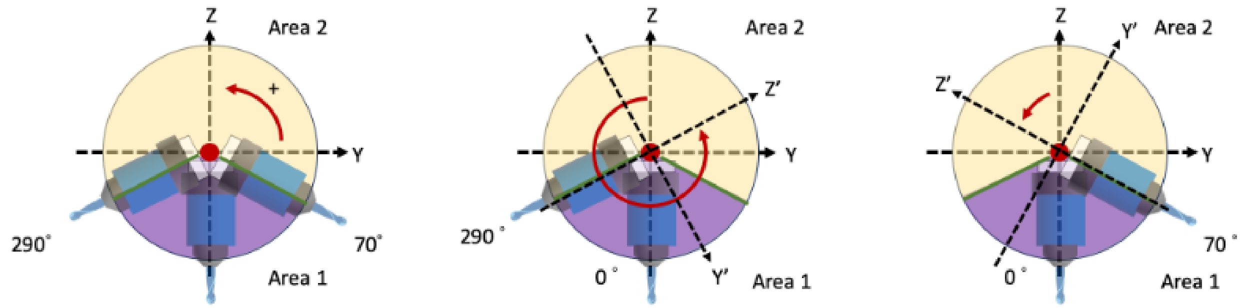


figure 14

No	Descriptions	Range	Unit	Default	Effective

3013	1st Tool Holder Offset	[0,999999999]	BLU	0	Reset

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.

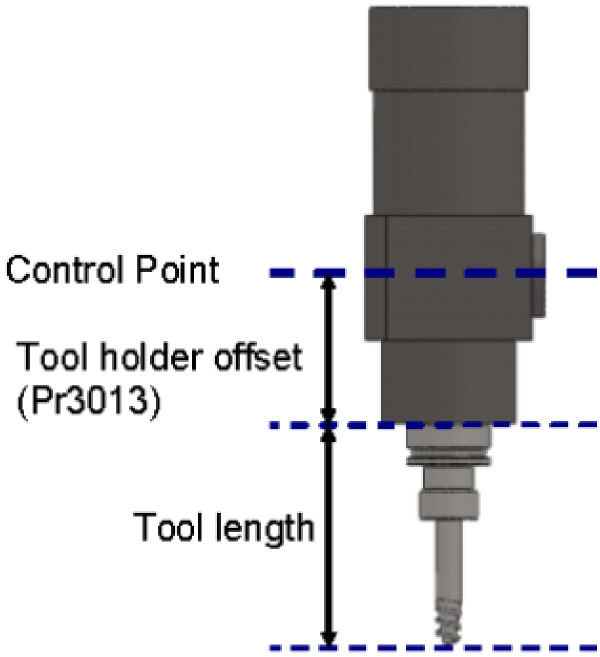


Figure 15

No	Descriptions	Range	Unit	Details	Effective
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.	Reset

No	Descriptions	Range	Unit	Default	Effective
3015	1st A-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset

3016	1st B-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset
3017	1st C-component of offset for first rotation axis	[-360000,360000]	BLU	0	Reset
3018	1st A-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset
3019	1st B-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset
3020	1st C-component of offset for second rotation axis	[-360000,360000]	BLU	0	Reset

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.

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

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

Figure 16

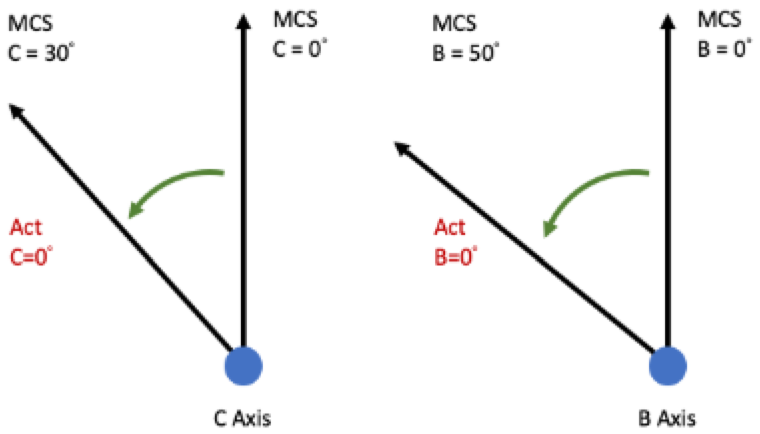


Figure 17.

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

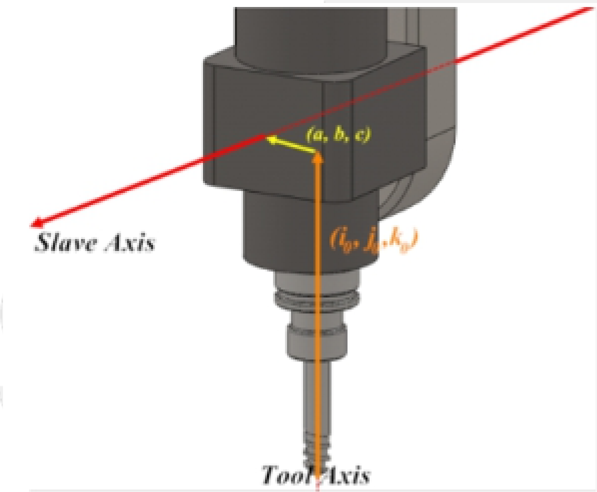


Figure 19

No	Descriptions	Range	Unit	Default	Effective

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

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

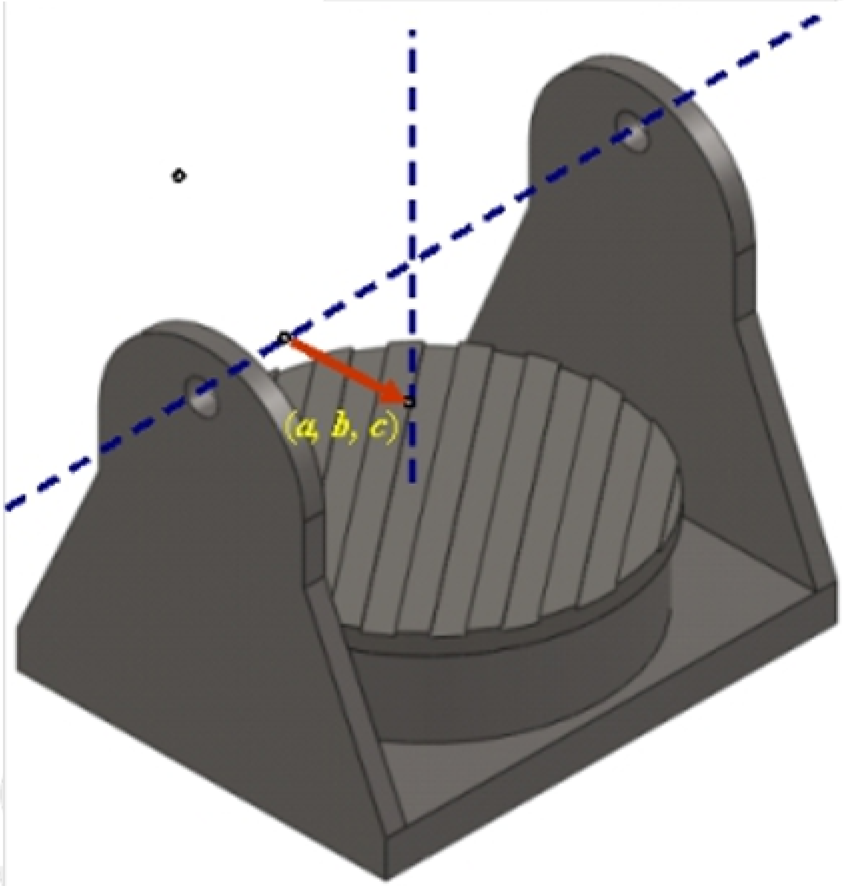


Figure 21

No	Descriptions	Range	Unit	Default	Effective
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

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)

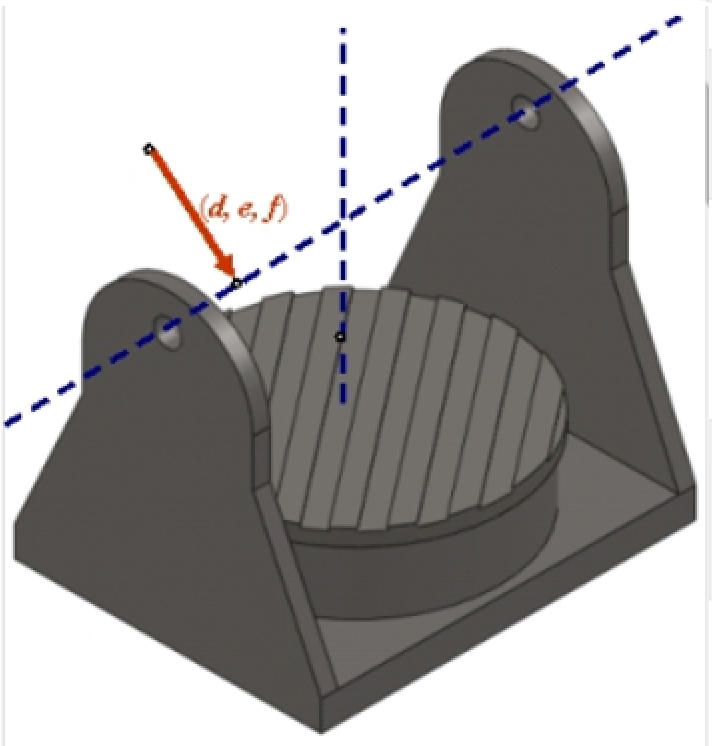


Figure 22

No	Descriptions	Range	Unit	Default	Effective

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

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

No	Descriptions	Range	Unit	Default	Effective
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

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)

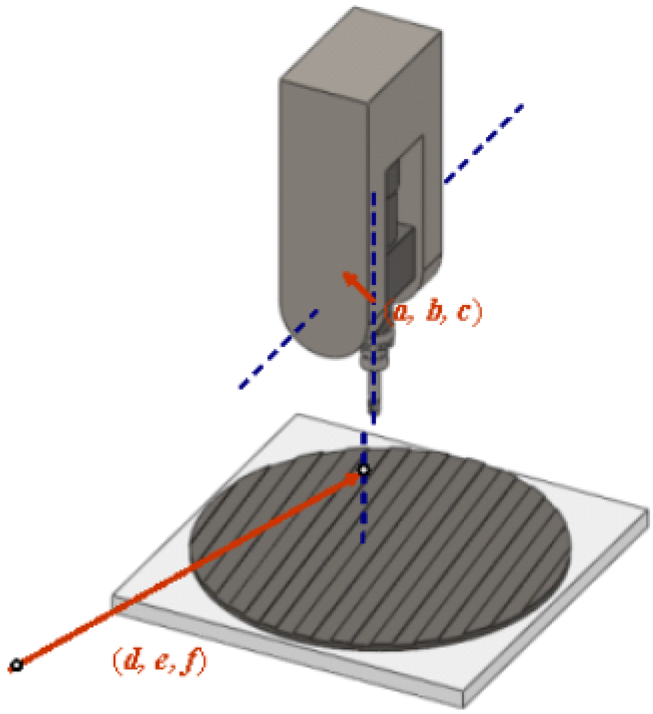


Figure 24

No	Descriptions	Range	Unit	Default	Effective
3051	Enable smooth RTCP function (0:No;1:Yes)	[0,1]	-	0	Reset

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.

No	Descriptions	Range	Unit	Default	Effective
3052	First rotation axis smoothness tolerance (0.001deg)	[1, 360000]	0.001 deg	500	Reset
3053	Second rotation axis smoothness tolerance (0.001deg)	[1, 360000]	0.001 deg	500	Reset

No	Descriptions	Range	Unit	Default	Effective

3054	1st RTCP interpolation mode (0: Five axis simultaneously; 1: Tool vector)	[0,1]	-	0	Reset

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.

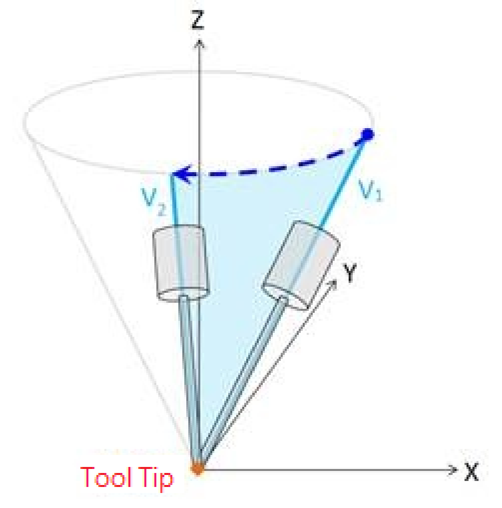
Note

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.

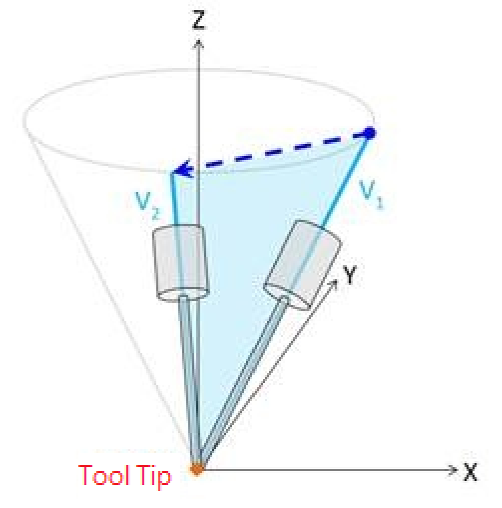
Example 1

```
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 Vector**
Tool posture changes during movement	**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.

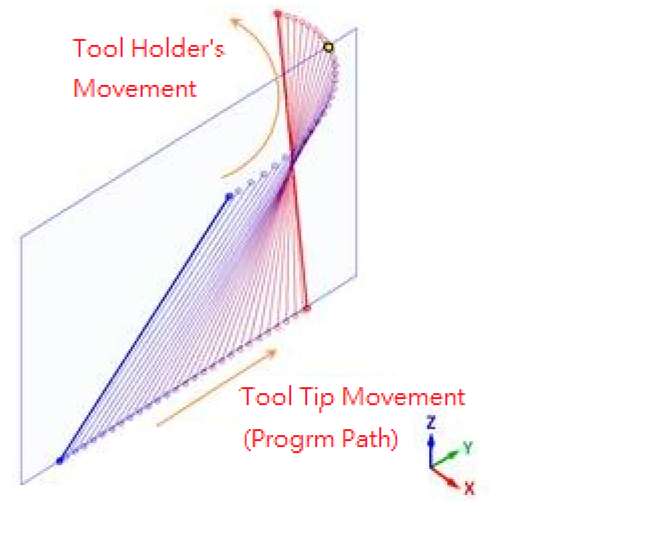


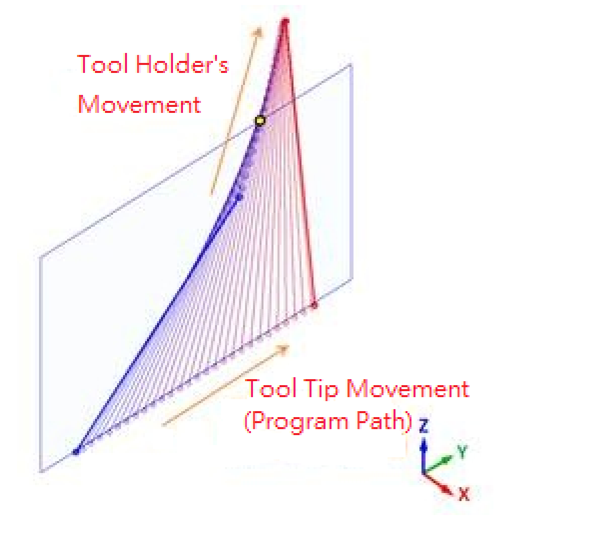
To stay on the plane composed of V_1 and V_2 during movement, B axis rotates without command and eventually returns to its original angle.

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 Vector**
Tool posture changes during movement	**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.

No	Descriptions	Range	Unit	Default	Effective
3055	Basic position of table coordinate system	[0,1]	-	0	Reset

Value

0: RTCP start position

1: zero position of workpiece coordinate

Description

- Table coordinate system：
  Table coordinate system is a workpiece coordinate system exclusive to RTCP and only exist during RTCP is enabled.
  When RTCP is enabled, table coordinate will be established on the table.
  During RTCP is enabled, the commands in NC program will be interpreted as the commands of table coordinate.
- Rotation of the table coordinate system：
  When RTCP is enabled, zero position of rotary axis in table coordinate will be fixed according to Pr3055.

Pr3055 = 0 :

Zero position of rotary axis in table coordinate will be fixed on "current machine coordinate of rotary axis".

Pr3055 = 1 :

Zero position of rotary axis in table coordinate will be fixed on "zero position of rotary axis in workpiece coordinate", regardless of current angle of rotary axis.

During RTCP is enabled, the table coordinate will rotate along with the table.

- Difference in application:

If Pr3055 is 0, user should locate rotary axis to the specific angle before enabling RTCP.

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

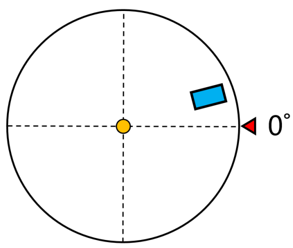
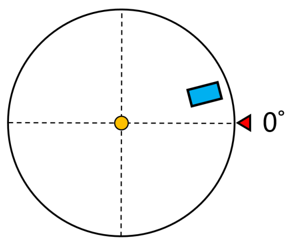
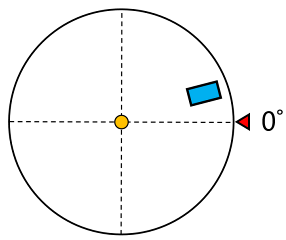
- If the parameter is modified when RTCP is enabled, it will take effect when RTCP being enabled next time.
- This parameter is only valid for the rotary axes on the table side.

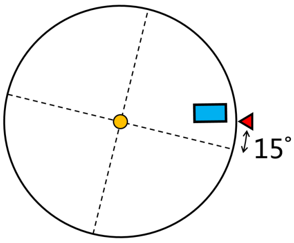
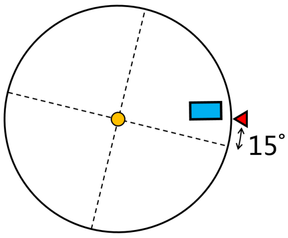
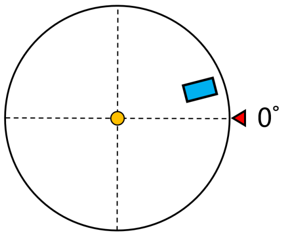
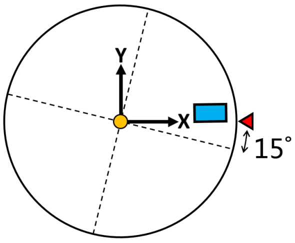
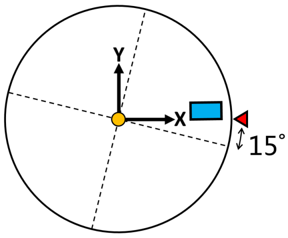
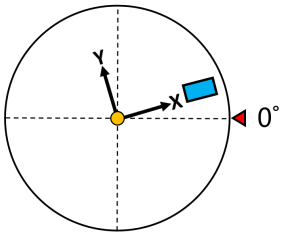
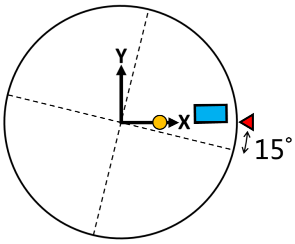
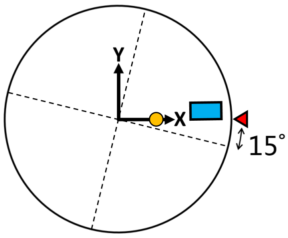
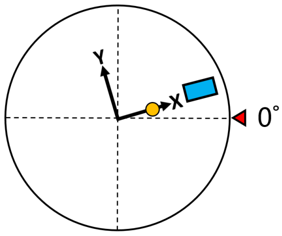
Example

- The purpose of following examples is making the table coordinate to align to the angle of the workpiece (that is C15. in machine coordinate).
- If Pr3055 is 0, like example A, user should locate C axis to C15. in advance before enabling RTCP.
  If Pr3055 is 1, like example B and C, user only need to fill C15. in G54 offset.

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	无	C+15.	C+15.
NC Command	(locate C axis, then enable RTCP) G90 G49 G54 N1 C15. // C positioning N2 G43.4 H1 Z0. // RTCP ON N3 X10. Y0. Z0. // tool tip moves	(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	C15. // C positioning 	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 
Note	The machine coordinate of C axis is 15. when RTCP is enabled. Therefore, the zero position of C axis in table coordinate is at C15. in machine coordinate.	Regardless of the angle of C axis when RTCP is enabled, the zero position of C axis in table coordinate is the same as zero position of C axis in workpiece coordinate.	

10 2. Rotation Tool Center Point (RTCP)

10.1 2. Rotation Tool Center Point (RTCP)

RTCP function will be introduced in this chapter.

10.1.1 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.

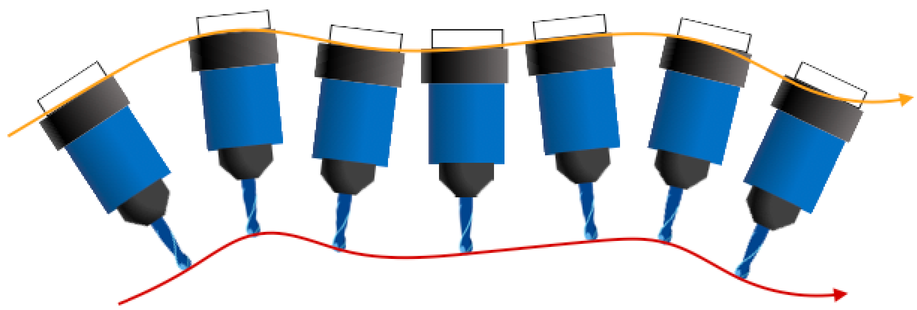


Fig.25

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.

10.1.2 2.2 RTCP Type1

Command Format

```
G43.4 H_；
G49；

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

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】

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.

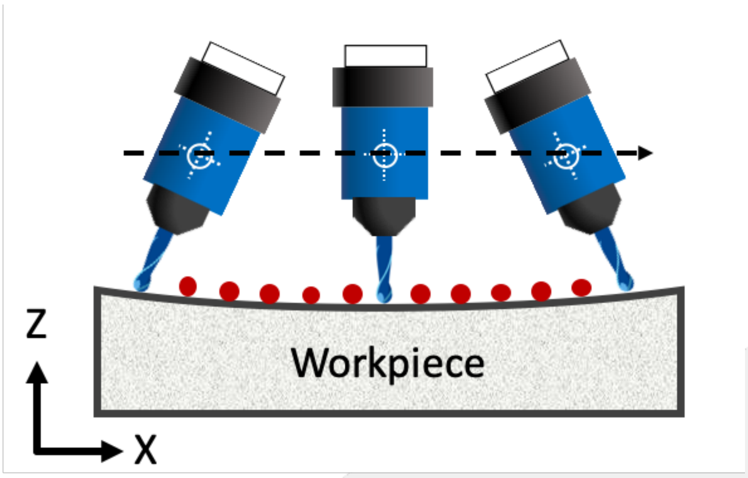


Fig.26

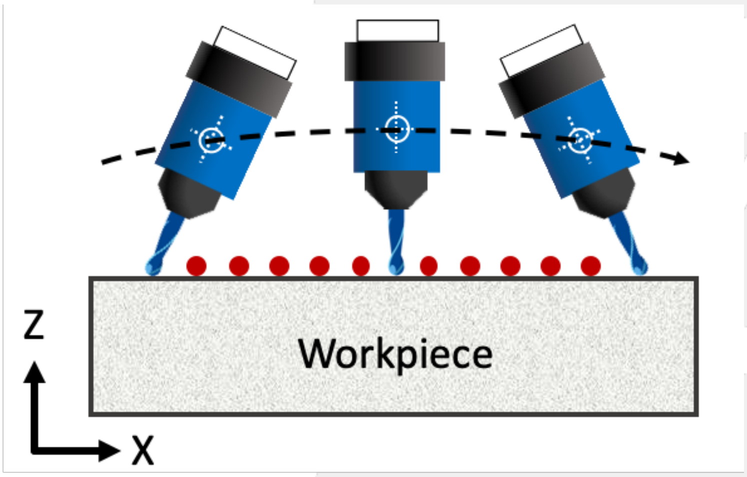


Fig.27

10.1.3 2.3 RTCP Type2

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)
```

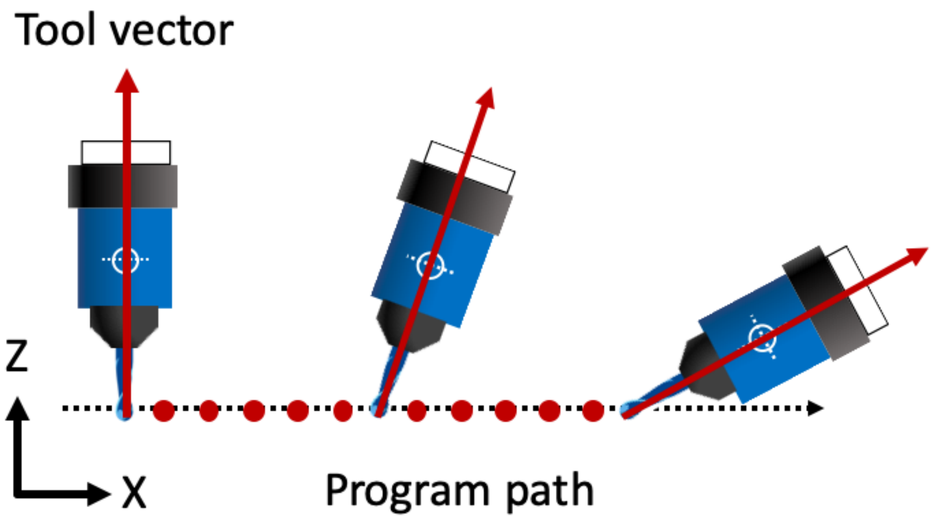


Fig.28

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

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.

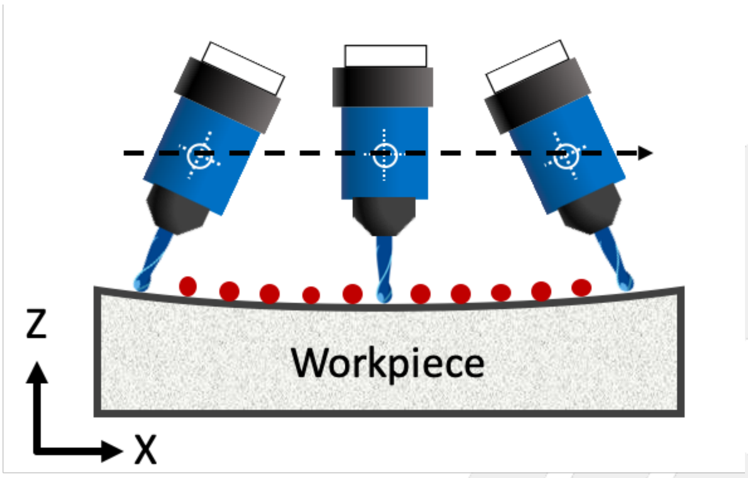


Fig.29

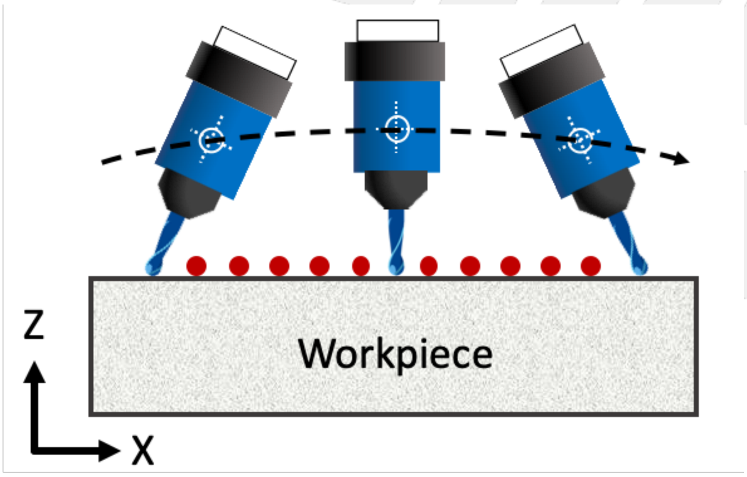


Fig.30

10.2 2.1 Introduction of RTCP Function.

10.2.1 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.

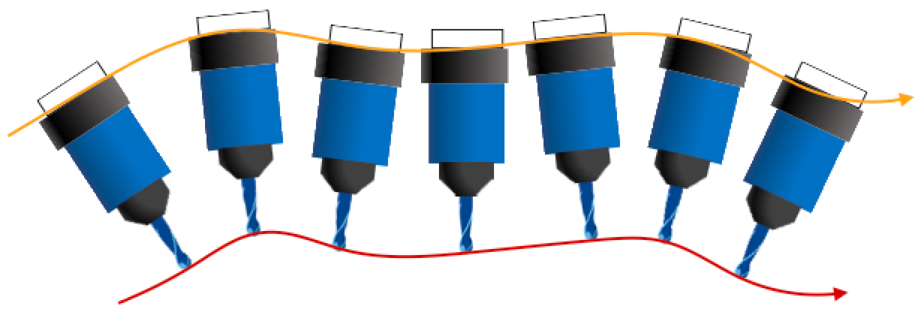


Fig.25

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.

10.3 2.2 G43.4 RTCP Type1 (ENG)

10.3.1 2.2 RTCP Type1

Command Format

```
G43.4 H_；
G49；

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

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】

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.

