



5-AXIS MACHINING FUNCTIONS

Scope of the Document

This document is a collection of Technical Reports, Specifications and content from manuals that was officially released by FANUC. The source documents are listed in the Appendix.

Only functions for 30i and 31i-A5 are described. There might be differences to 16i / 18i-A5.

This manual does not replace official FANUC documentation.



FANUC

Contents

I 5-AXIS MACHINES.....	3
1 Classification.....	4
II SIMULTANEOUS 5-AXIS MACHINING.....	6
1 Overview.....	7
2 Tool Center Point Control and Cutting Point Control.....	8
2.1 Overview.....	8
2.2 Programming.....	12
2.3 Explanations.....	20
2.4 Programming Examples.....	44
2.5 Limitations.....	49
3 Cutter Compensation for 5-axis machining.....	51
3.1 Cutter Compensation in Tool Rotation Type Machine.....	51
3.2 Cutter Compensation in Table Rotation Type Machine.....	70
3.3 Cutter Compensation in Mixed-Type Machine.....	75
3.4 Interference check and interference avoidance.....	80
3.5 Restrictions.....	84
3.6 Examples.....	90
4 NURBS for 5-axis machining.....	95
4.1 Programming.....	95
4.2 Explanations.....	96
4.3 Limitations.....	96
5 Nano smoothing for 5-axis machining.....	97
5.1 Programming.....	97
5.2 Explanations.....	98
5.3 Limitations.....	99
III TILTED WORKING PLANE.....	100
1 Tilted Working Plane Command.....	101
1.1 Overview.....	101
1.2 Programming.....	103
1.3 Explanations.....	113
1.4 Restrictions.....	115
2 Workpiece Setting Error Compensation.....	116
2.1 Programming.....	116
2.2 Explanation.....	116
2.3 Setting Operations.....	121
2.4 Examples.....	142
2.5 Restrictions.....	147
IV MANUAL OPERATIONS.....	153
1 Manual Intervention during Tool Center Point.....	154
1.1 Explanation.....	154
1.2 Table rotation type and Mixed type machine.....	155
2 Manual Feed for 5-Axis Machining.....	156
2.1 Tool Axis Direction Handle Feed/Tool Axis Direction JOG Feed/Tool Axis Direction Incremental Feed.....	156
2.2 Tool Axis Right-Angle Direction Handle Feed/Tool Axis Right-Angle Direction JOG Feed/Tool Axis Right-Angle Direction Incremental Feed.....	158
2.3 Tool Tip Center Rotation Handle Feed/Tool Tip Center Rotation JOG Feed/Tool Tip Center Rotation Incremental Feed.....	161
2.4 Table Vertical Direction Handle Feed/Table Vertical Direction JOG Feed/Table Vertical Direction Incremental Feed.....	164
2.5 Table Horizontal Direction Handle Feed/Table Horizontal Direction JOG Feed/Table Horizontal Direction Incremental Feed.....	165
V APPENDIX.....	170
1 Parameters.....	171
2 Documentation Reference.....	190
3 Document History.....	191

I 5-AXIS MACHINES

1 Classification

Five axis machining is defined by five axes of motion enabling five degrees of freedom.

Most five axes applications are configured with 3 linear axes which are situated in orthogonal, Cartesian coordinate system and two rotary axes. The configuration of the two rotary axes can be classified into three basic types:

1. Rotary tool type (head-head configuration of rotary axes),
2. Rotary table type (table-table configuration of rotary axes),
3. Mixed type (head-table configuration of rotary axes)