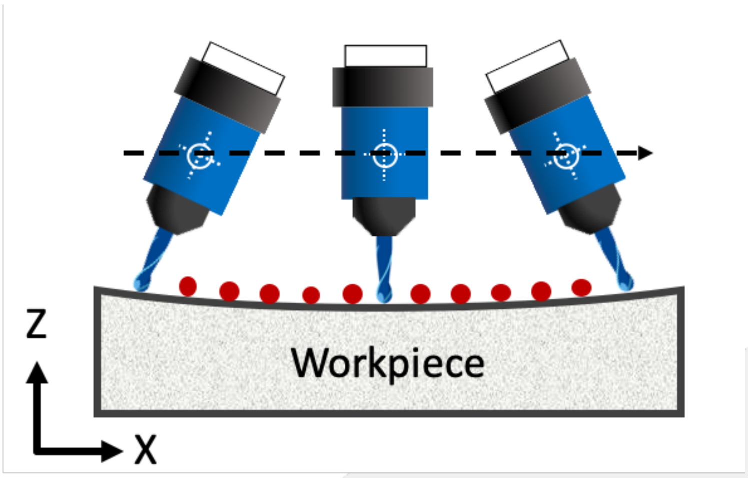


Fig.26

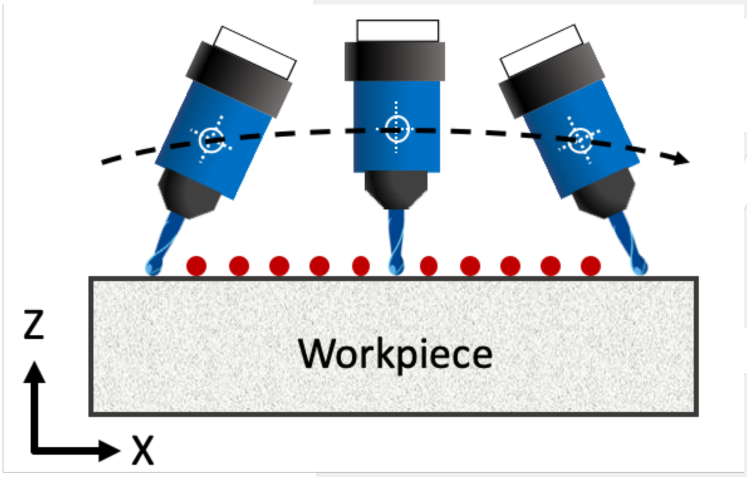


Fig.27

10.4 2.3 G43.5 RTCP Type2 (ENG)

10.4.1 2.3 RTCP Type2

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)
```

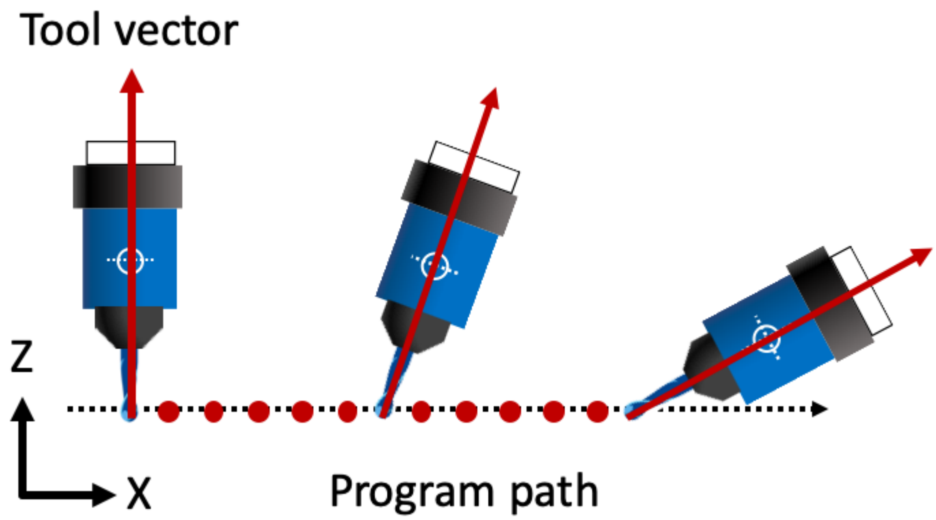


Fig.28

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

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.

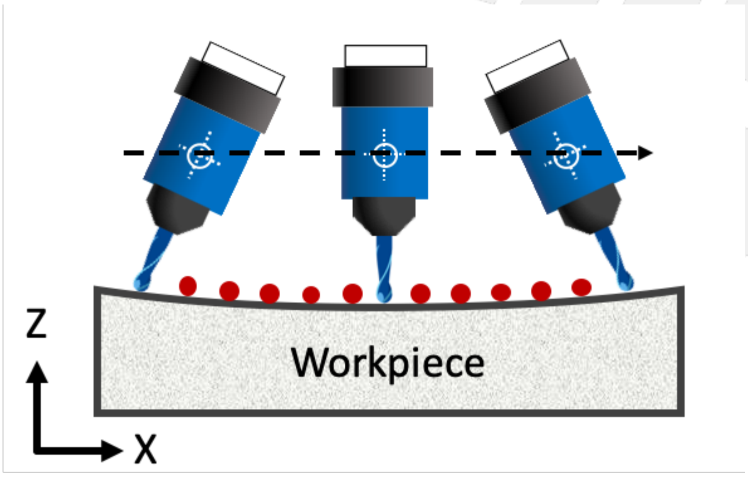


Fig.29

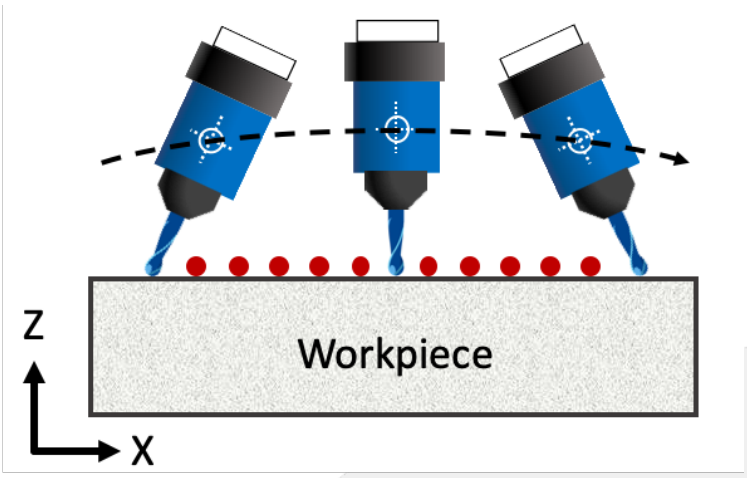


Fig.30

11 3. Extended Functions of RTCP.

11.1 3. Extended Functions of RTCP

Extended Functions of RTCP will be introduced in this chapter.

11.1.1 3.1 Smooth Tool Center Point (STCP)

STCP (Smooth Tool Center Point) is the smoothing function of tool center point control.

When STCP is enabled, in addition to RTCP function, the tool orientation and the path of tool center point will also be smoothed to make the cutting process and workpiece surface smoother.

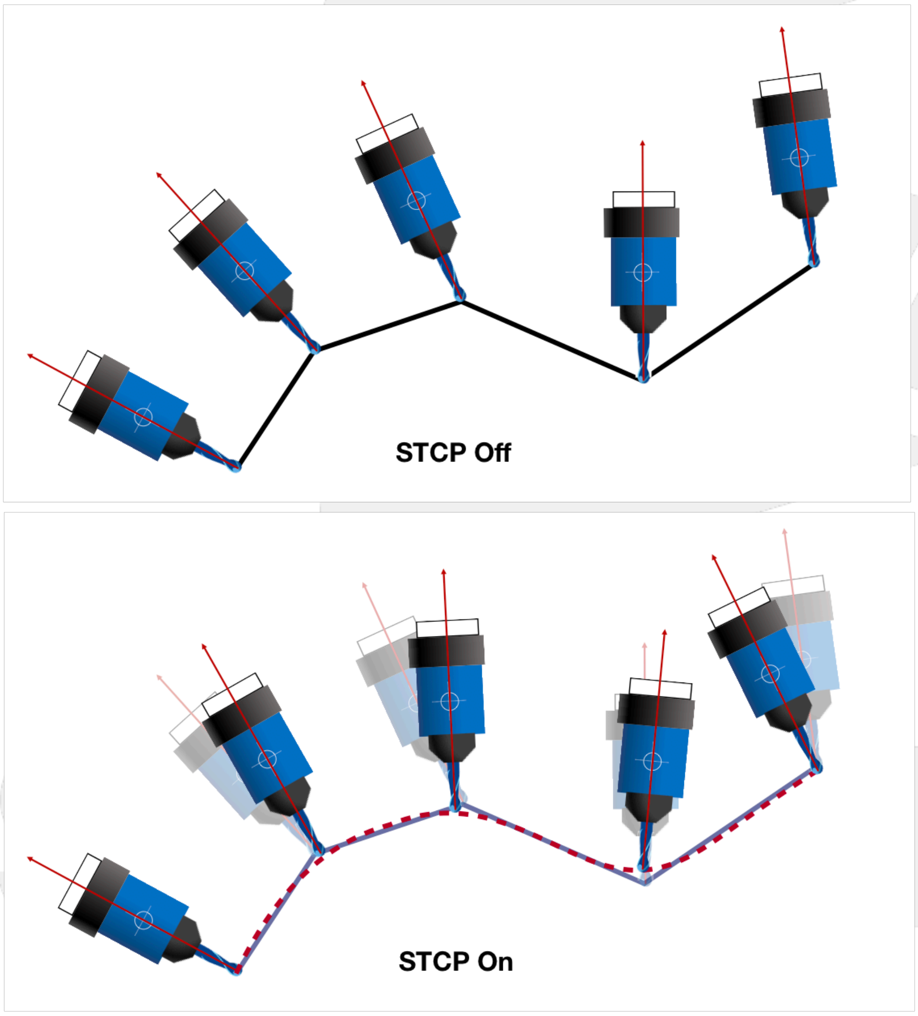


Fig.31

Command Format

```
G43.4 H_ [L_]；
G49；
```

G43.4：enable RTCP;
H ：tool compensation number;
L1：enable STCP;
L0：disable STCP;
G49：disable RTCP, which disable STCP at the same time;

In short, the L argument added behind G43.4 decides whether to enable STCP function, L1: enable; L0: disable. When Pr3051 is set as 1, the L argument can be omitted and STCP will still be enabled.

Application Limitations

1. For STCP, RTCP and HPCC options are all required; after version 10.116.38D, 10.116.54B, 10.118.0A (included), RTCP and STCP options are all required.
2. Only the commands for 5 related axis are allowed under STCP mode, or alarm【COR-107 The format of G5.1/G05 is incorrect】will be triggered.
3. Enable HPCC function with G05 P10000 under STCP mode will trigger alarm【COR-140 G05 HPCC cannot apply under RTCP mode】.

Notifications

1. The word HPCC will be shown on monitor page when STCP function is enabled.
2. Pr3051~Pr3053 are only visible when STCP and RTCP options are both enabled.

Related Parameters

No.	Description	Range	Unit	Initial Value	Take Effect
407	HPCC smoothing tolerance	[2~20]	um	3	Reset
3051	enable smooth RTCP function	[0,1]	-	0	Reset
3052	first rotation axis smooth tolerance	[1,360000]	0.001 deg	500	Reset
3053	second rotation axis smooth tolerance	[1,360000]	0.001 deg	500	Reset

11.1.2 3.2 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).

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.

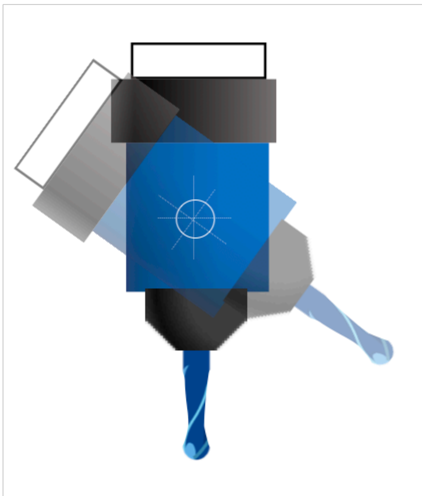
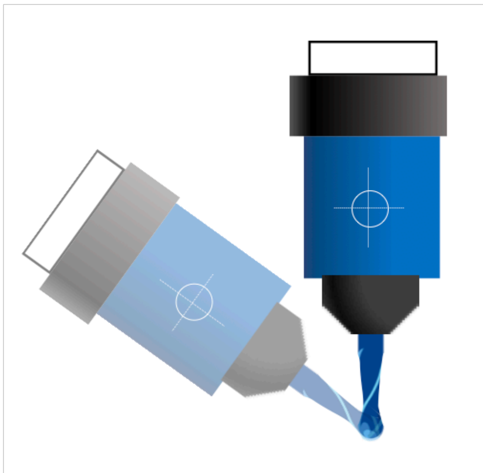
	
R519 = 0	R519 = 1

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

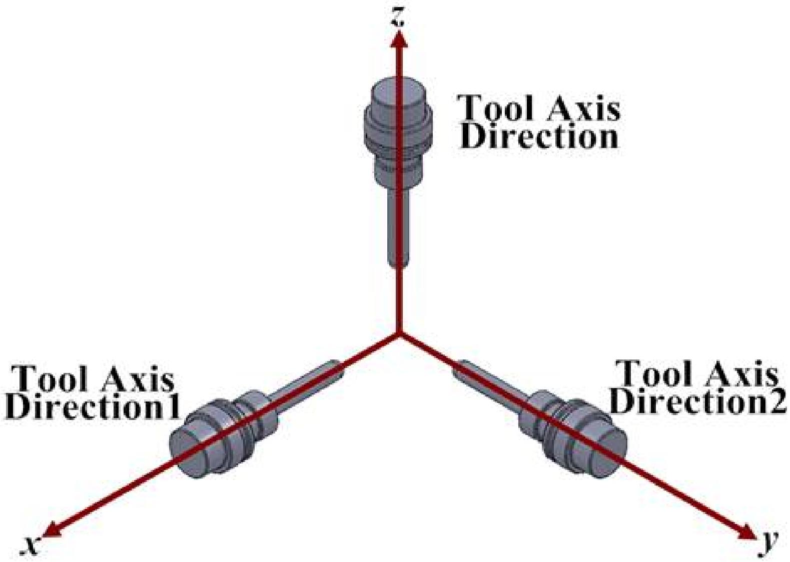


Fig.33

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.

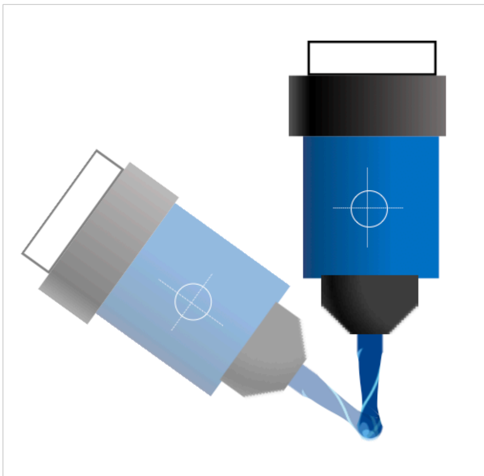
Manual function with tool coordinate is only applicable when there's a rotary axis on spindle side, such as spindle type or mix type 5-axis machines.

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.



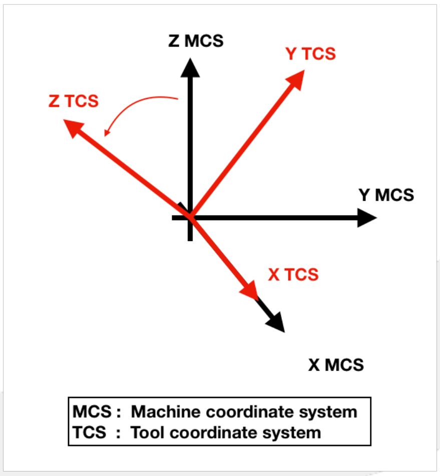


Fig.34

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.

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.

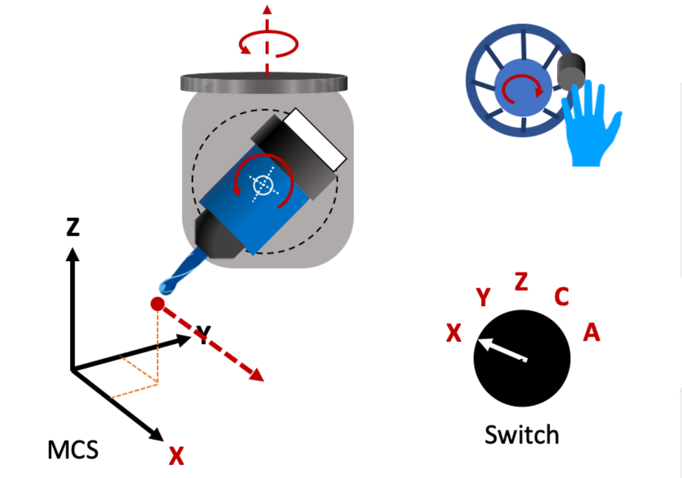


Fig.35

R518 = 1

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

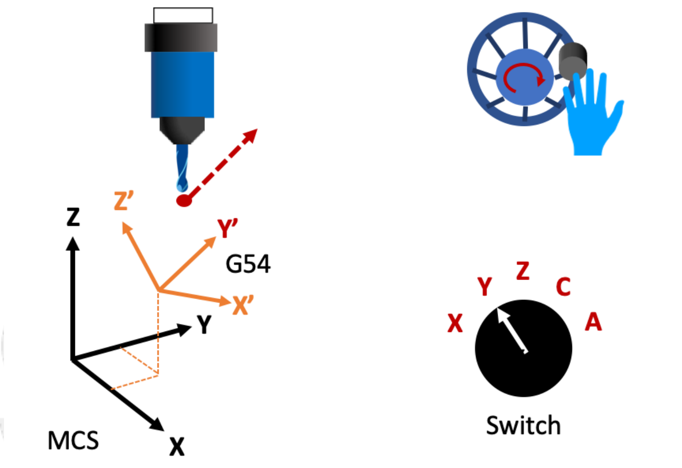


Fig.36

R518 = 2

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

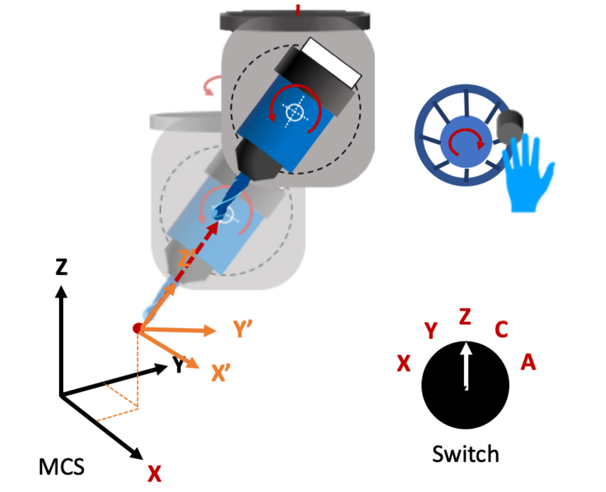


Fig.37

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.

11.1.3 3.3 G68.2 Tilted Working Plane Teach Function

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

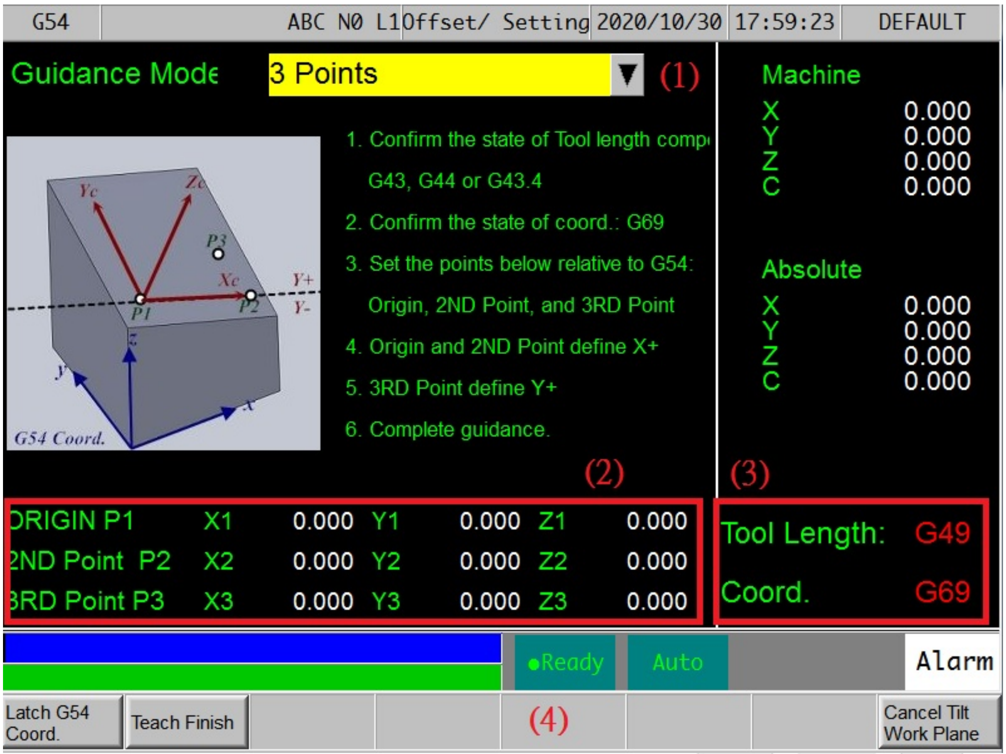


Fig.38

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.

11.1.4 3.4 Teach Modes Description

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.

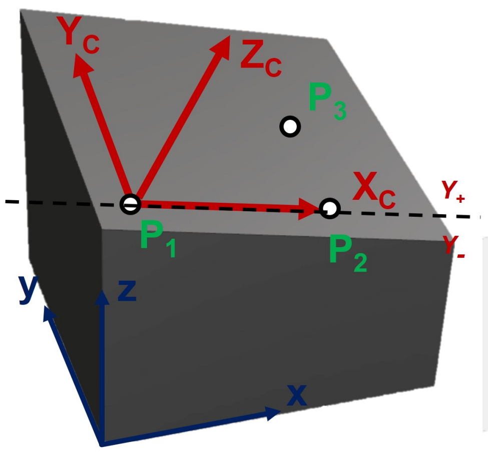


Fig.39

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.

Tool Direction

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

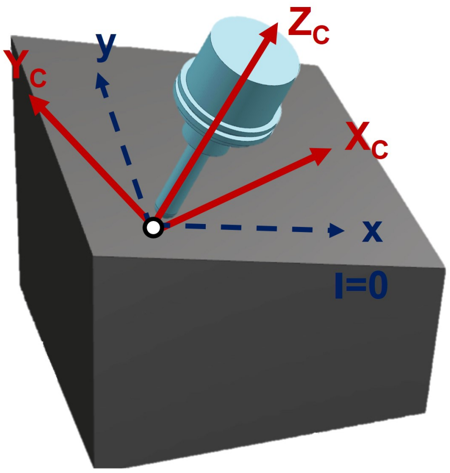


Fig.40

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.

Euler Angle

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

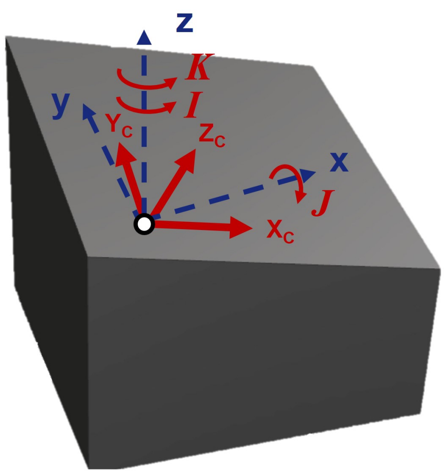


Fig.41

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 4.1 G68.2 Tilted Working Plane Machining (Euler Angle) for the definition of Euler angle.

2 Vectors

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

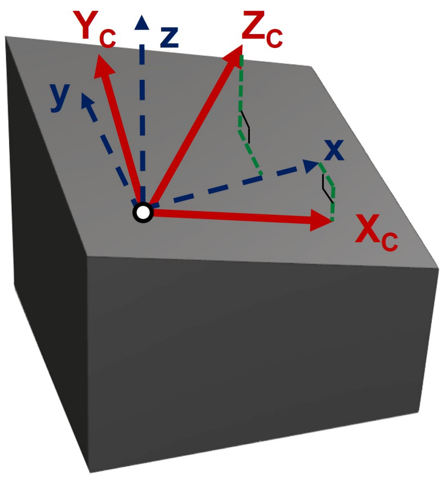


Fig.42

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:

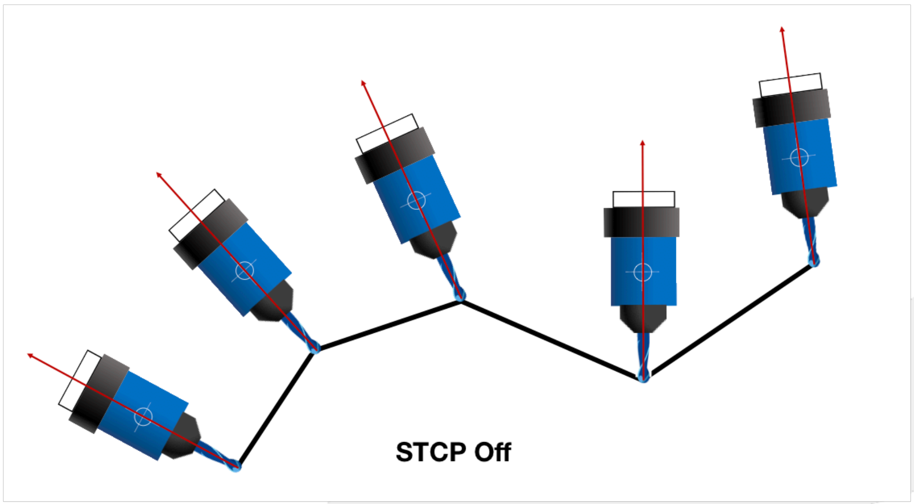
1. The setting X axis and Z axis are not orthogonal.
2. The setting X axis or Z axis is a zero-vector.

11.2 3.1 Smooth Tool Center Point (STCP)

11.2.1 3.1 Smooth Tool Center Point (STCP)

STCP (Smooth Tool Center Point) is the smoothing function of tool center point control.

When STCP is enabled, in addition to RTCP function, the tool orientation and the path of tool center point will also be smoothed to make the cutting process and workpiece surface smoother.



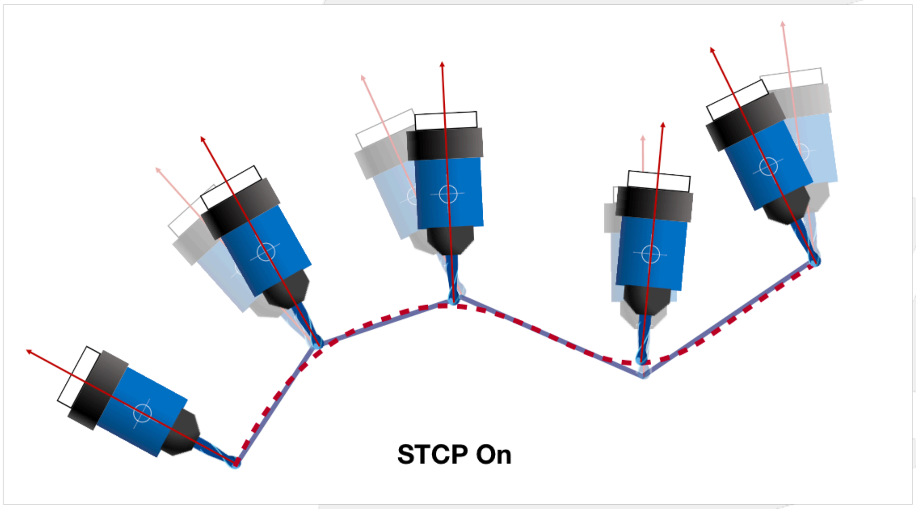


Fig.31

Command Format

```
G43.4 H_ [L_];
G49;
```

G43.4：enable RTCP;
H ：tool compensation number;
L1：enable STCP;
L0：disable STCP;
G49：disable RTCP, which disable STCP at the same time;

In short, the L argument added behind G43.4 decides whether to enable STCP function, L1: enable; L0: disable. When Pr3051 is set as 1, the L argument can be omitted and STCP will still be enabled.

Application Limitations

1. For STCP, RTCP and HPCC options are all required; after version 10.116.38D, 10.116.54B, 10.118.0A (included), RTCP and STCP options are all required.
2. Only the commands for 5 related axis are allowed under STCP mode, or alarm【COR-107 The format of G5.1/G05 is

incorrect】will be triggered.
3. Enable HPCC function with G05 P10000 under STCP mode will trigger alarm【COR-140 G05 HPCC cannot apply under RTCP mode】.

Notifications

1. The word HPCC will be shown on monitor page when STCP function is enabled.
2. Pr3051~Pr3053 are only visible when STCP and RTCP options are both enabled.

Related Parameters

No.	Description	Range	Unit	Initial Value	Take Effect
407	HPCC smoothing tolerance	[2~20]	um	3	Reset
3051	enable smooth RTCP function	[0,1]	-	0	Reset
3052	first rotation axis smooth tolerance	[1,360000]	0.001 deg	500	Reset
3053	second rotation axis smooth tolerance	[1,360000]	0.001 deg	500	Reset

11.3 3.2 Manual RTCP Functions.

11.3.1 3.2 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).

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.

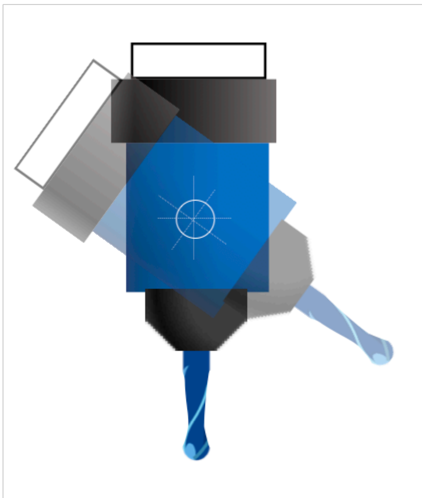
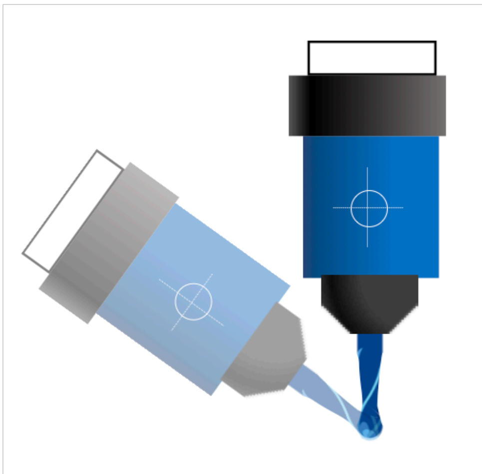
	
R519 = 0	R519 = 1

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

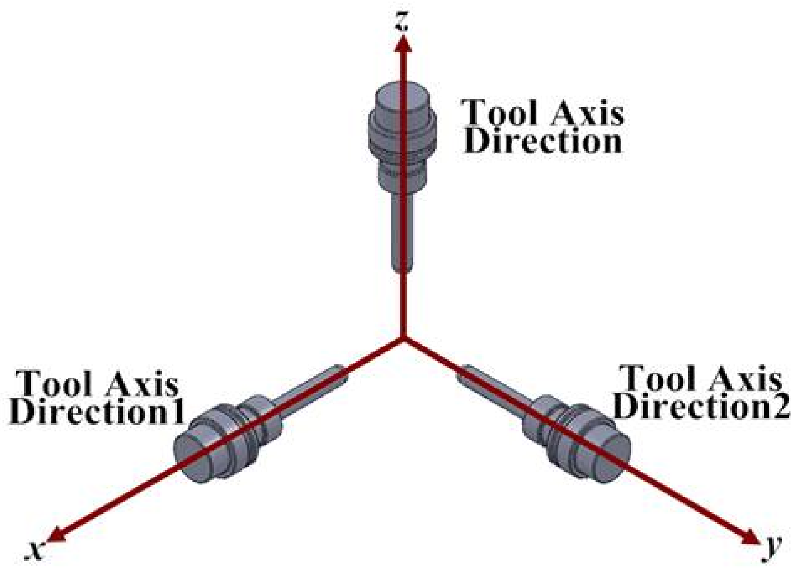


Fig.33

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.

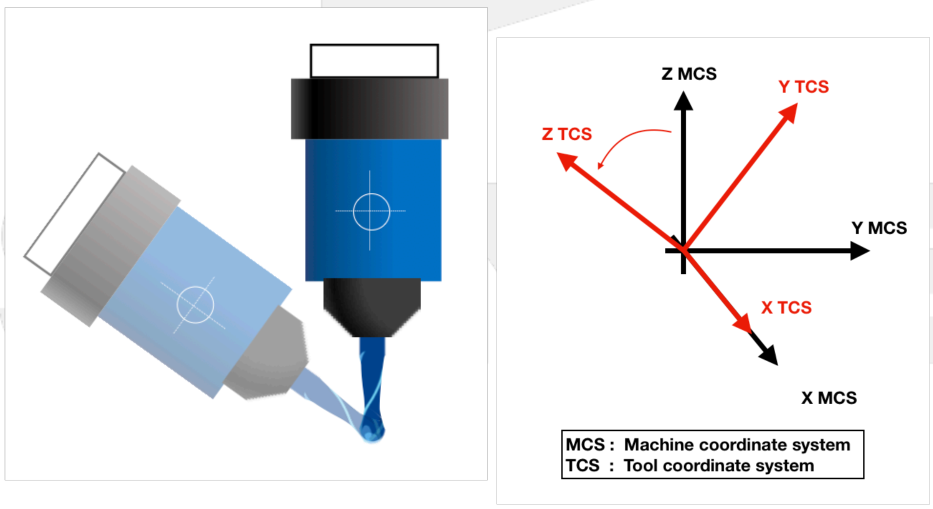


Fig.34

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.

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.

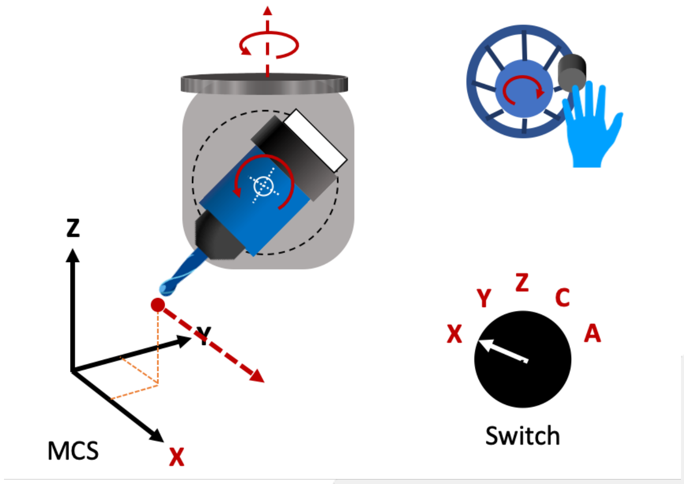


Fig.35

R518 = 1

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

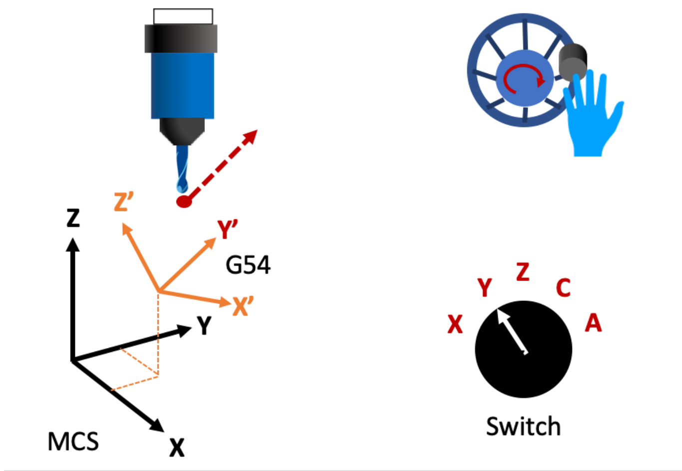


Fig.36

R518 = 2

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

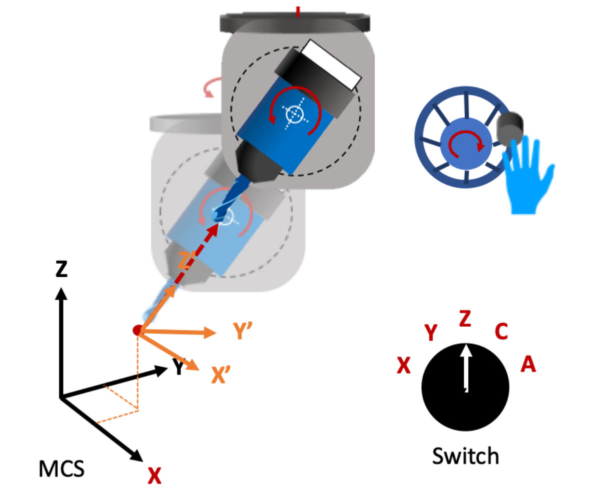


Fig.37

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.

11.4 3.3 G68.2 Tilted Working Plane Teach Function.

11.4.1 3.3 G68.2 Tilted Working Plane Teach Function

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

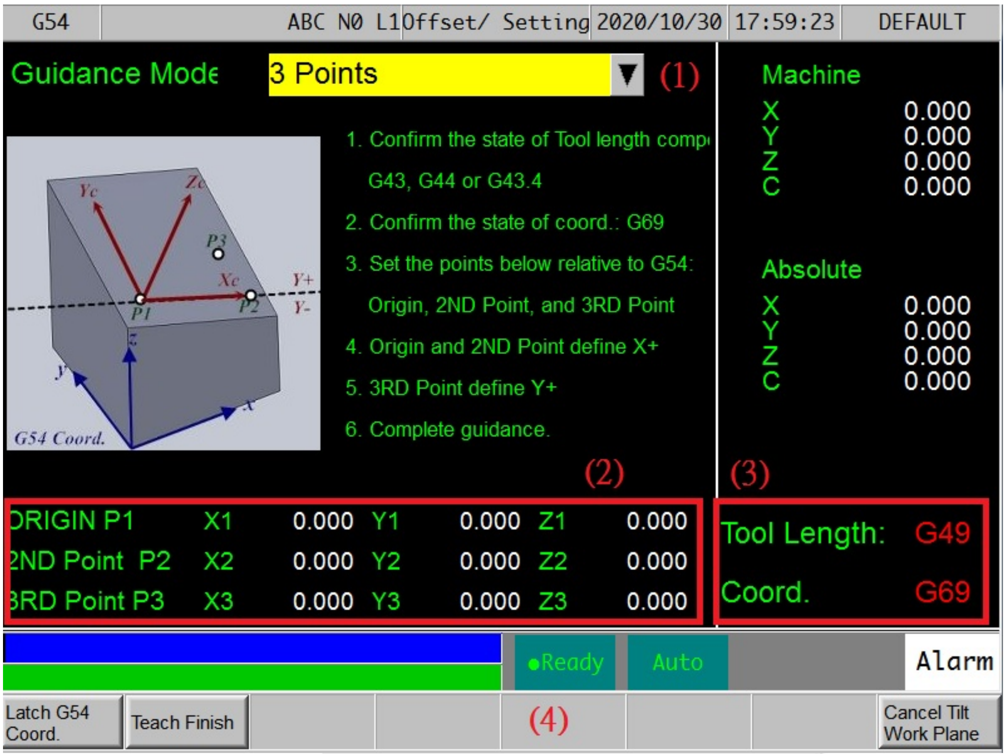


Fig.38

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.

11.5 3.4 Teach Modes Description.

11.5.1 3.4 Teach Modes Description

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.

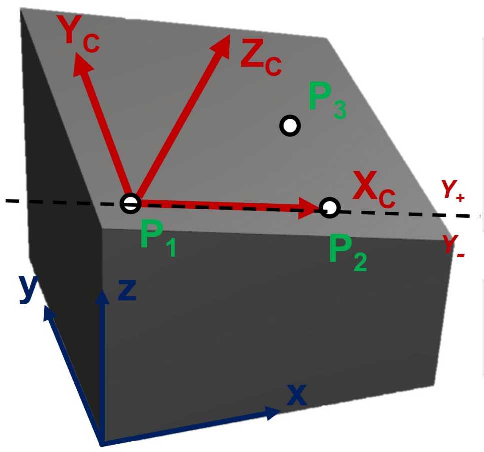


Fig.39

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.

Tool Direction

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

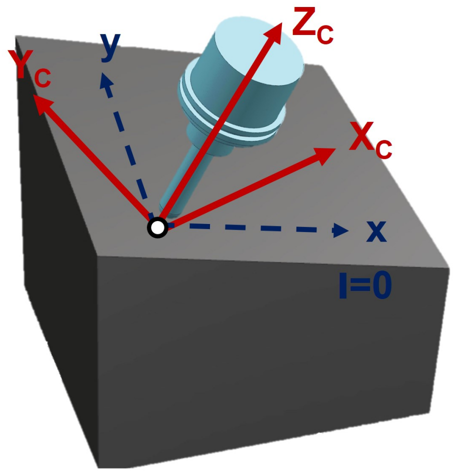


Fig.40

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.

Euler Angle

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

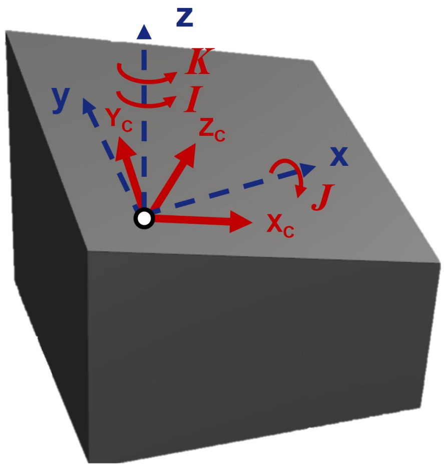


Fig.41

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 4.1 G68.2 Tilted Working Plane Machining (Euler Angle) for the definition of Euler angle.

2 Vectors

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

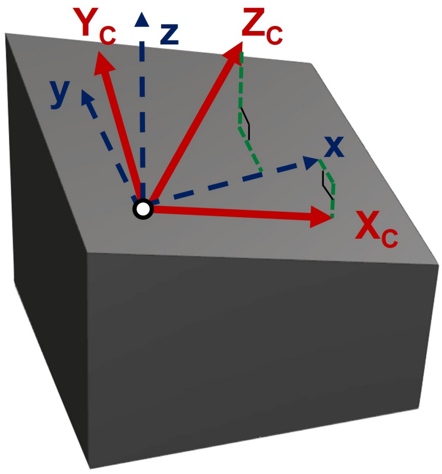


Fig.42

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.

12 4. Tilted Working Plane Machining.

12.1 4. Tilted Working Plane Machining

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

12.1.1 4.1 G68.2 Tilted Working Plane Machining (Euler Angle)

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.

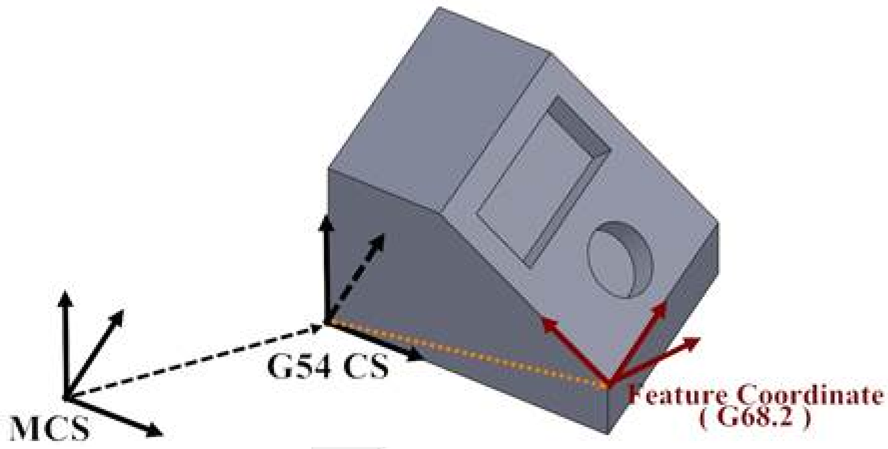


Fig.43

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.

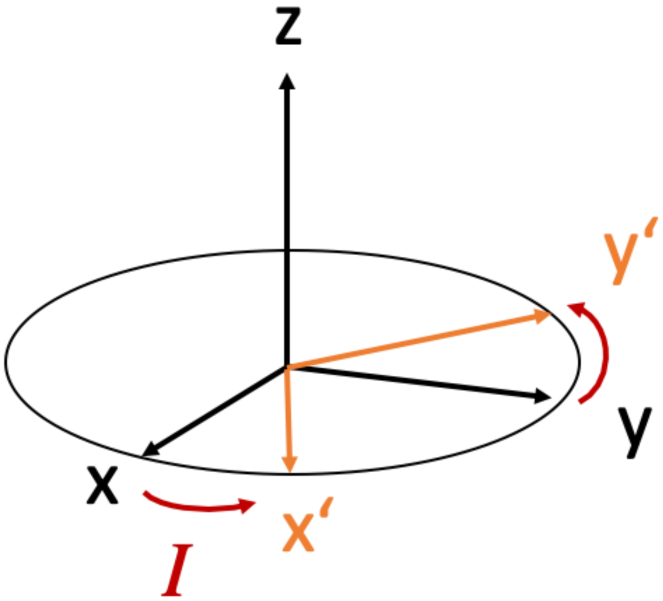


Fig.44

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.

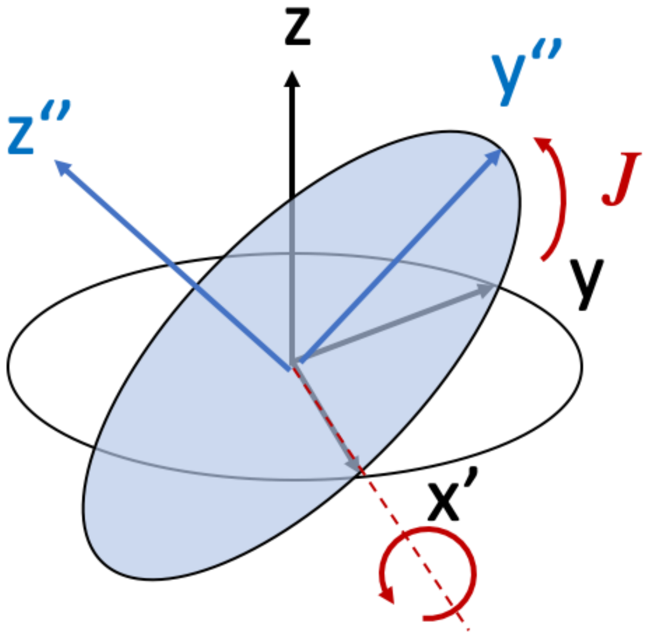


Fig.45

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.

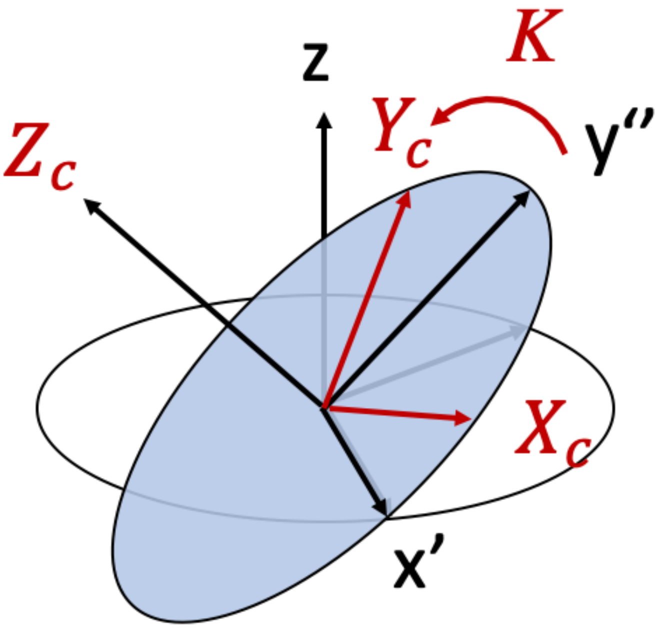


Fig.46

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;

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.

Related Parameters

No	Descriptions	Range	Unit	Details	Effective
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.	Reset

12.1.2 4.2 G53.1 Tool Alignment Function for Tilted Working Plane

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.

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.

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;
```

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.
```

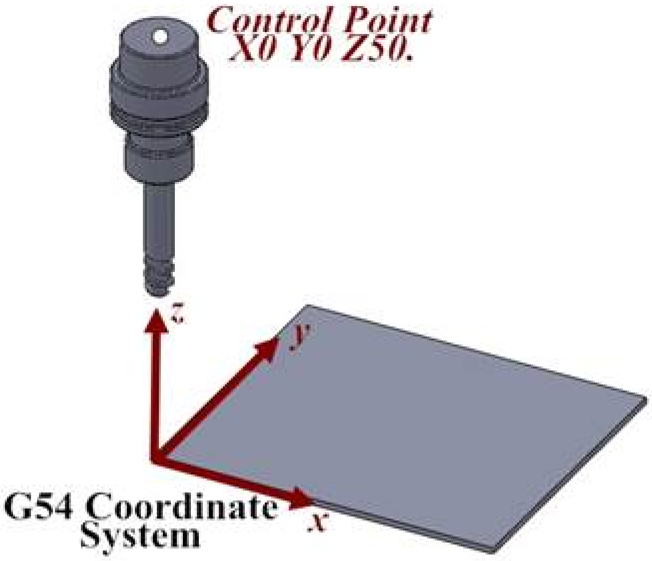


Fig.47

```
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.
```

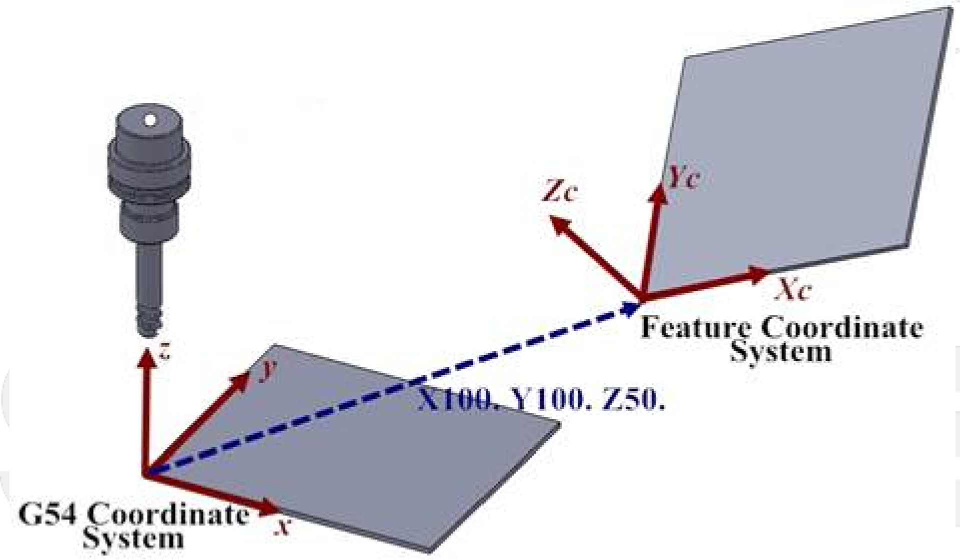


Fig.48

```
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.
```

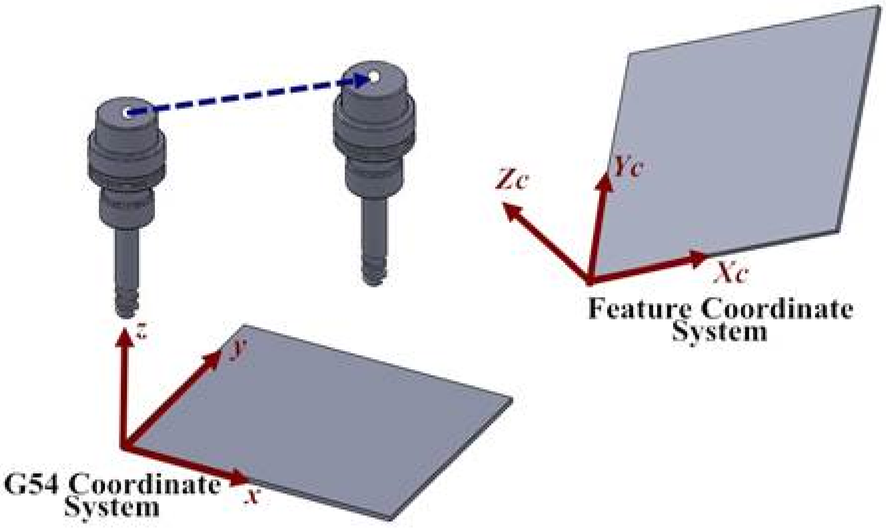


Fig.49

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

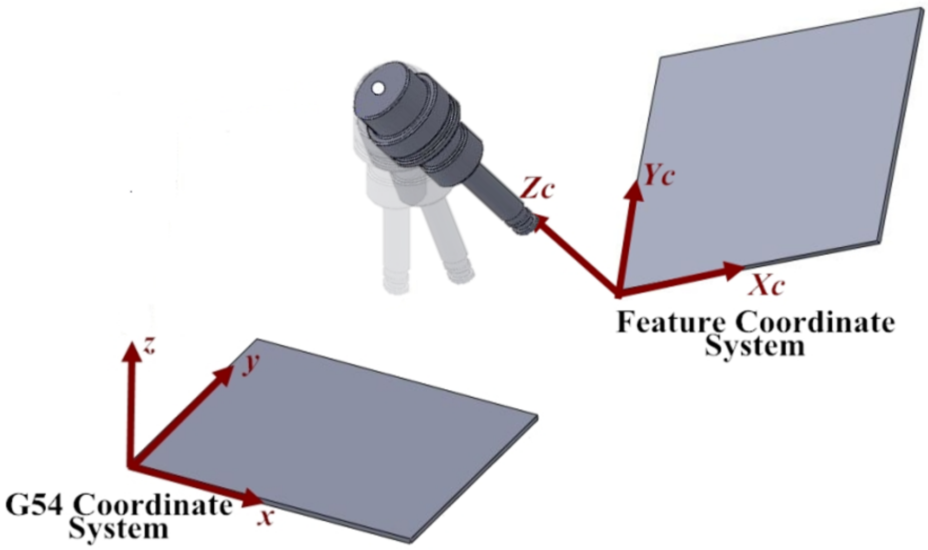


Fig.50

```
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.
```

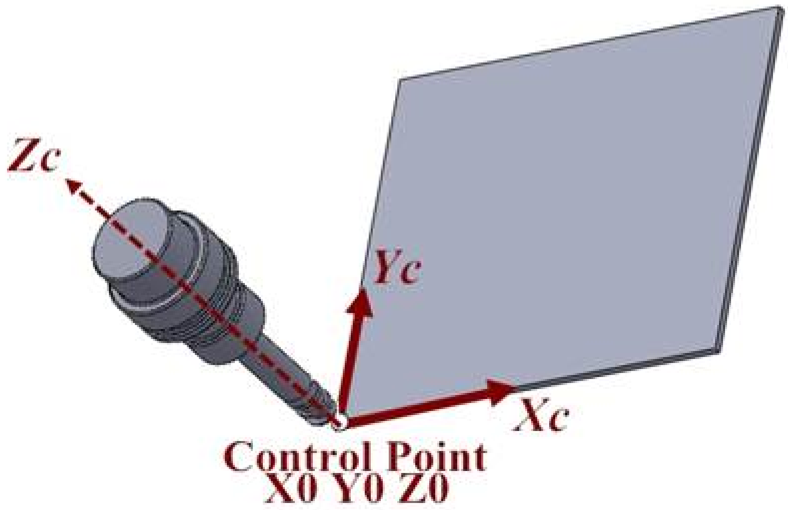


Fig.51

```
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.
```

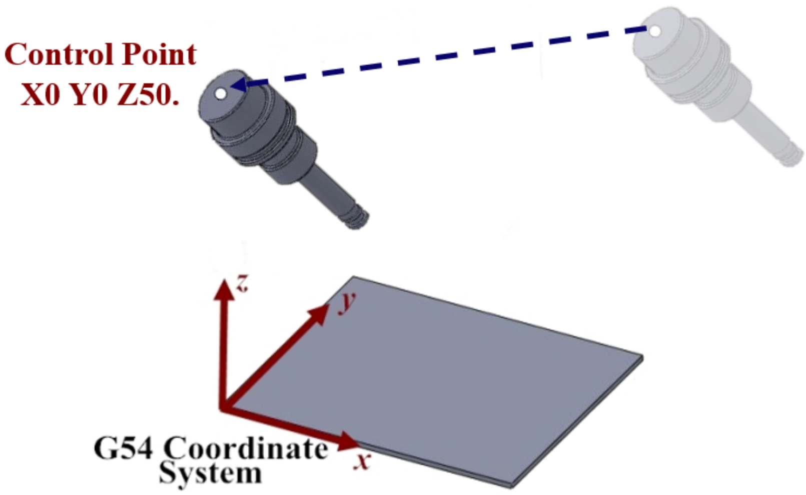


Fig.52

12.1.3 4.3 G53.3 Tool Alignment Function for Tilted Working Plane (5-Axis simultaneous motion)

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.

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.

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

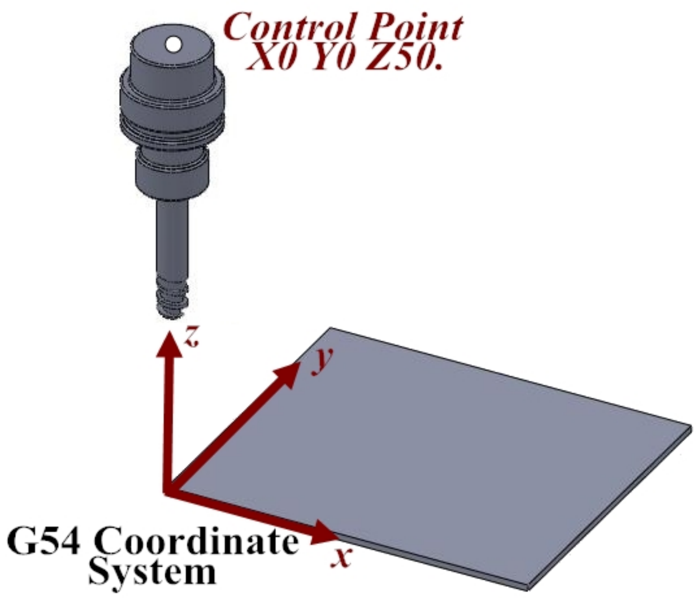
Program Example

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

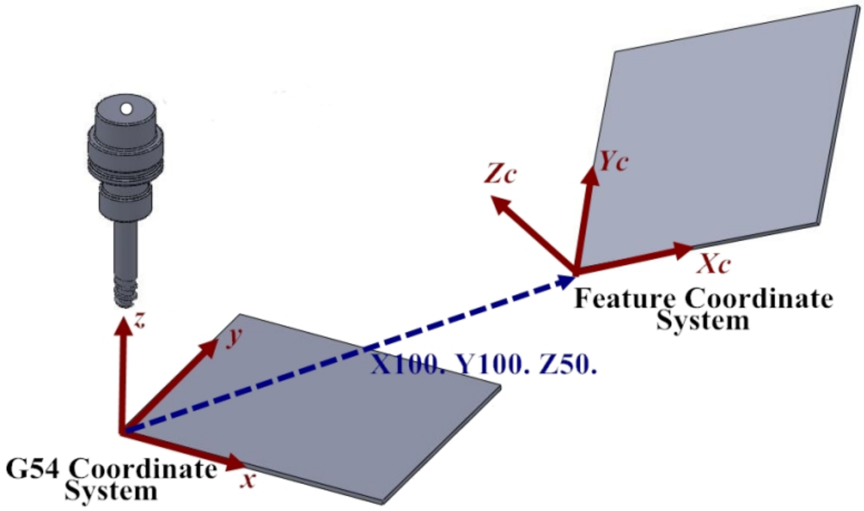
```
N1 G90 G54 G01 X0 Y0 Z50. F1000；
N2 G68.2 X100. Y100. Z50. I30. J15. K20.；
N3 G01 X0 Y0 Z50. F1000；
N4 G53.3 X0 Y0 Z0 H1；

... //  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:

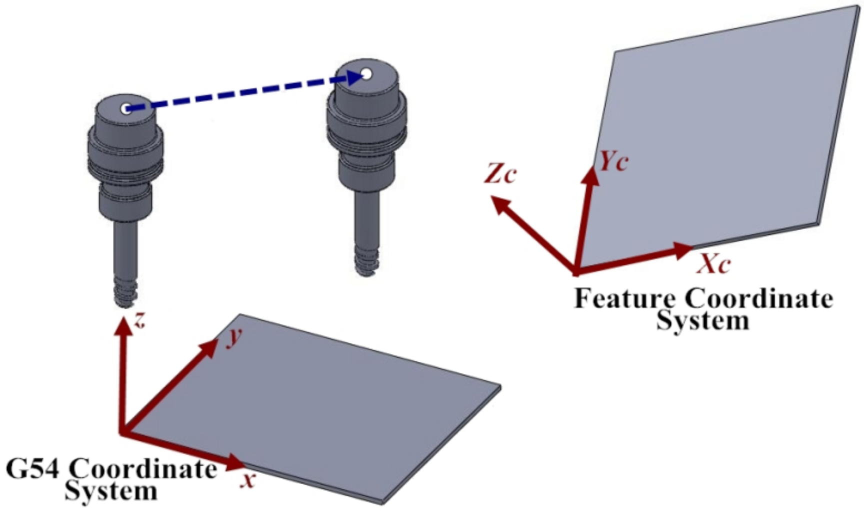
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of G54 coordinate by G01 in speed of 1000 mm/min.
```



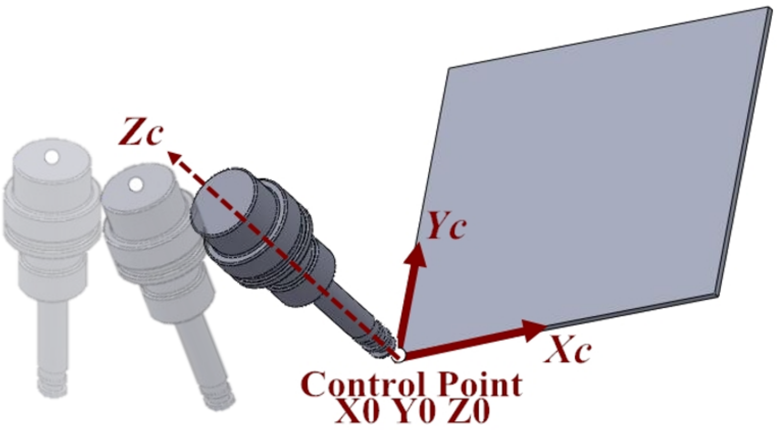
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
// Specify X100. Y100. Z50. relative to the origin of G54 coordinate as the origin of Tilted Working Plane, and the Euler angles are I30. J15. K20.
// The program coordinate will transform to Tilted Working Plane after G68.2 is executed.
```



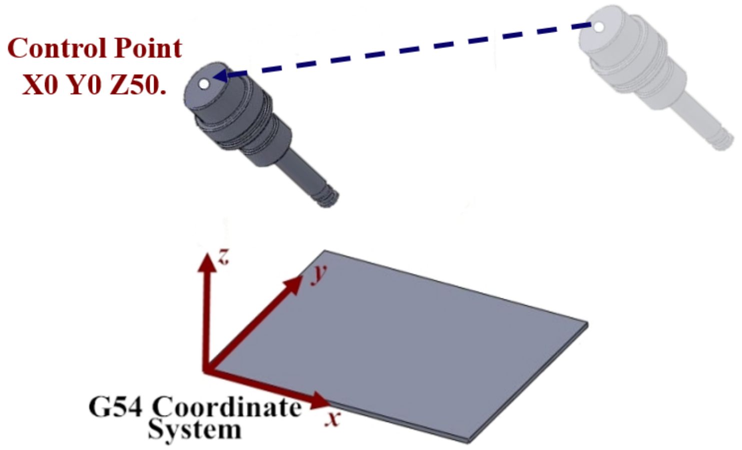

```
N3 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of Tilted Working Plane by G01 in speed of 1000 mm/min, but the tool direction remains the same.
```



```
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.
```



12.1.4 4.4 G53.6 Tool Alignment Function for Tilted Working Plane (TCP/ Rotation Center)

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.

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.

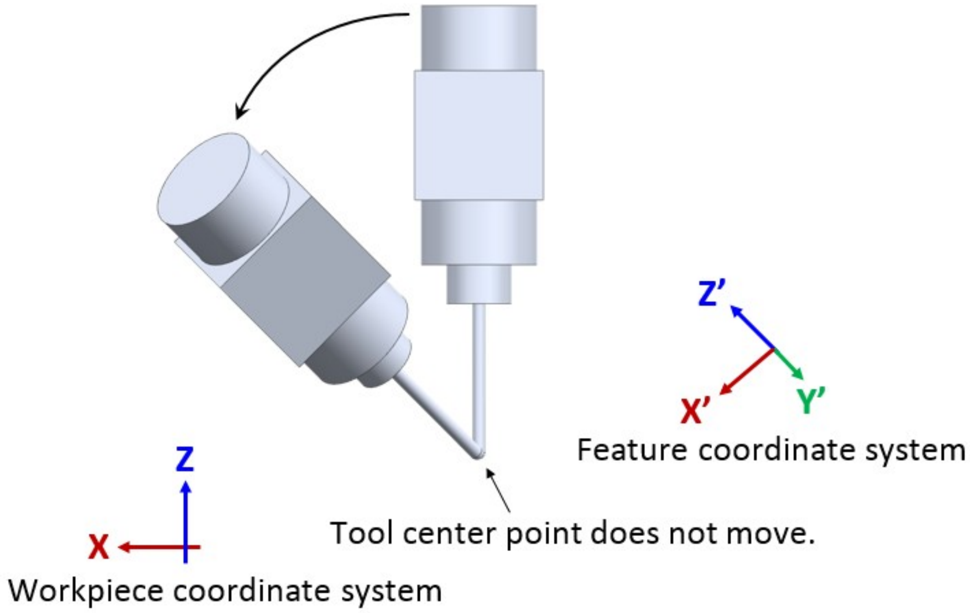


Fig.52

- **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.

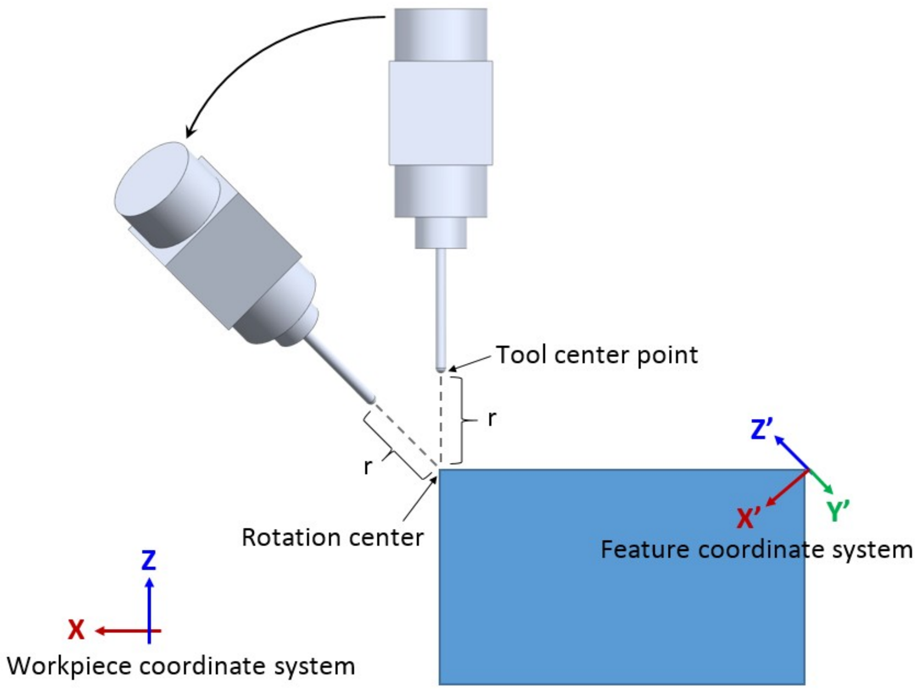


Fig.53

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

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.
```

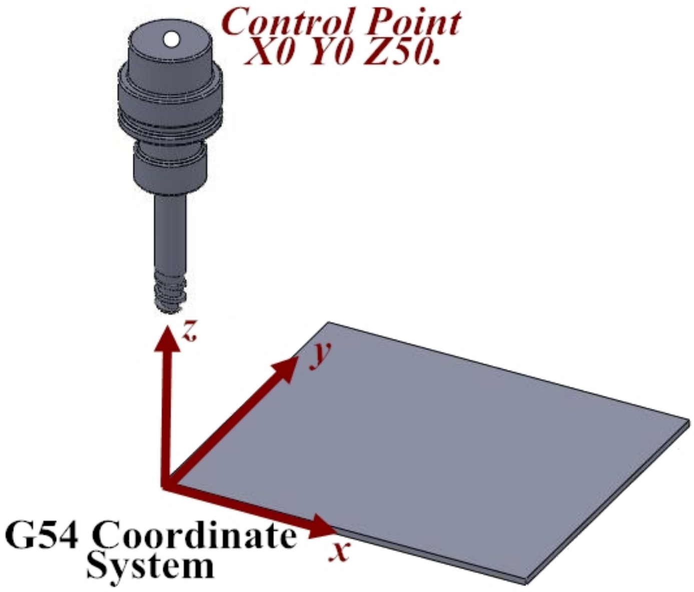


Fig.54

```
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.
```

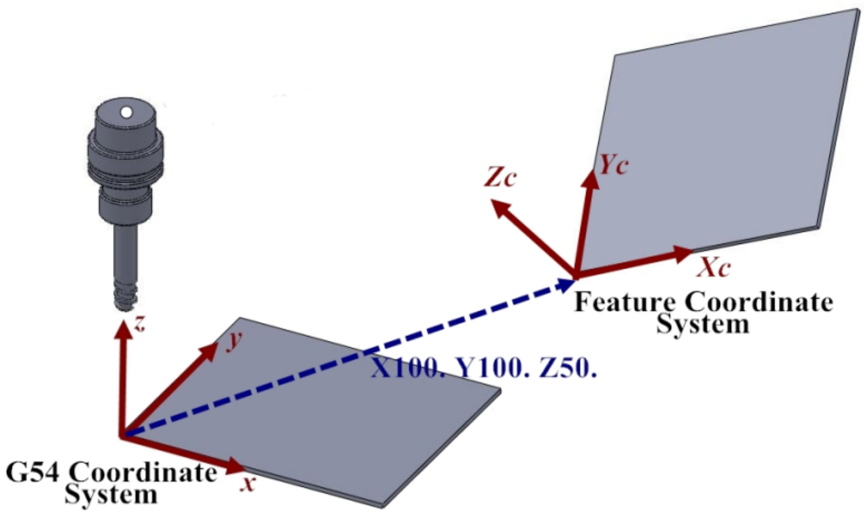


Fig.55

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

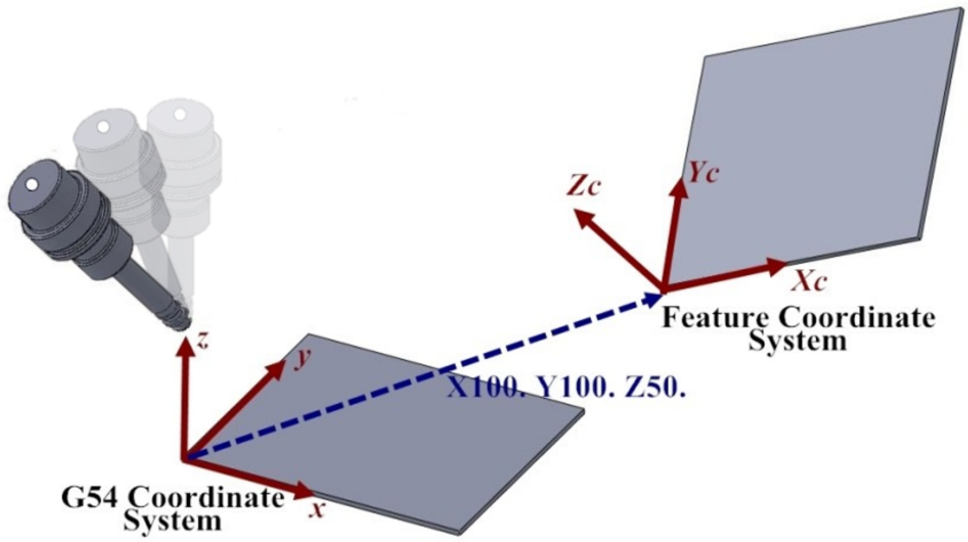


Fig.56

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

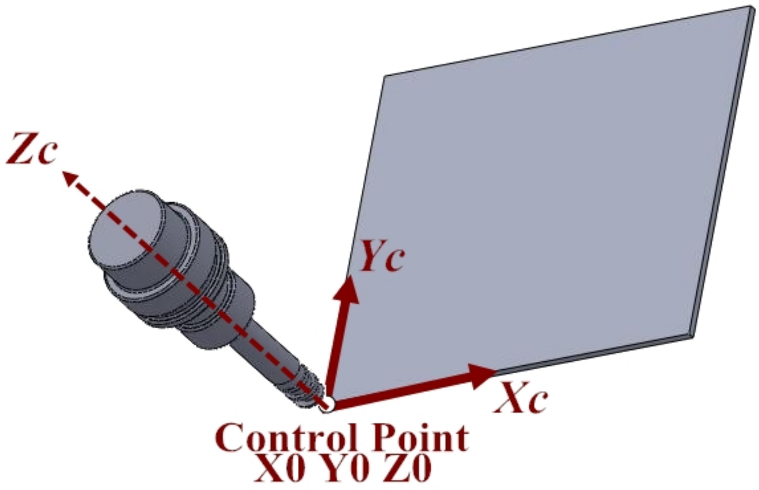


Fig.57

```
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.
```

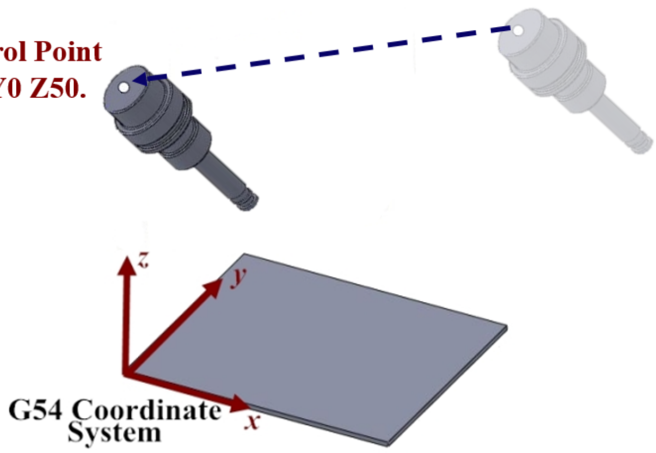
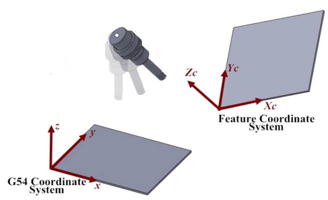
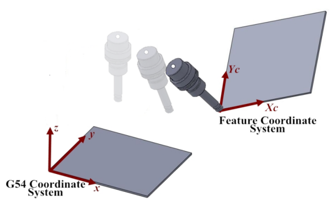
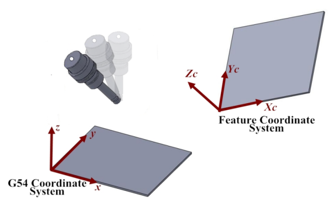


Fig.58

12.1.5 4.5 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
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

12.1.6 **4.6 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.

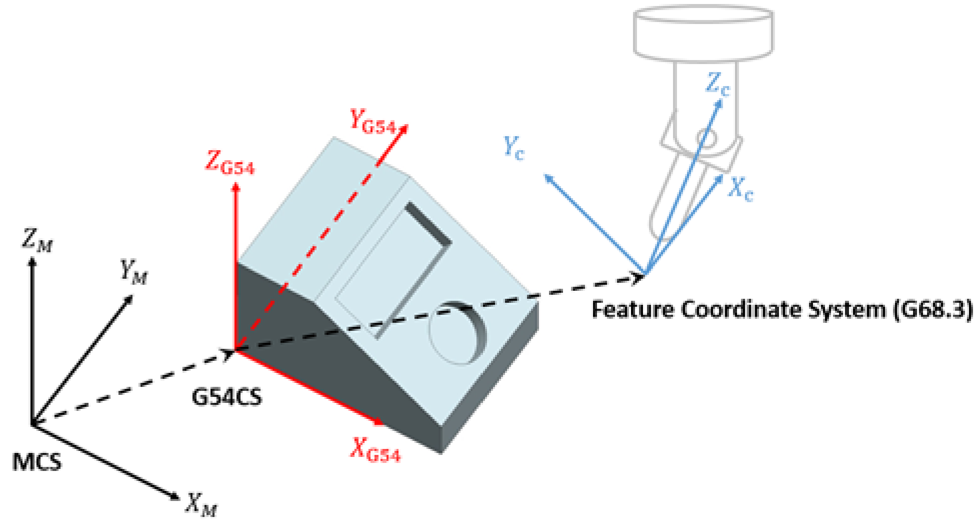


Fig.57

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***

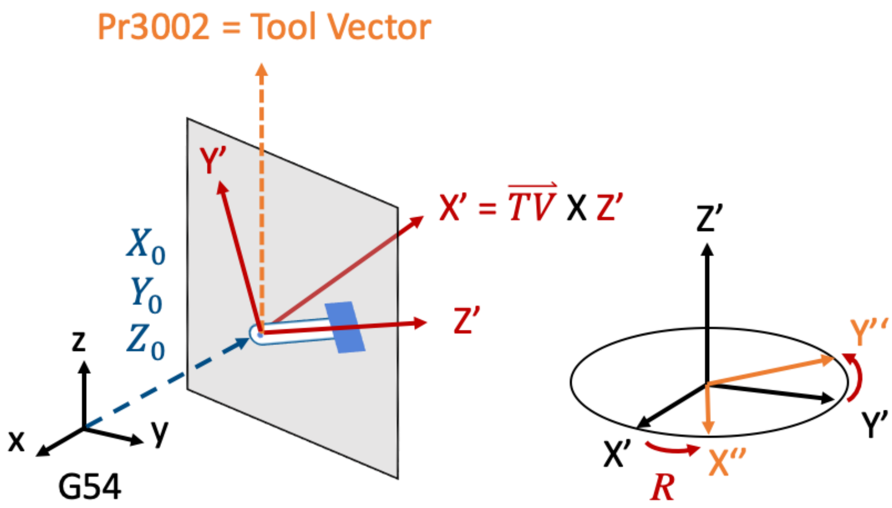


Fig.58

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***

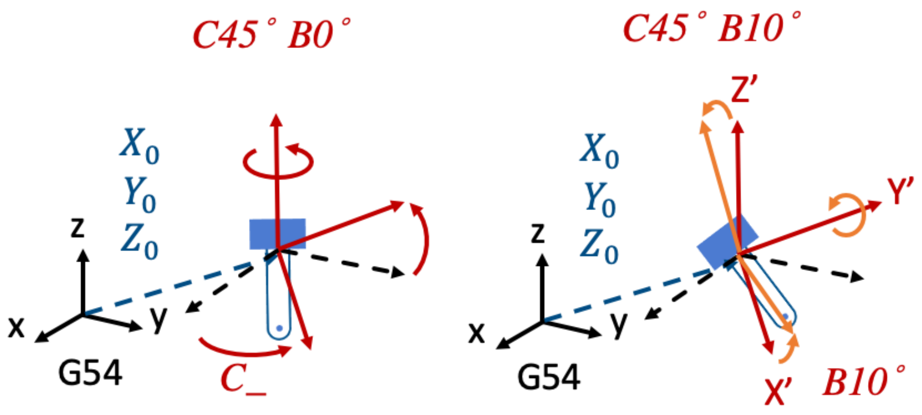


Fig.59

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. G43 should be executed after G68.3.
4. G68.3 can NOT be executed while RTCP (G43.4/G43.5) is enabled.
5. When G68.3 P1 is executed, R argument will be ignored if provided.
6. G68.3 can be executed for multiple times and each setting is relative to G54 coordinate.
7. G68.3 is only for 5-axis machines with option-13 enabled at the same time.

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.

Program Example:

(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;
```

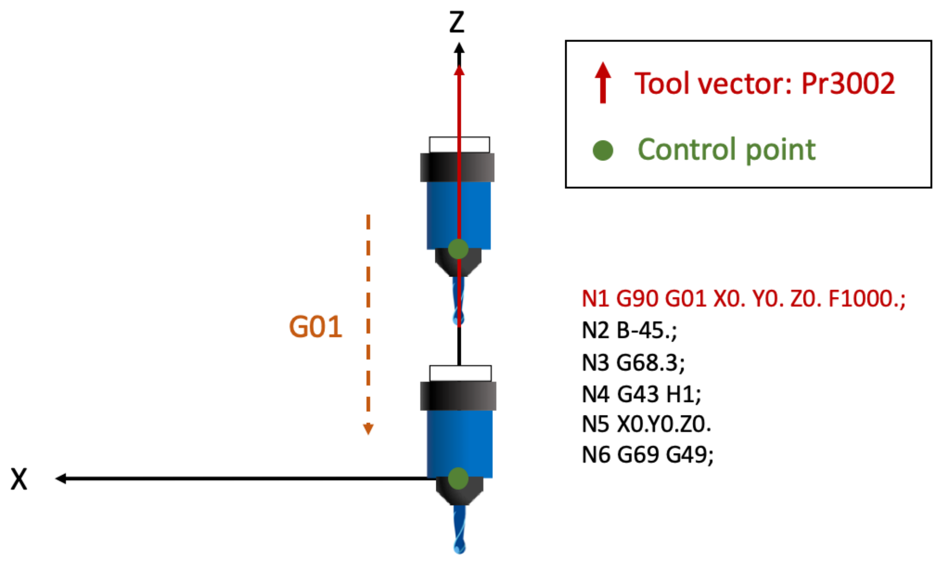


Fig.60

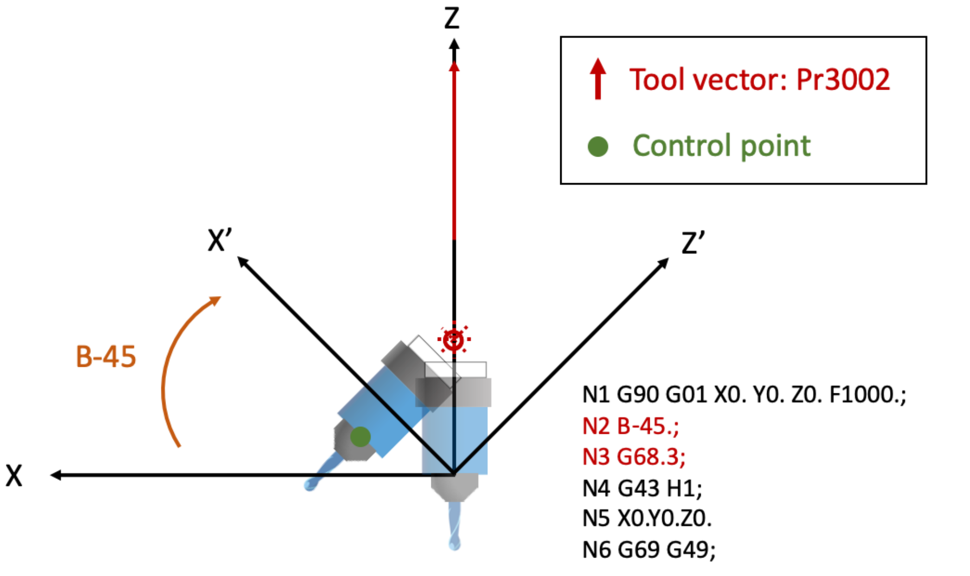

Z
Tool vector: Pr3002
Control point
X'
Z'
B-45
X
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;
N4 G43 H1;
N5 X0.Y0.Z0.
N6 G69 G49;

Fig.61

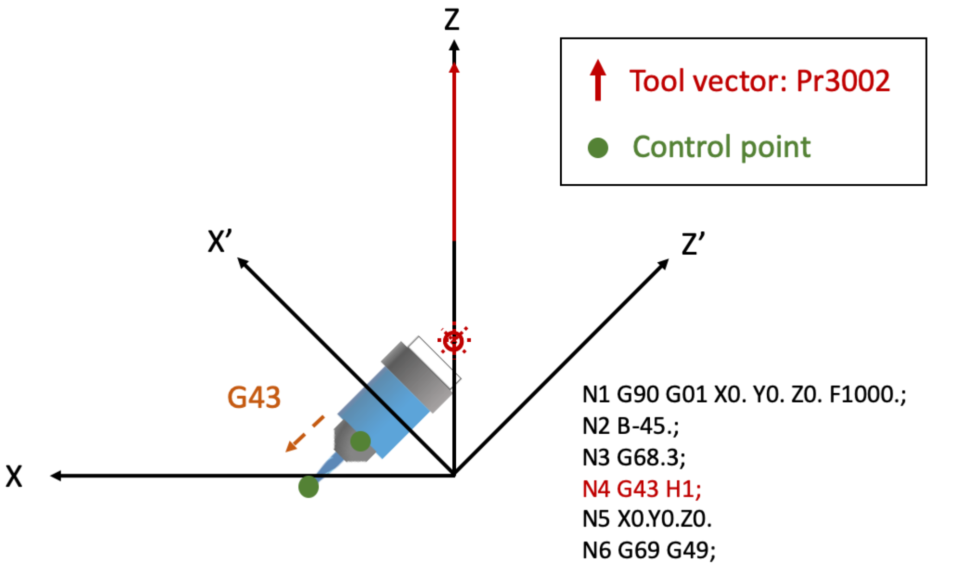

Z
Tool vector: Pr3002
Control point
X'
Z'
G43
X
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;
N4 G43 H1;
N5 X0.Y0.Z0.
N6 G69 G49;

Fig.62

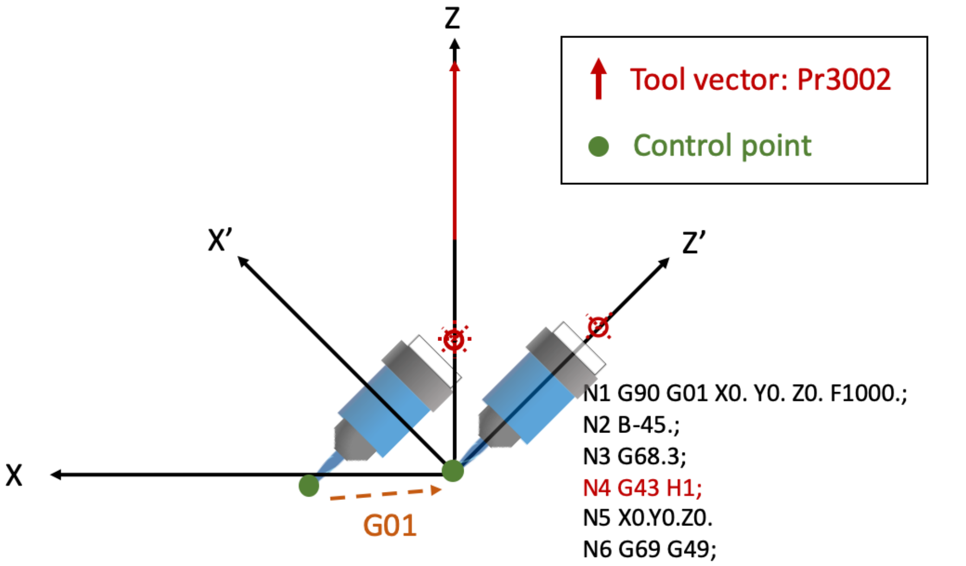
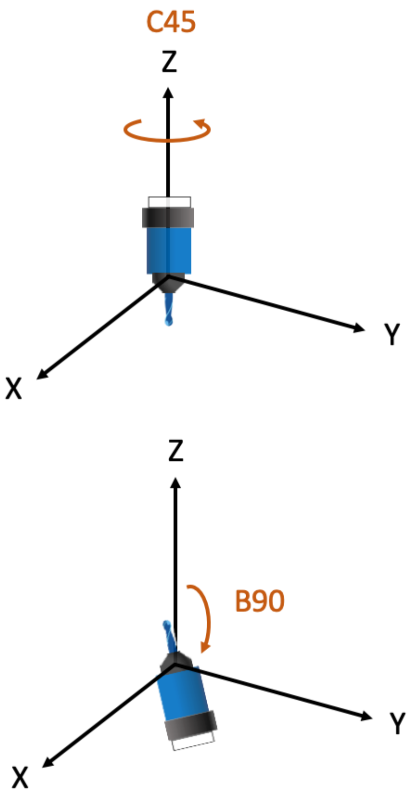


Fig.63

(2)

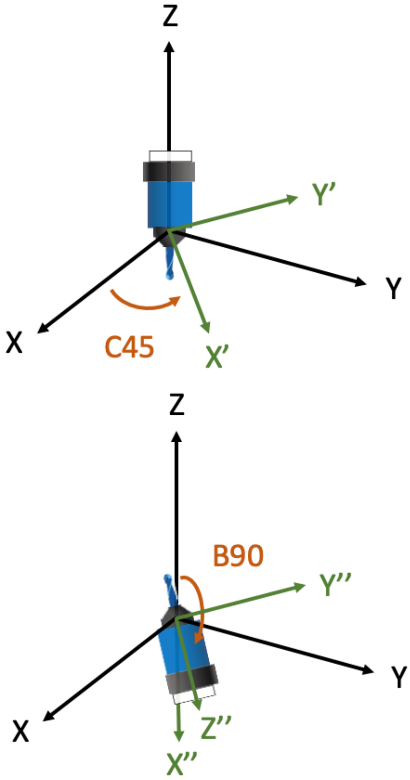
NC program and Fig.64~66 are used to explain the motions within Tilted Working Plane when G68.3 is enabled.

```
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;
```



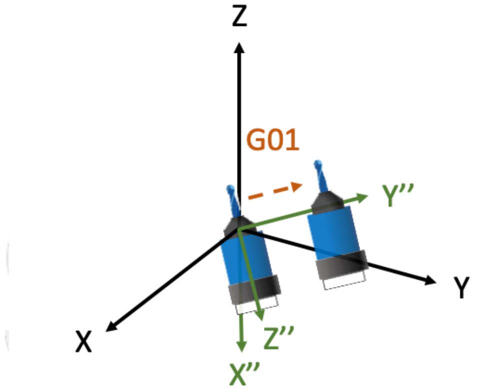

```
N1 G54 G90 G00 B0. C0.;
N2 C45.;
N3 B90.;
N4 G68.3 P1 X0. Y0. Z0.;
N5 G01 Y10.;
N6 G69;
N7 G00 X0. Y0. Z0. B0. C0.;
N8 M30;
```

Fig.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)

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;
```

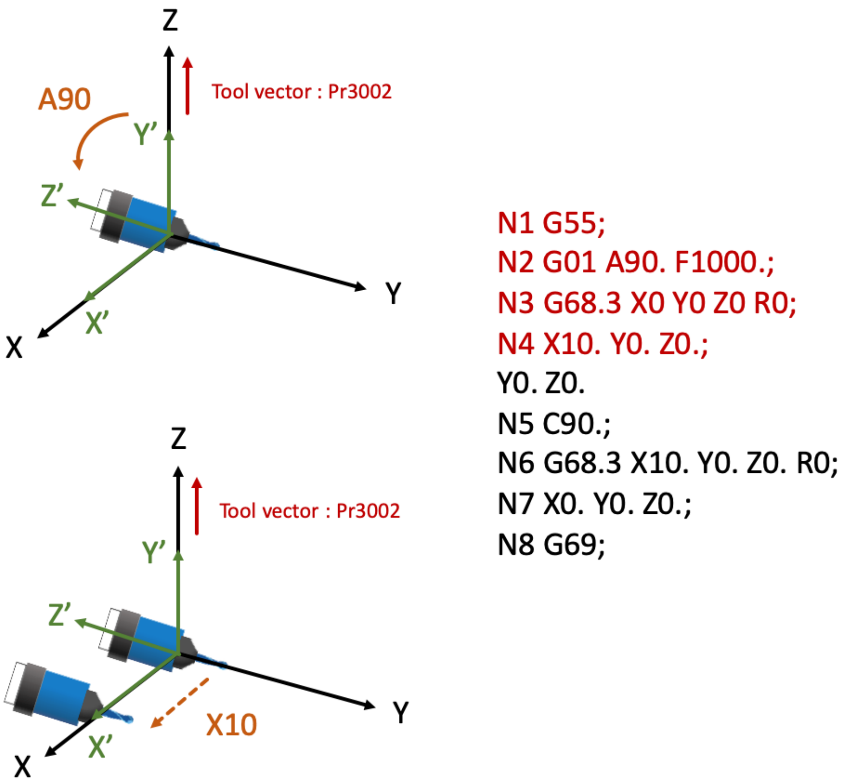
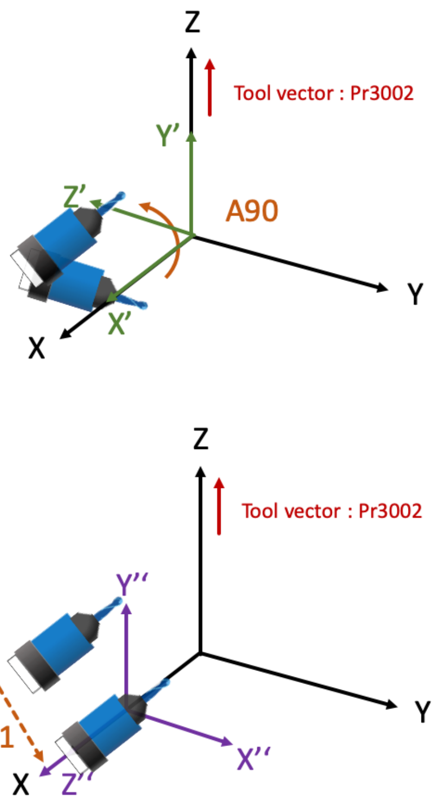


Fig.67




```
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

12.1.7 4.7 Program Example of Tilted Working Plane Machining

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.

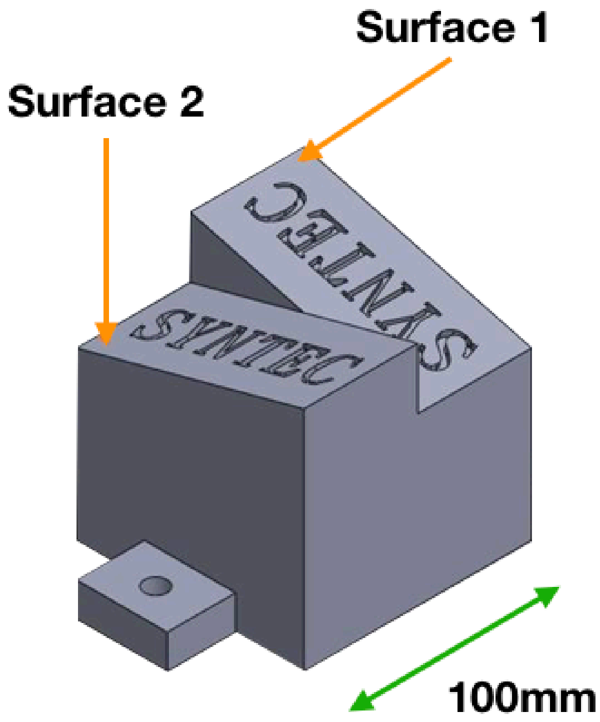


Fig.69

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
………
```

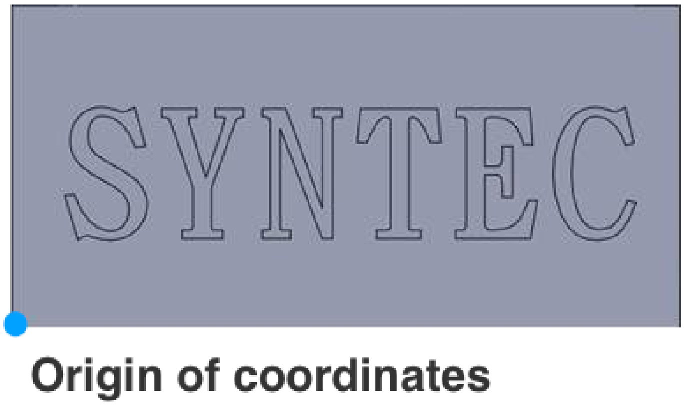


Fig.70

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
.........
```

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.

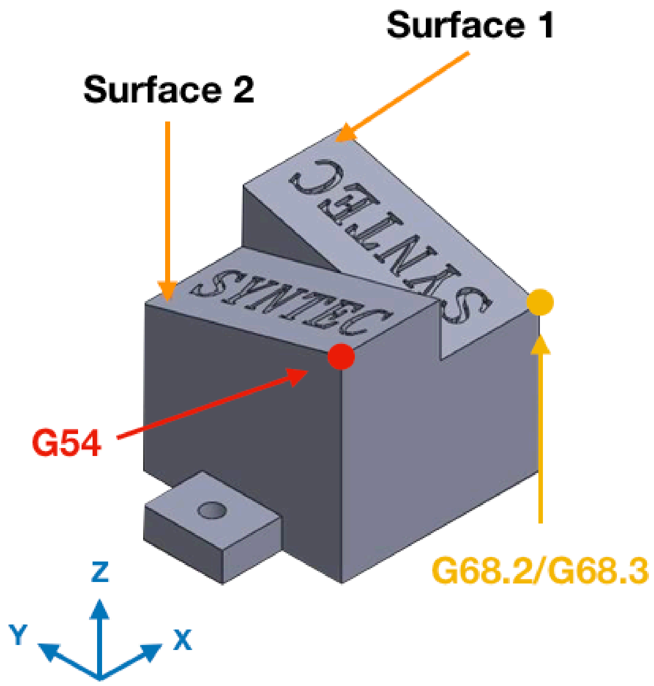


Fig.71

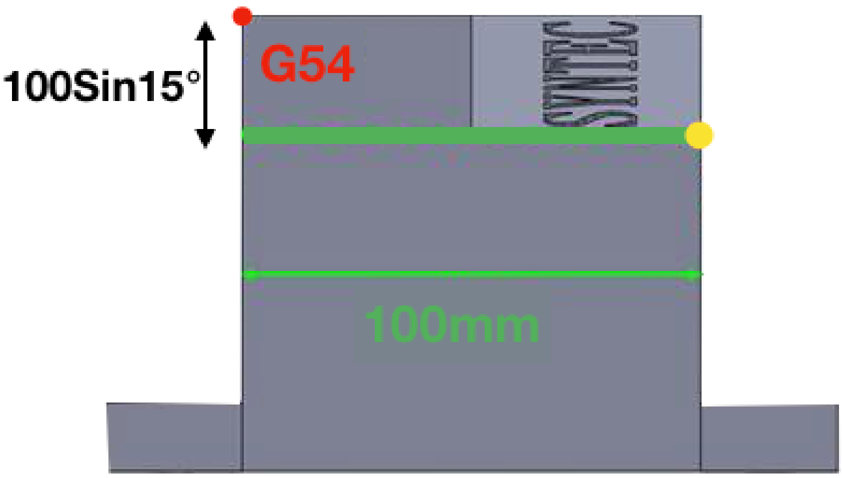


Fig.72

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.

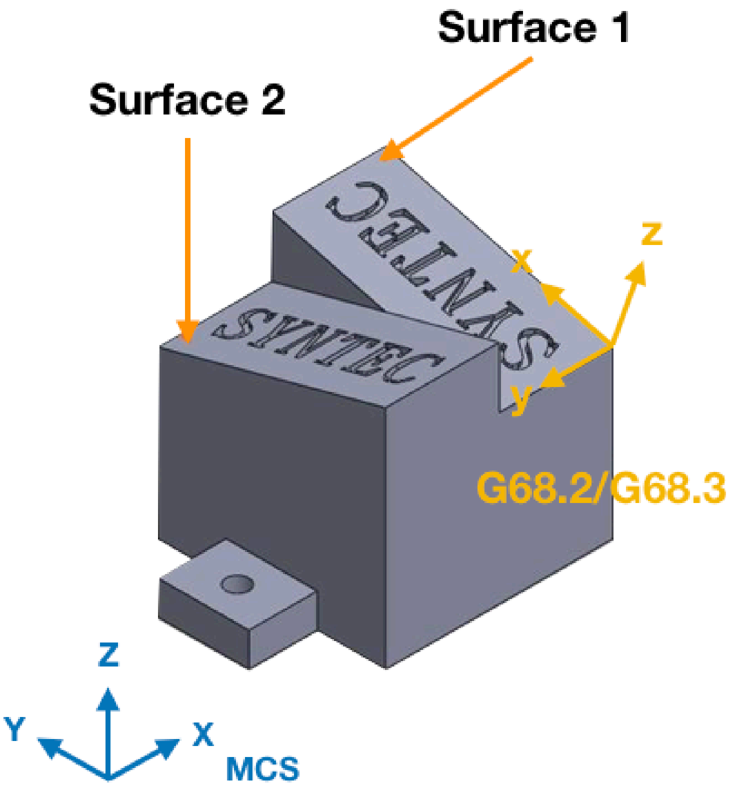


Fig.73

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.

12.2 4.1 G68.2 Tilted Working Plane Machining (Euler Angle)

12.2.1 4.1 G68.2 Tilted Working Plane Machining (Euler Angle)

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.

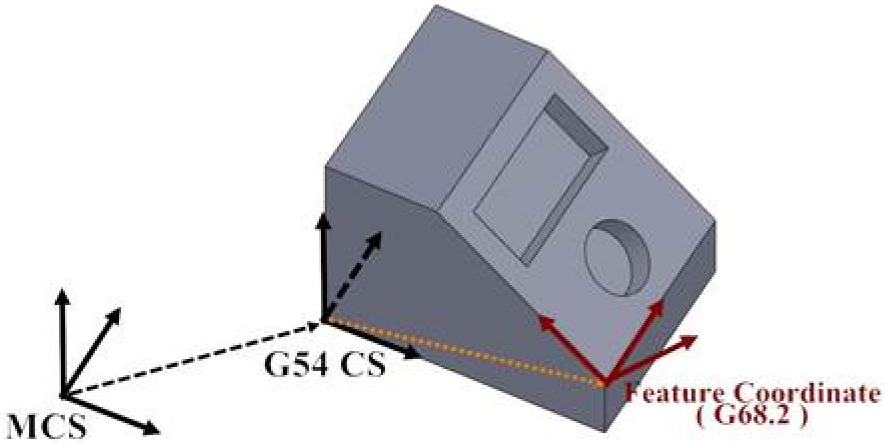


Fig.43

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.

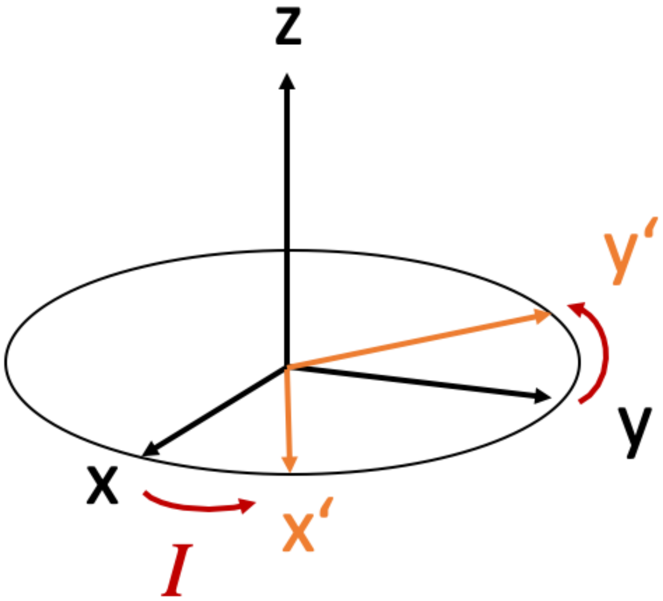


Fig.44

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.

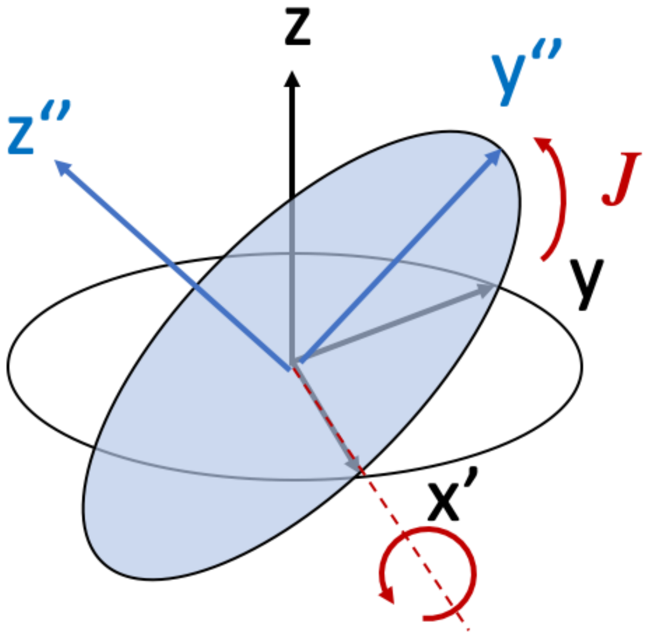


Fig.45

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.

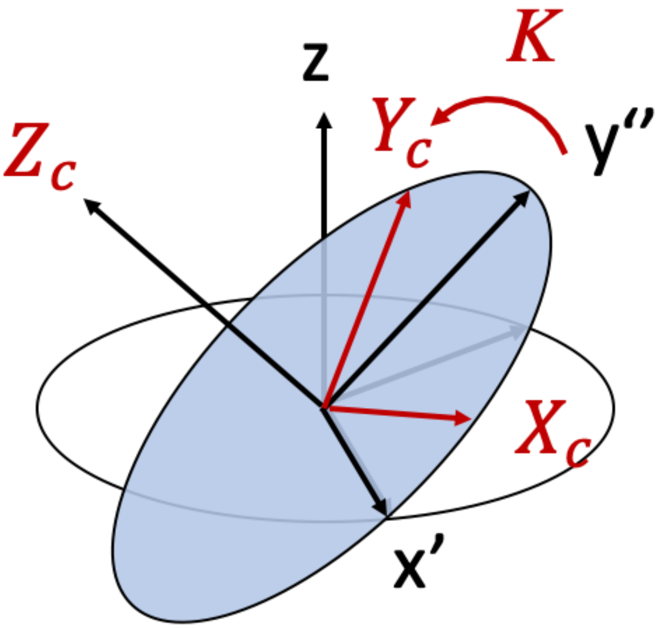


Fig.46

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;

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.

Related Parameters

No	Descriptions	Range	Unit	Details	Effective
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.	Reset

12.3 4.2 G53.1 Tool Alignment Function for Tilted Working Plane.

12.3.1 4.2 G53.1 Tool Alignment Function for Tilted Working Plane

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.

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.

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；
```

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.
```

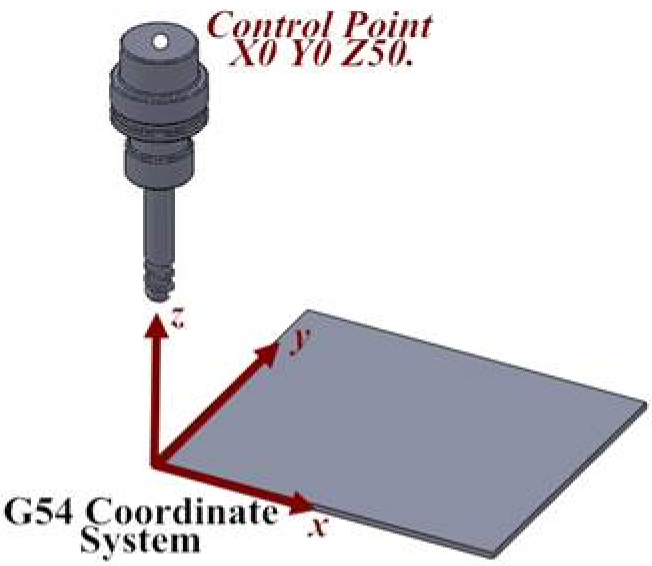


Fig.47

```
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.
```

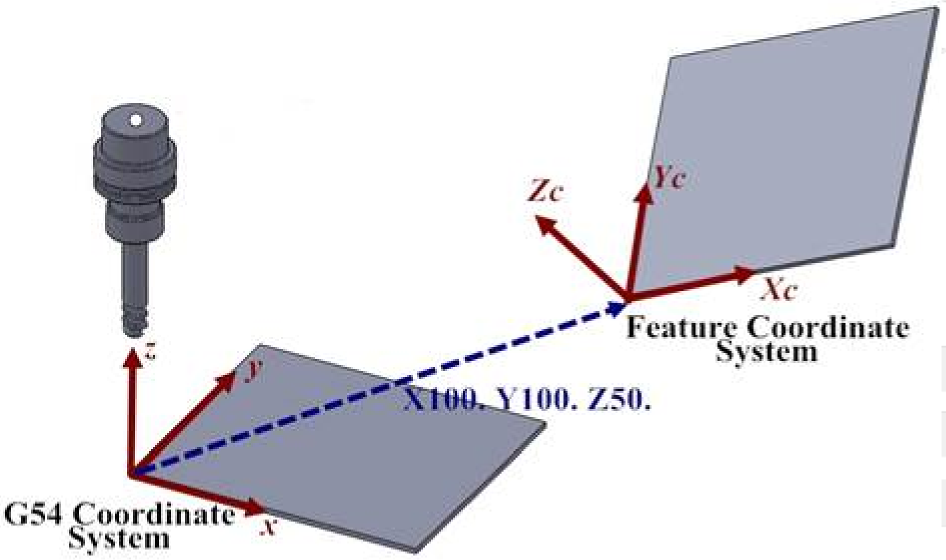


Fig.48

```
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.
```

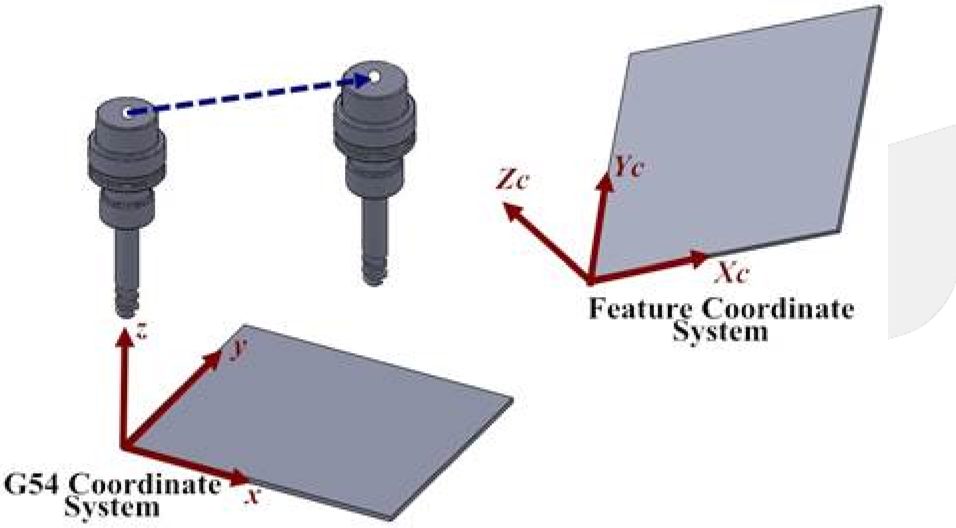


Fig.49

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

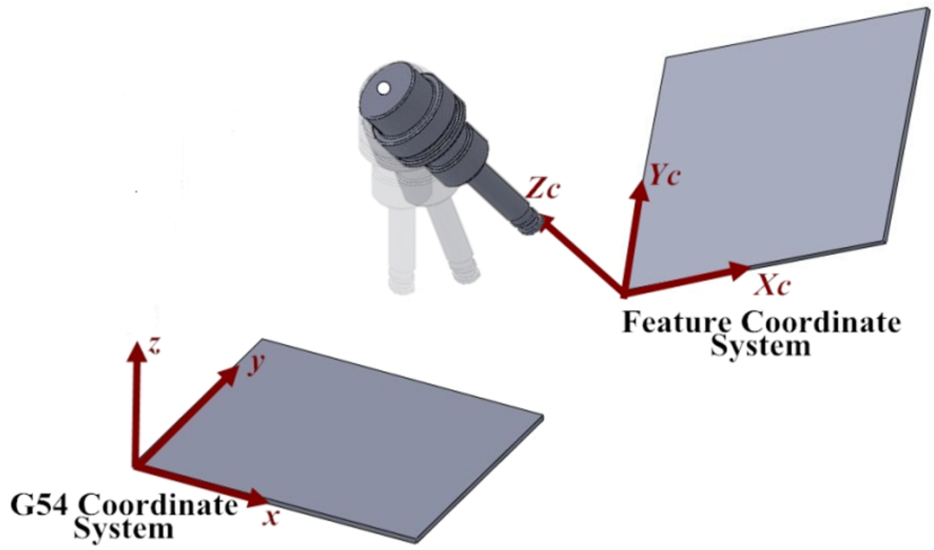


Fig.50

```
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.
```

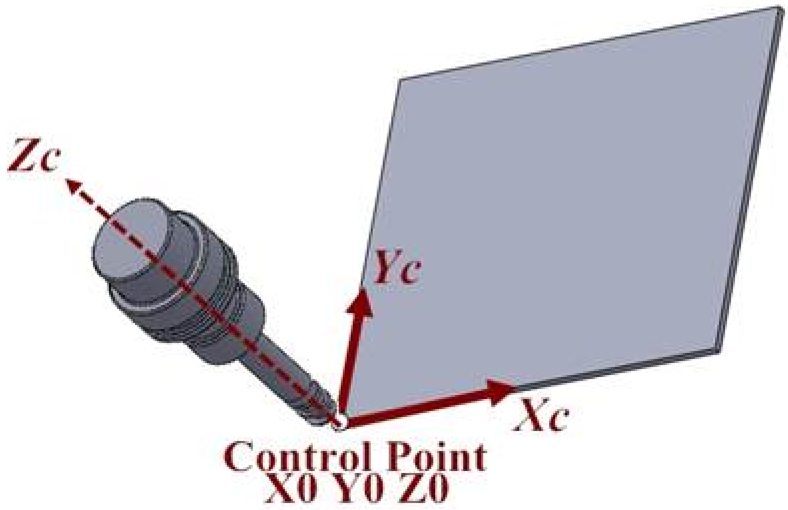


Fig.51

```
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.
```

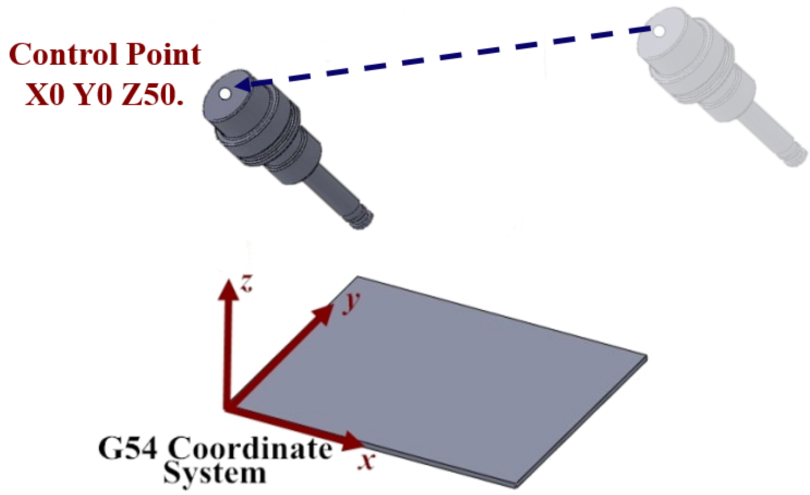


Fig.52

12.4 4.3 G53.3 Tool Alignment Function for Tilted Working Plane (5-Axis simul. motion)

12.4.1 4.3 G53.3 Tool Alignment Function for Tilted Working Plane (5-Axis simultaneous motion)

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.

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.

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

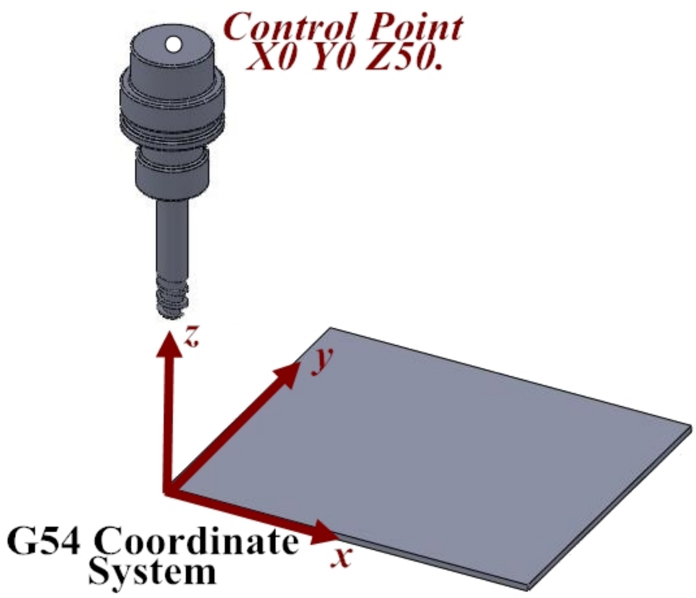
Program Example

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

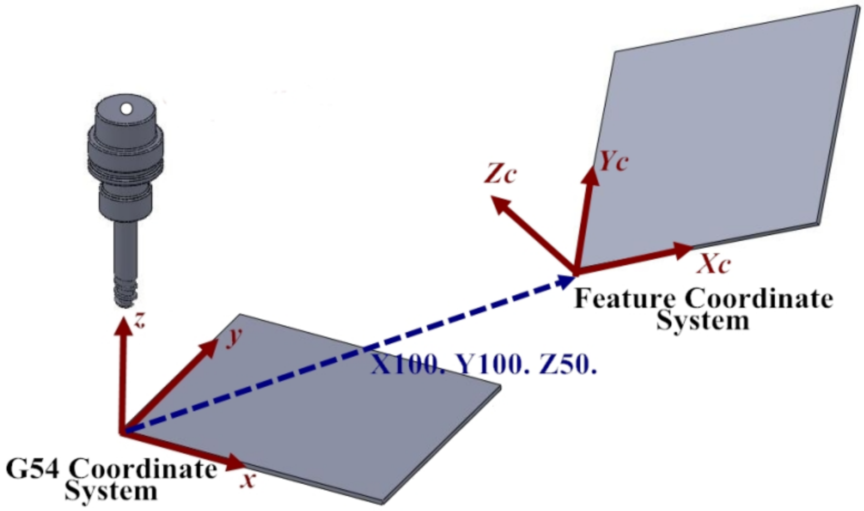
```
N1 G90 G54 G01 X0 Y0 Z50. F1000；
N2 G68.2 X100. Y100. Z50. I30. J15. K20.；
N3 G01 X0 Y0 Z50. F1000；
N4 G53.3 X0 Y0 Z0 H1；

... //  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:

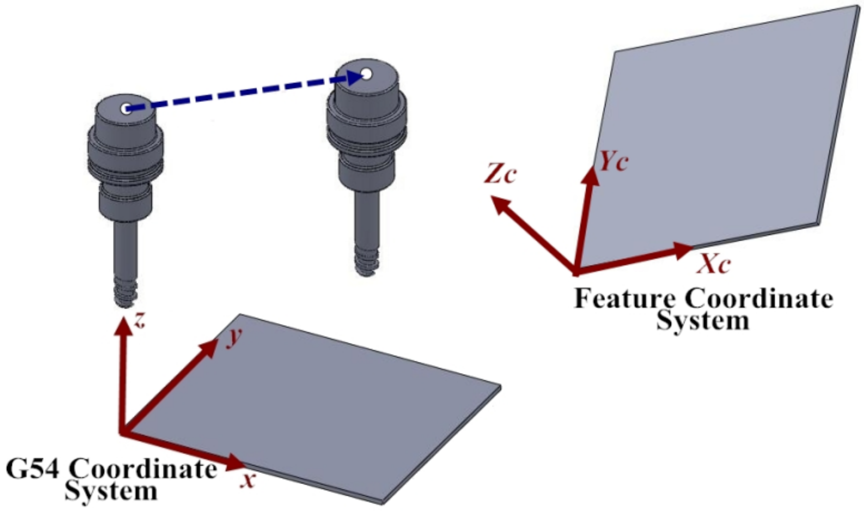
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of G54 coordinate by G01 in speed of 1000 mm/min.
```



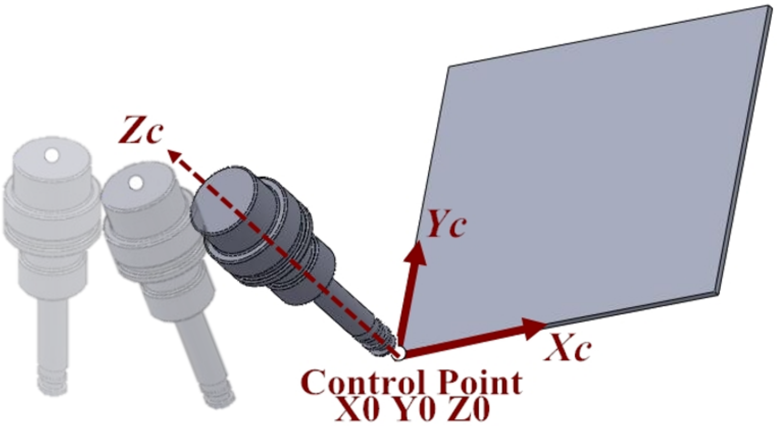
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;
// Specify X100. Y100. Z50. relative to the origin of G54 coordinate as the origin of Tilted Working Plane, and the Euler angles are I30. J15. K20.
// The program coordinate will transform to Tilted Working Plane after G68.2 is executed.
```



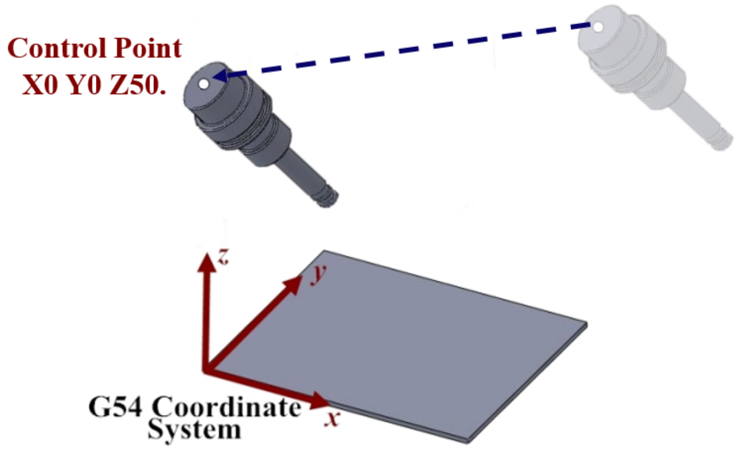

```
N3 G01 X0 Y0 Z50. F1000;
// Moves to Z50. of Tilted Working Plane by G01 in speed of 1000 mm/min, but the tool direction remains the same.
```



```
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.
```



12.5 4.4 G53.6 Tool Alignment Function for Tilted Working Plane (TCP/ Rotation Center)

12.5.1 4.4 G53.6 Tool Alignment Function for Tilted Working Plane (TCP/ Rotation Center)

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.

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.

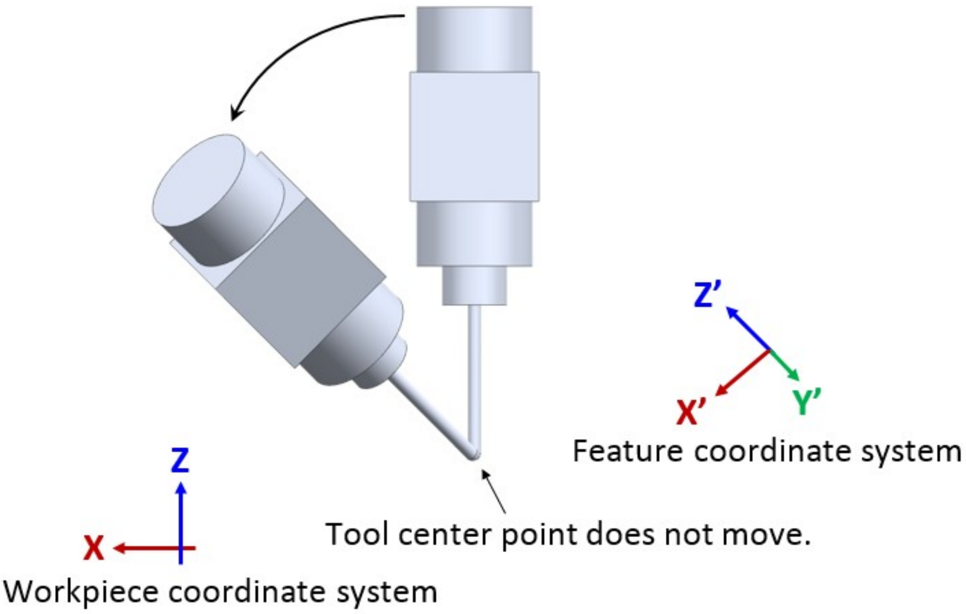


Fig.52

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

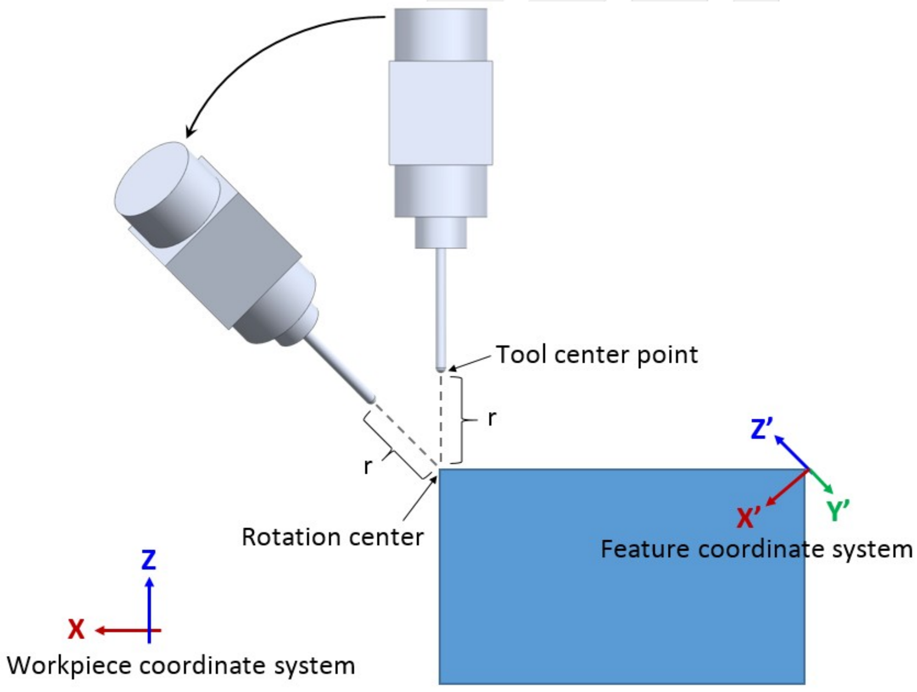


Fig.53

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

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.
```

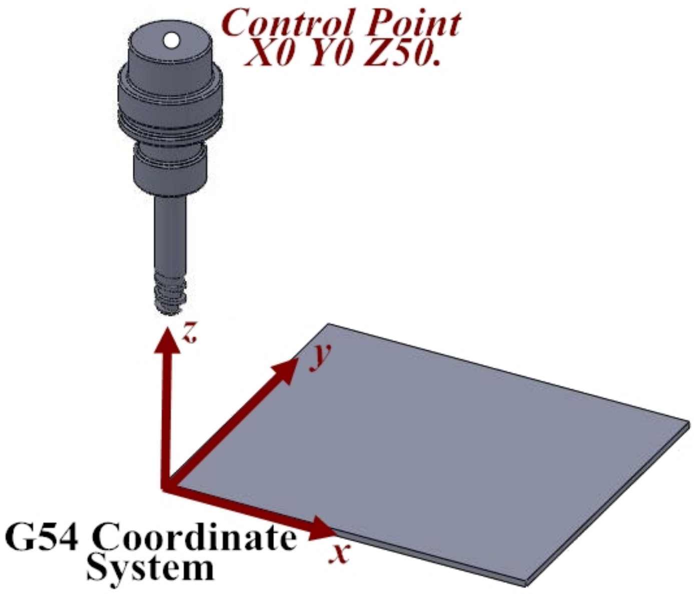


Fig.54

```
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.
```

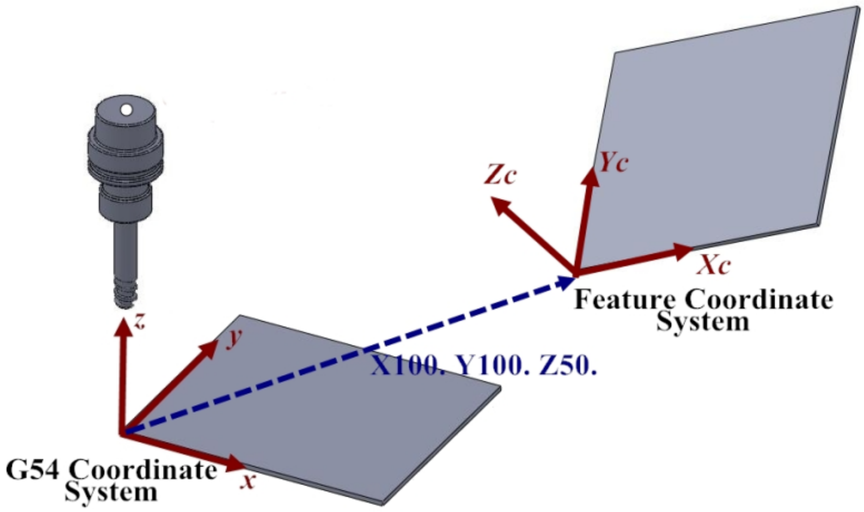


Fig.55

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

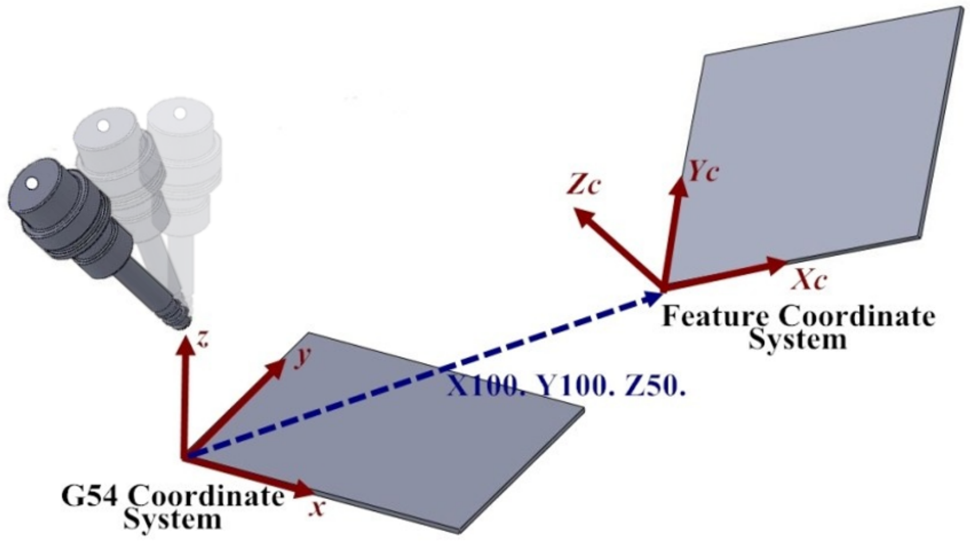


Fig.56

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

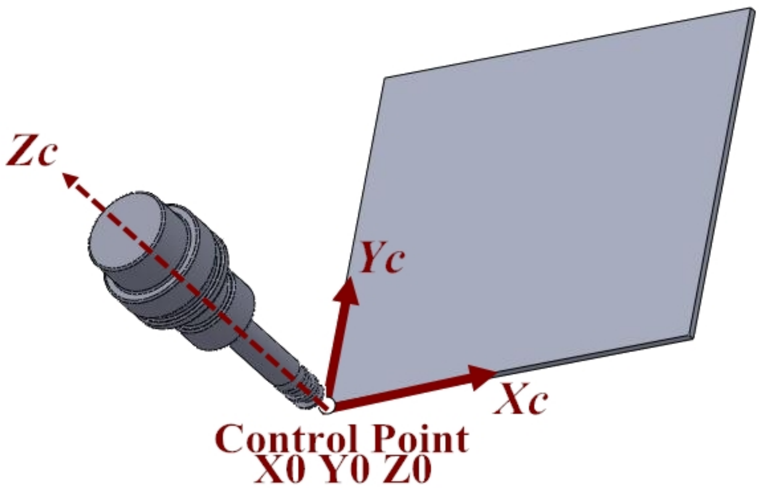


Fig.57

```
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.
```

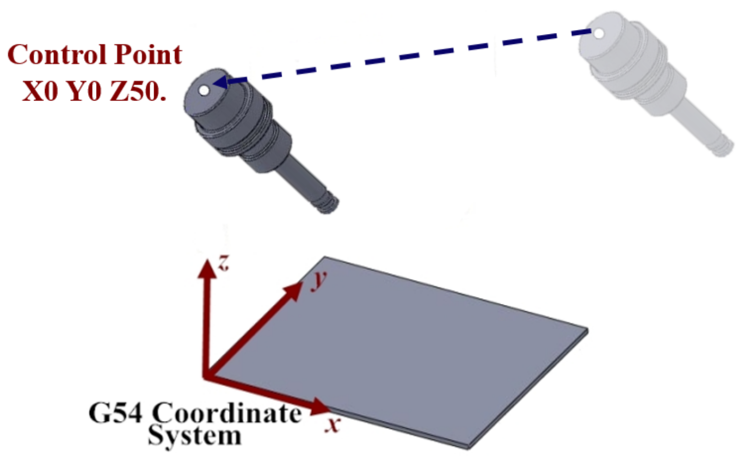
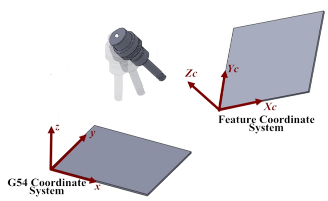
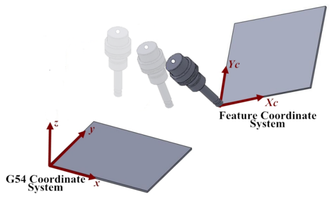
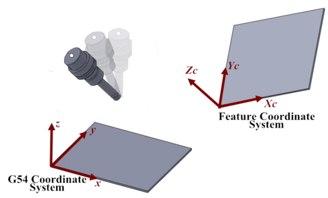


Fig.58

12.6 4.5 Comparison Between Tool Alignment Functions.

12.6.1 4.5 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

Command Format	**Description**	**Figure**	**Application**	**Valid Version**
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
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

12.7 4.6 G68.3 Tilted Working Plane Machining (Tool Direction)

12.7.1 **4.6 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.

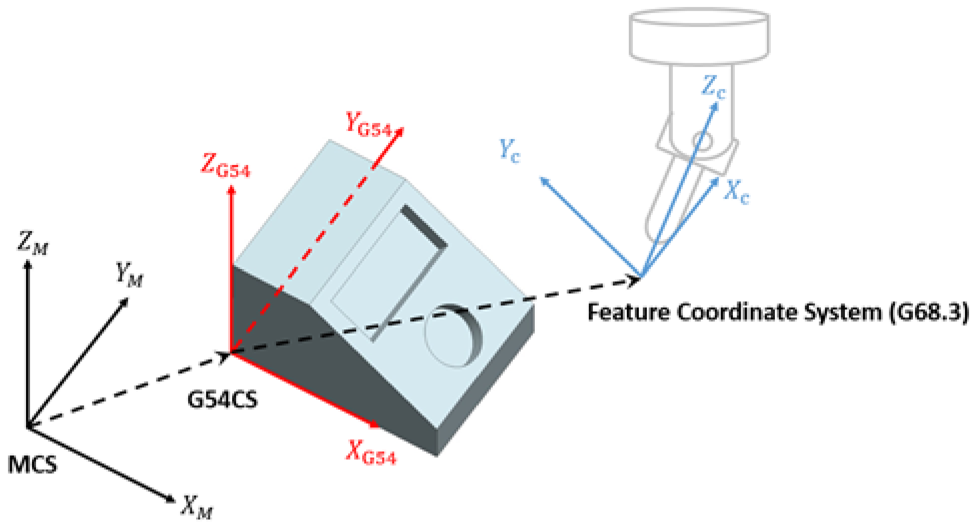


Fig.57

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***

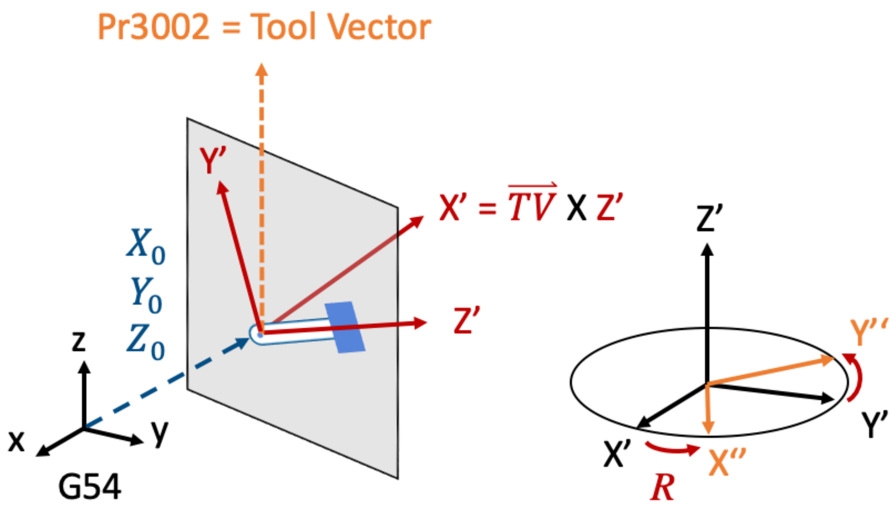


Fig.58

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***

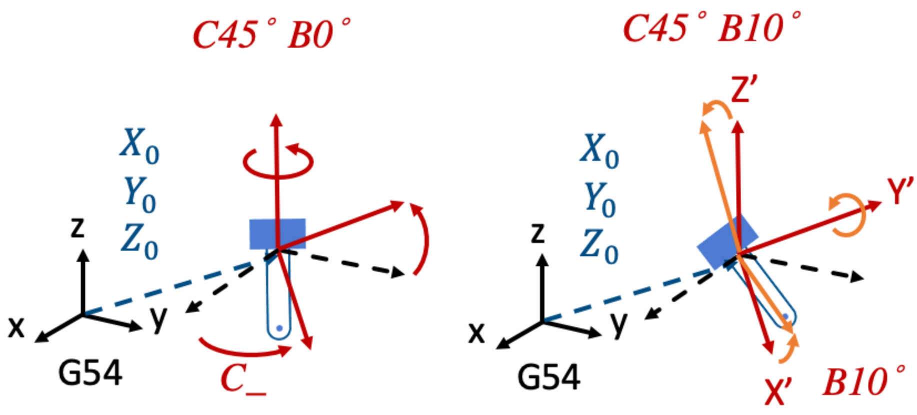


Fig.59

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. G43 should be executed after G68.3.
4. G68.3 can NOT be executed while RTCP (G43.4/G43.5) is enabled.
5. When G68.3 P1 is executed, R argument will be ignored if provided.
6. G68.3 can be executed for multiple times and each setting is relative to G54 coordinate.
7. G68.3 is only for 5-axis machines with option-13 enabled at the same time.

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.

Program Example:

(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;
```

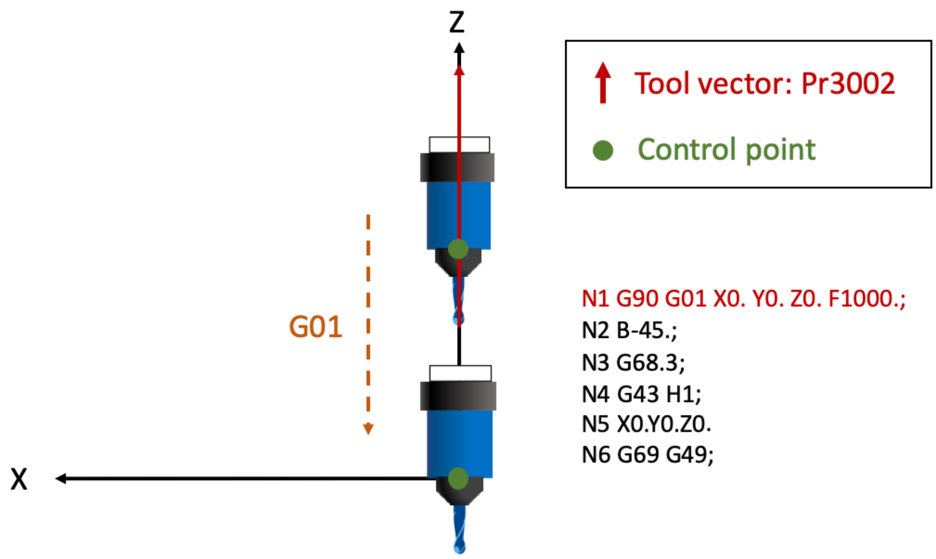


Fig.60

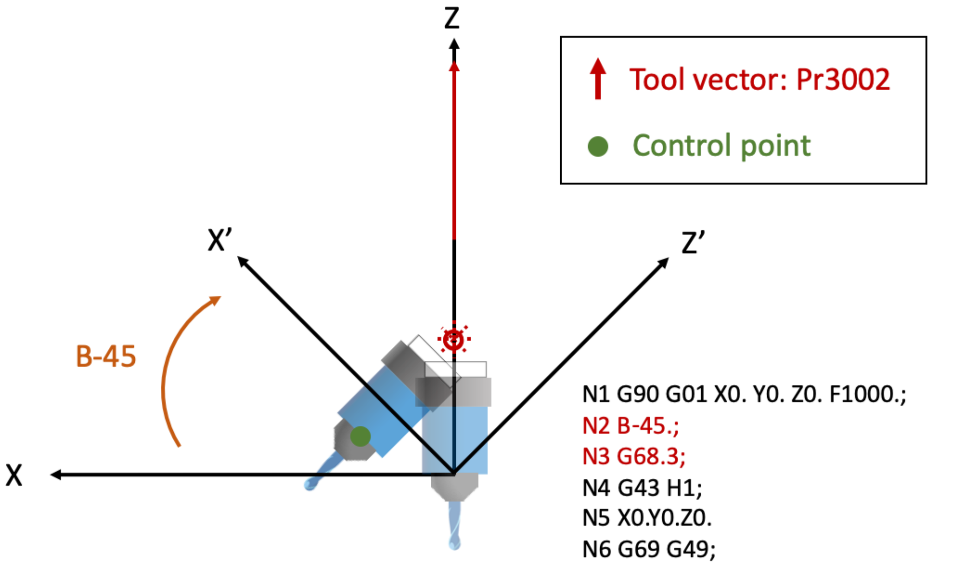

Z
Tool vector: Pr3002
Control point
X'
Z'
B-45
X
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;
N4 G43 H1;
N5 X0.Y0.Z0.
N6 G69 G49;

Fig.61

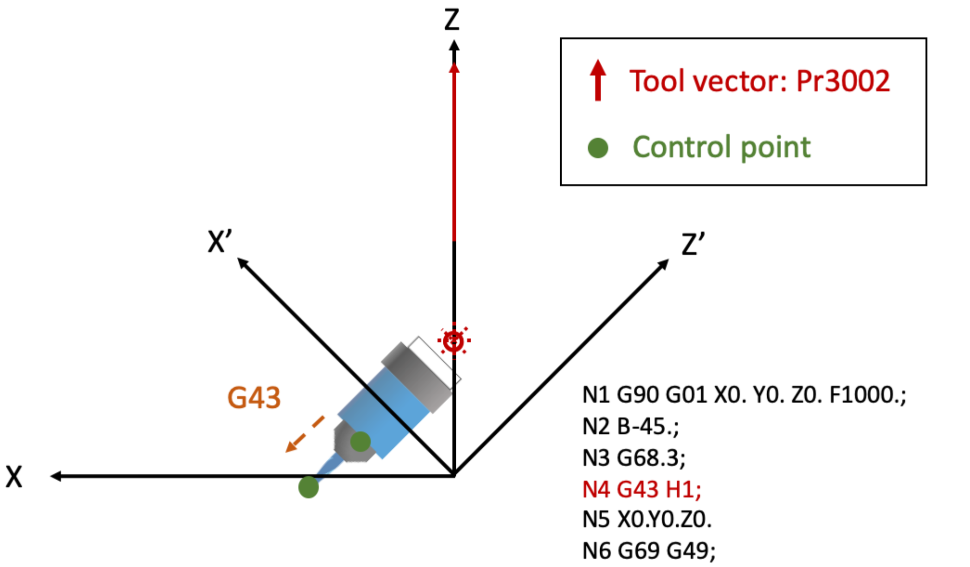

Z
Tool vector: Pr3002
Control point
X'
Z'
G43
X
N1 G90 G01 X0. Y0. Z0. F1000.;
N2 B-45.;
N3 G68.3;
N4 G43 H1;
N5 X0.Y0.Z0.
N6 G69 G49;

Fig.62

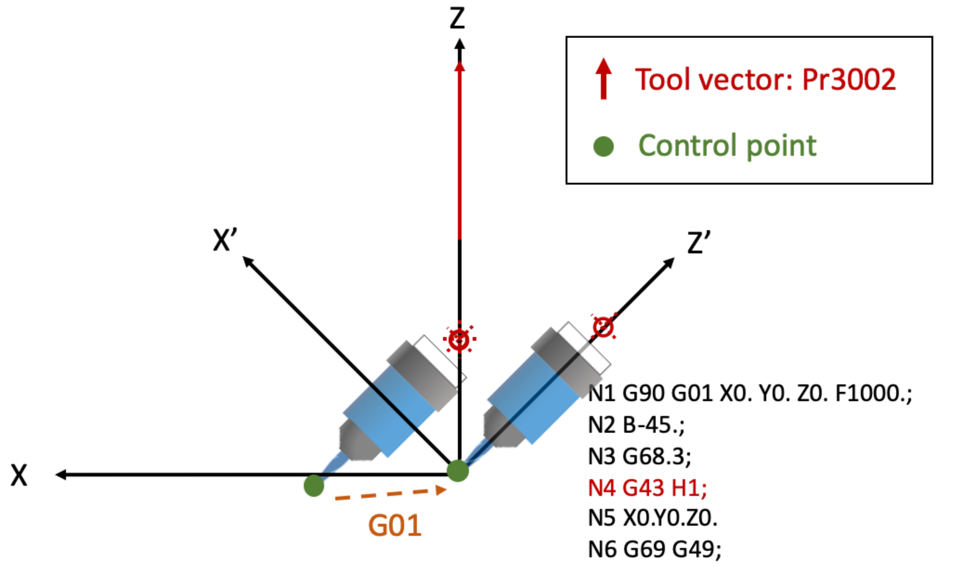
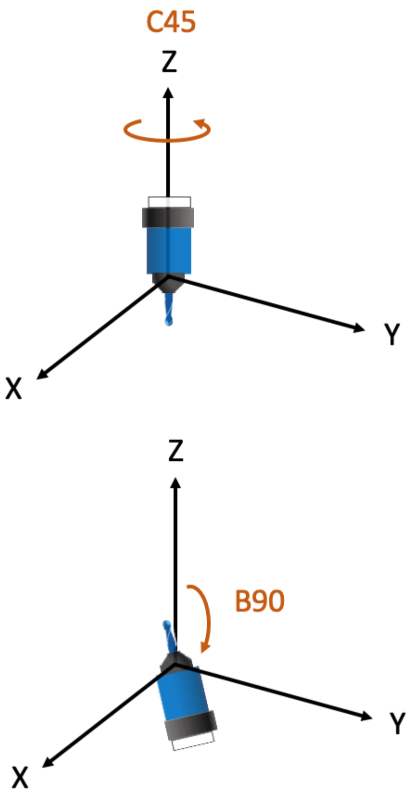


Fig.63

(2)

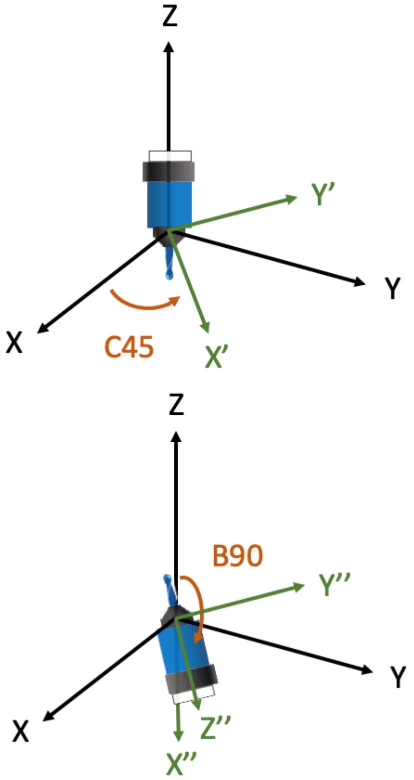
NC program and Fig.64~66 are used to explain the motions within Tilted Working Plane when G68.3 is enabled.

```
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;
```



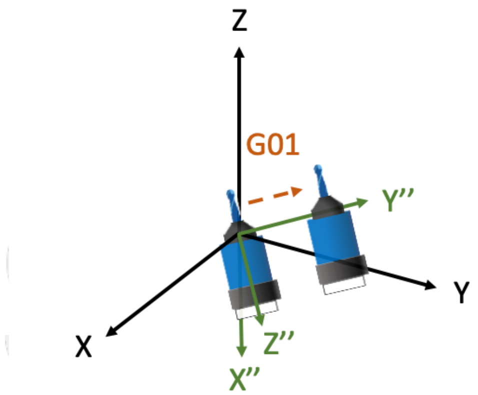

```
N1 G54 G90 G00 B0. C0.;
N2 C45.;
N3 B90.;
N4 G68.3 P1 X0. Y0. Z0.;
N5 G01 Y10.;
N6 G69;
N7 G00 X0. Y0. Z0. B0. C0.;
N8 M30;
```

Fig.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)

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;
```

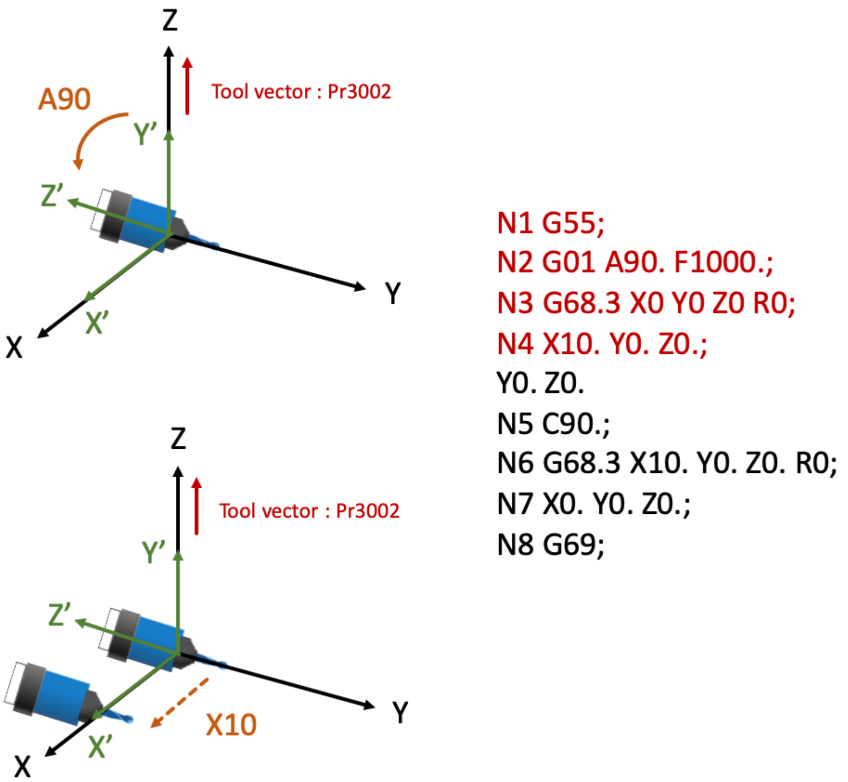
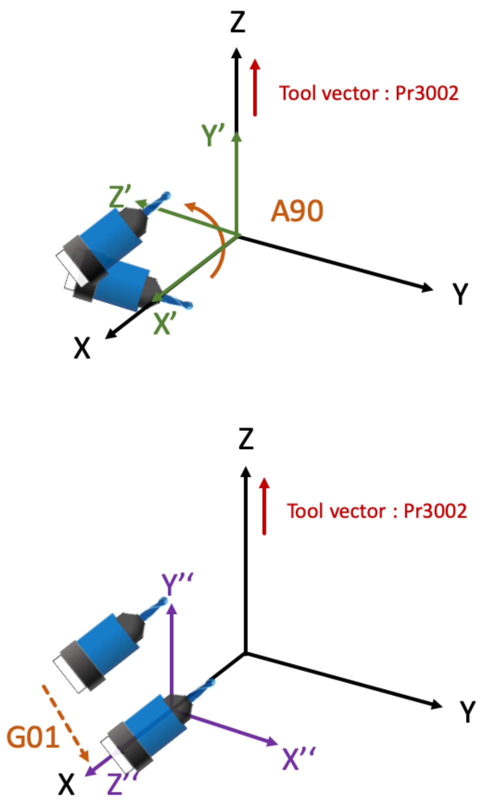


Fig.67




```
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

12.8 4.7 Program Example of Tilted Working Plane Machining.

12.8.1 4.7 Program Example of Tilted Working Plane Machining

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.

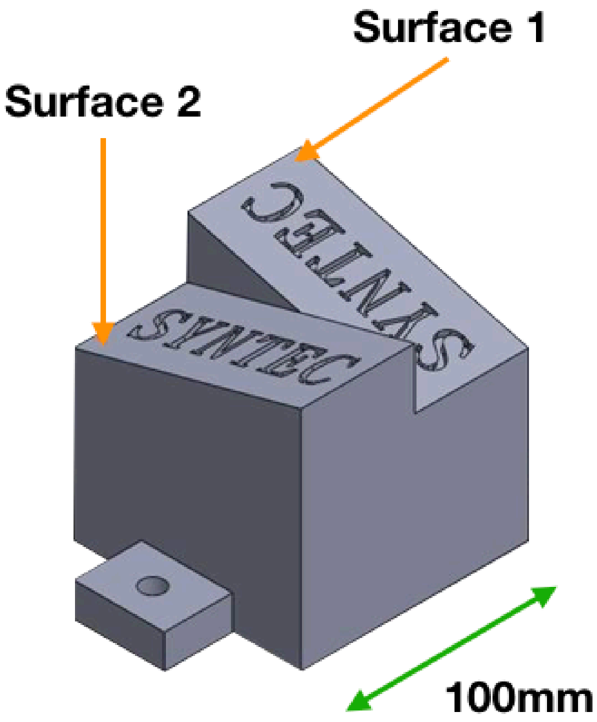


Fig.69

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
………
```

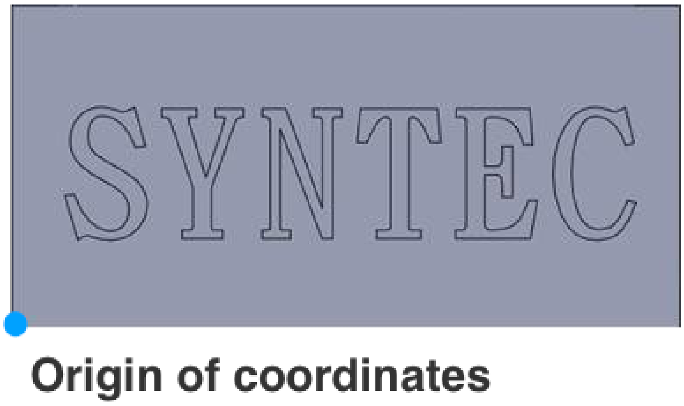


Fig.70

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
………
```

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.

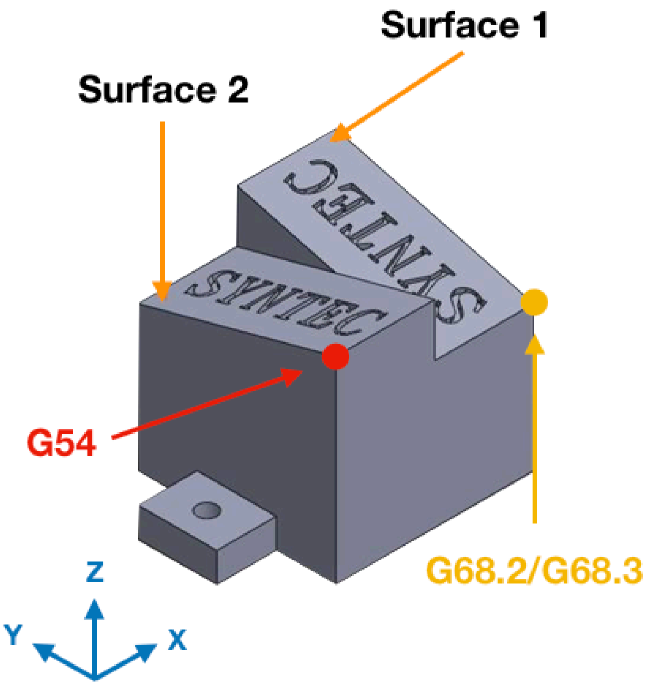


Fig.71

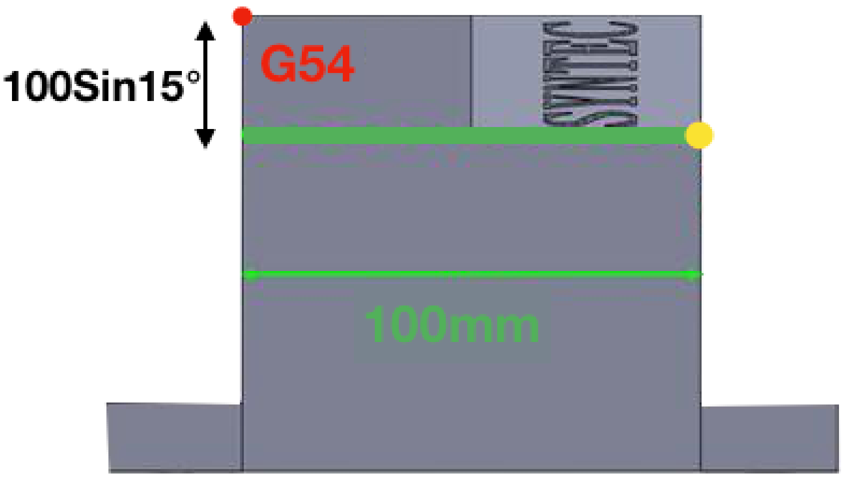


Fig.72

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.

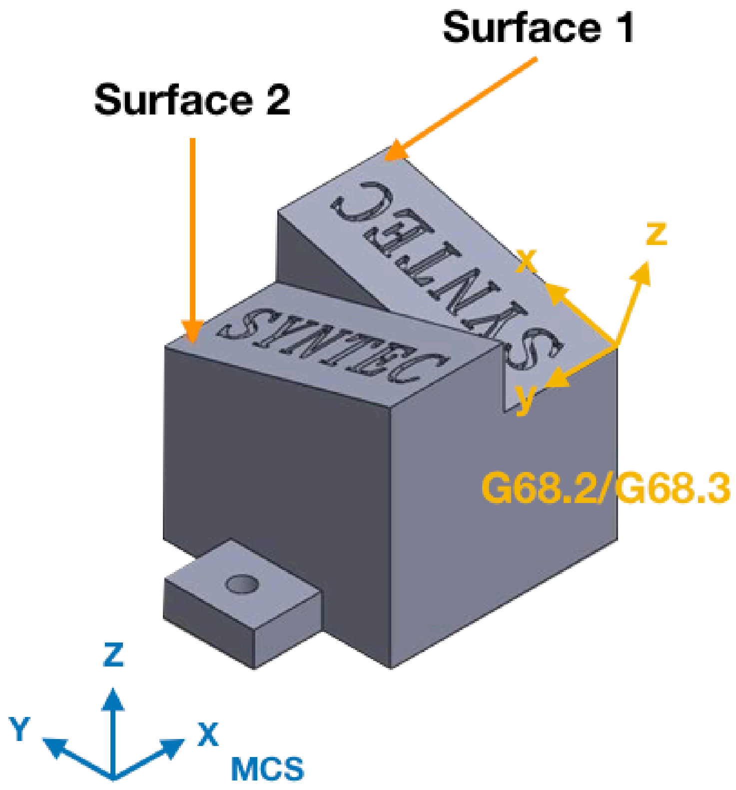


Fig.73

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.

12.9 4.8 Tilted Working Plane Related Parameters

12.9.1 4.8 Tilted Working Plane Related Parameters

Parameter Description

No	Descriptions	Range	Unit	Details	Effective

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.	Reset

13 5. Error and Compensation of 5-Axis Machine.

13.1 5. Error and Compensation of 5-Axis Machine

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

13.1.1 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.

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.

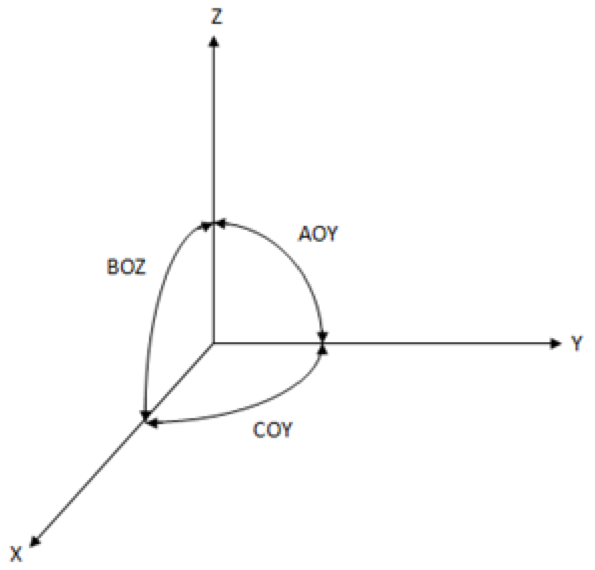


Fig.55

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.

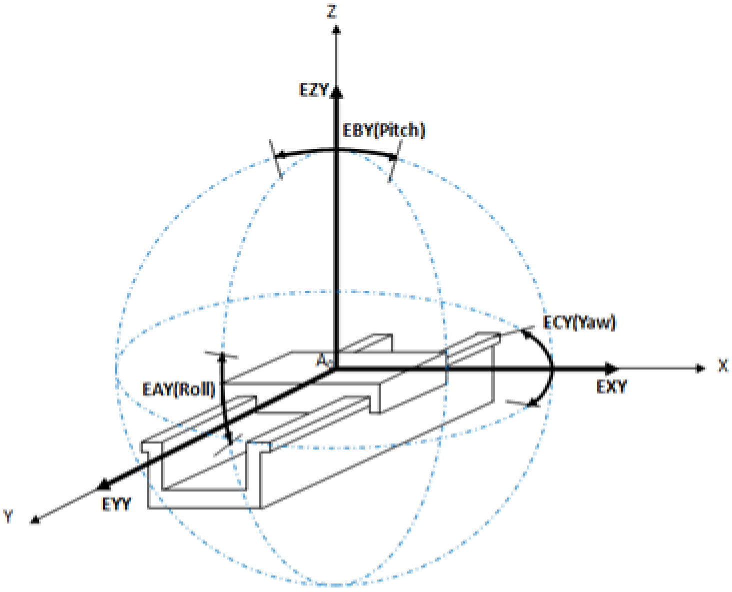


Fig.56

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.

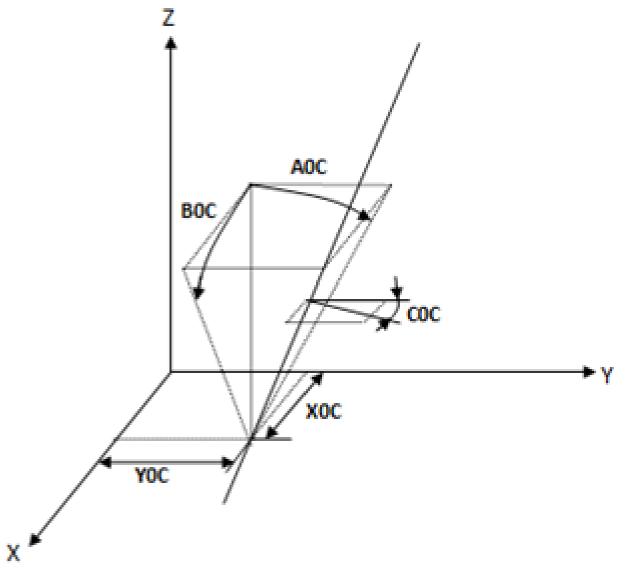


Fig.57

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.

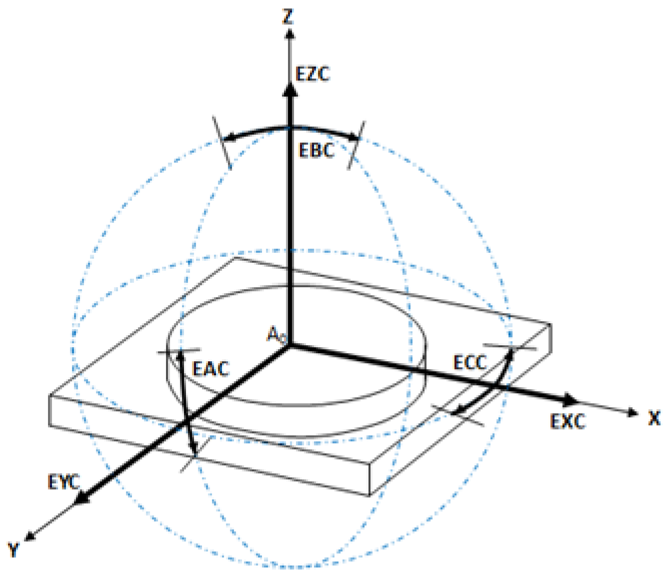


Fig.58

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

13.2 5.1 Measurememt and Compensation.

13.2.1 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.

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.

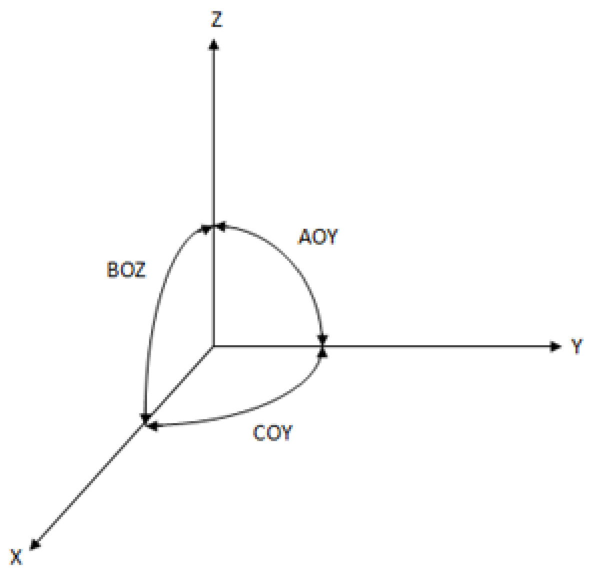


Fig.55

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.

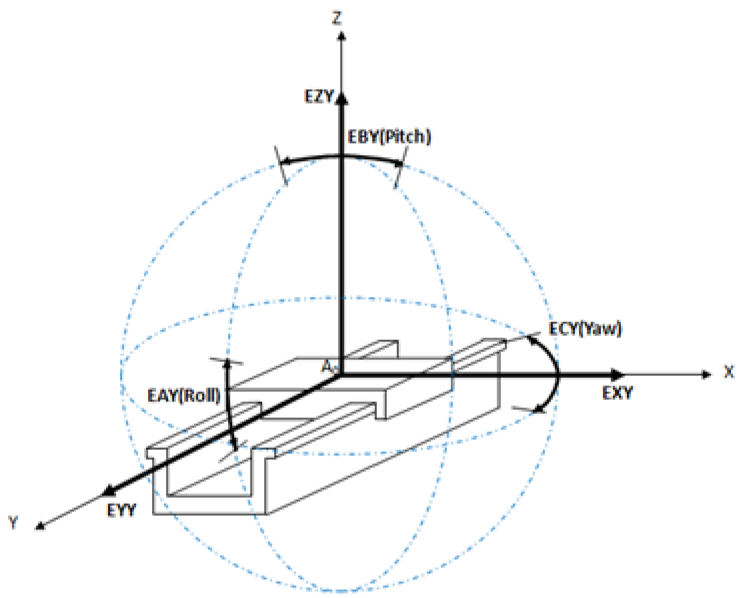


Fig.56

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.

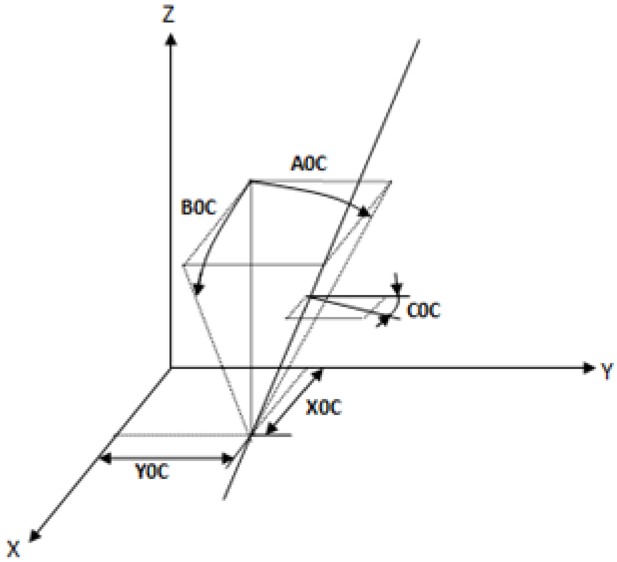


Fig.57

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.

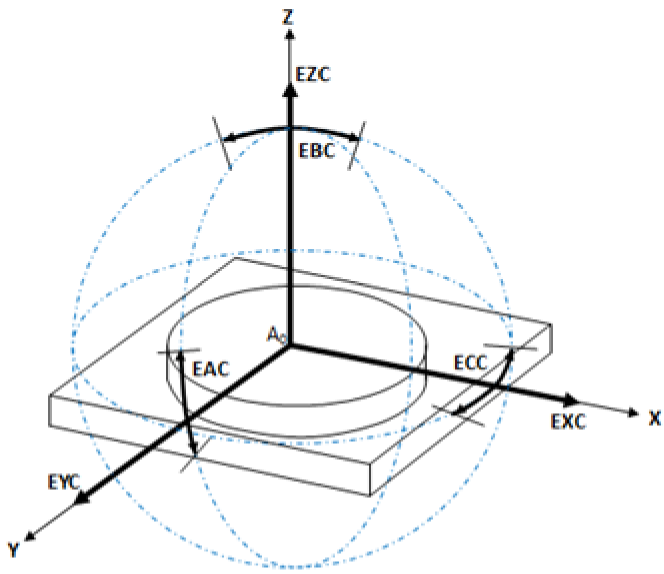


Fig.58

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

14 6. Related Alarms.

14.1 6. Related Alarms

14.1.1 COR Alarm

Alarm ID	**COR–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**	**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 Woodworking label 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.		
Solution	1-4. Please contact administrator. 5. Do not use Loader path and Woodworking label path to do process operation. 6. Set Pr3802 to 0.		

Alarm ID	**COR-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.		

Alarm ID	COR-107	Alarm Title	G5.1/G05 command format error
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	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.		
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	Alarm Title	Prohibit G05 in tool tip control mode
Description	Turn on G05 high-speed high-precision mode in the RTCP/STCP mode.		
Possible Cause	In the RTCP/STCP mode, turn on the G05 high-speed high-precision mode with commands, such as G05 P10000.		

Alarm ID	COR-140	Alarm Title	Prohibit G05 in tool tip control mode

Solution	Check the mode to be turned on is (1) RTCP/STCP mode or (2) G05 high-speed high-precision mode.
	If (1), remove the command to turn on the G05 high-speed high-precision mode in the RTCP/STCP mode.
	If (2), turn off the RTCP/STCP mode before turning on the G05 high-speed high-precision mode.		

Alarm ID	COR-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	Alarm Title	1st rotation axis entering illegal range
Description	1st rotation axis entering illegal range.		
Possible Cause	1. Pr3007, Pr3009, or Pr3010 configuration error. 2. The angle of 1st 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	Alarm Title	2nd rotation axis entering illegal range
Description	2nd rotation axis entering illegal range		

Alarm ID	COR-152	Alarm Title	2nd rotation axis entering illegal range
Possible Cause	1. Pr3008, Pr3011 or Pr3012 configuration error. 2. The angle of 2nd 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	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	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	Alarm Title	5-axis tool direction error
Description	5-axis tool direction error.		
Possible Cause	5-axis tool direction (Pr3002) or the 1st and 2nd 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 2nd rotation axis is parallel to the Spindle in the Spindle type, or the 1st rotation axis is parallel to the Spindle in the workbench type.		

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

Alarm ID	**COR-162**	**Alarm Title**	**4-axis RTCP configuration error**
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**	**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**	**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**	**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.		
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, ...		

Alarm ID	**COR-165**	**Alarm Title**	**Multi-kinematic chain command not illegal.**
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**	**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**	**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.		
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		

14.1.2 OP Alarm

Alarm ID	**OP-032**	**Alarm Title**	**Mechanism type configuration conflicted**
Description	Pr3201 Machine Type setting conflicted.		

Alarm ID	**OP-032**	**Alarm Title**	**Mechanism type configuration 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.		
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).		

14.1.3 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.

14.2 6.1 Alarm List

14.2.1 6.1 Alarm List

- Coordinate alarm 70：Invalid G Code
- Coordinate alarm 100：Unsupported G Code Command or Option Software is Not Activated
- Coordinate alarm 107：G5.1/G05 Command Format Error
- Coordinate alarm 118：Prohibit G53 in RTCP Mode
- Coordinate alarm 140：Prohibit G05 in RTCP Mode
- Coordinate alarm 141：G68.3 Command Format Error
- Coordinate alarm 151：1st Rotation Axis Entering Illegal Range
- Coordinate alarm 152：2nd Rotation Axis Entering Illegal Range
- Coordinate alarm 153：Tool Direction Unknown
- Coordinate alarm 154：No 5-Axis Function
- Coordinate alarm 155：5-Axis Tool Direction Error
- Coordinate alarm 156：5-Axis Axial Direction Error
- Coordinate alarm 157：Incompatible direction of 5-Axis Tool Direction and This of Rotation Axis
- Coordinate alarm 158：Prohibit the 1st and 2nd Rotary Axis Commands in G43.5 Mode
- Coordinate alarm 159：Illegal Tool Vector
- Coordinate alarm 160：5-Axis Mechanism Switched when the 5-Axis Function is On
- Coordinate alarm 161：Selected 5-Axis Mechanism Chain is Not On
- Coordinate alarm 162：4-Axis RTCP Configuration Error
- Coordinate alarm 163：Multi-Kinematic Chain Command Q Argument Setting Error
- Coordinate alarm 164：Multi-Kinematic Chain Command Related 5-Axis Mechanism Setting Error
- Coordinate alarm 165：Multi-Kinematic Chain Command Not Legal
- Coordinate alarm 305：Relative Position Input Method is Forbidden in Current Mode
- Macro alarm 406：G53.6 Needs to be Enabled in G40 Mode
- Macro alarm 407：The Selected Tool Number of G53.6 Cannot Be 0
- Operation alarm 32：Mechanism Type Configuration Conflicted

14.3 6.2 Alarm Solution

14.3.1 6.2 Alarm Solution

Coordinate Alarms

- Alarm 70：Check if Pr3001 is set correctly.
- Alarm 100：Check if the 5-axis option (Op12, OP13, OP27) is enabled. If alarm still on with option enabled, check if the machine module is 200MA-5.
- Alarm 107：In STCP mode, only the RTCP 5 axis commands are allowed for moving commands, this alarm will be triggered if non-RTCP axis command is included (i.e., 6th axis movements). When this alarm is triggered, modify the program and disable STCP before moving the other non-RTCP axis.
- Alarm 118：Check the programming, make sure command G53 is not in the effective range of RTCP mode (i.e.：G43.4, G43.5).
- Alarm 140：Check the programming, male sure command G05 is not in the effective range of RTCP mode(i.e.：G43.4, G43.5).
- Alarm 141：Check if applying G68.3 or G68.3 P1 command, the argument can only be 1 if applying P argument; the XYZ arguments should exist at the same time when applying.
- Alarm 151：Check if Pr3009, 3010 is set correctly, the recognition of these 2 parameters is related to Pr3007. If alarm is triggered, it requires to reconfirm the setting of these 3 parameters.

- Alarm 152：Check if Pr3011, 3012 is set correctly, the recognition of these 2 parameters is related to Pr3008. If alarm is triggered, it requires to reconfirm the setting of these 3 parameters.
- Alarm 153：Tool unable to reach the target position, might be 5-axis parameters not matching the machine mechanism, please check all the 5-axis parameters.
- Alarm 154：Pr3001 not set when using G53.1 tool alignment command or G68.3 feature coordinate command.
- Alarm 155：Check if Pr3002 is set correctly.
- Alarm 156：Check if each axis is completely set (Pr21~), if Pr3005, 3006, 3007, 3008 is set correctly, if the axis name (Pr 321~) is corresponded to Pr3005, 3006.
- Alarm 157：Check if Pr3002, 3005, 3006 is set wrong, this alarm will be triggered when the 2nd rotary axis of spindle type machines or the 1st rotary axis of table type and mix type machines is parallel to the spindle.
- Alarm 158：Check the programming, make sure the 1st/2nd rotary axis commands are not in the effective range of G43.5 mode.
- Alarm 159：Check the programming, make sure tool vector values I, J, K are input correctly, this alarm will be triggered if it's I0. J0. K0.
- Alarm 160：Check the machining program and make sure 5-axis function is disabled for the alarmed block.
- Alarm 161：Check the 5-axis mechanism chain parameters and make sure the specified 5-axis mechanism chain function is enabled properly：1st set, check Pr3001; 2nd set, check Pr3101; 3rd set, check Pr5501; 4th set, check Pr5601.
- Alarm 162：
    a. Please correctly set the 5-axis mechanism parameter Pr3001, Pr3101, Pr5501, Pr5601 to 4 or 5.
    b. Please enable RTCP function (option-12)
- Alarm 163：Apply command G10 L5000 P_ Q_, please check the Q argument is between 0~4 and is an integer.
- Alarm 164：
    a. Please check the specified 5-axis mechanism chain, the 5-axis mechanism parameter should be spindle type 5-axis machine.
        i. 1st set, check if Pr3001 is 1
        ii. 2nd set, check if Pr3101 is 1
        iii. 3rd set, check if Pr5501 is 1
        iv. 4th set, check if Pr5601 is 1
- Alarm 165：
    a. Command G10 L5000 P_ Q_, Q argument set 2~4(non-1st set sub-mechanism chain) only supports the 5-axis machine function commands below.
        i. RTCP：G43.4。
        ii. RTCP：G43.5。
        iii. Tilted Working Plane：G68.2 + G53.1。
        iv. Tilted Working Plane：G68.2 + G53.6。
        v. Tilted Working Plane：G68.3。
- Alarm 305：Check the programming, make sure G91 is not executed in G43.5 mode and neither G43.5 is executed in G91 mode.

Macro Alarms

- Alarm 406：Check the programming, make sure the cutter radius compensation state is in G40 mode before executing G53.6.
- Alarm 407：Check the programming, make sure it's carrying the H argument which should be a non-0 tool number when executing G53.6; or not carrying the H argument but already applying tool length compensation with non-0 tool number.

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

15.1 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.

15.1.1 **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.

15.1.2 **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.

16 8. Q&A (5-axis manual)

16.1 8. 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.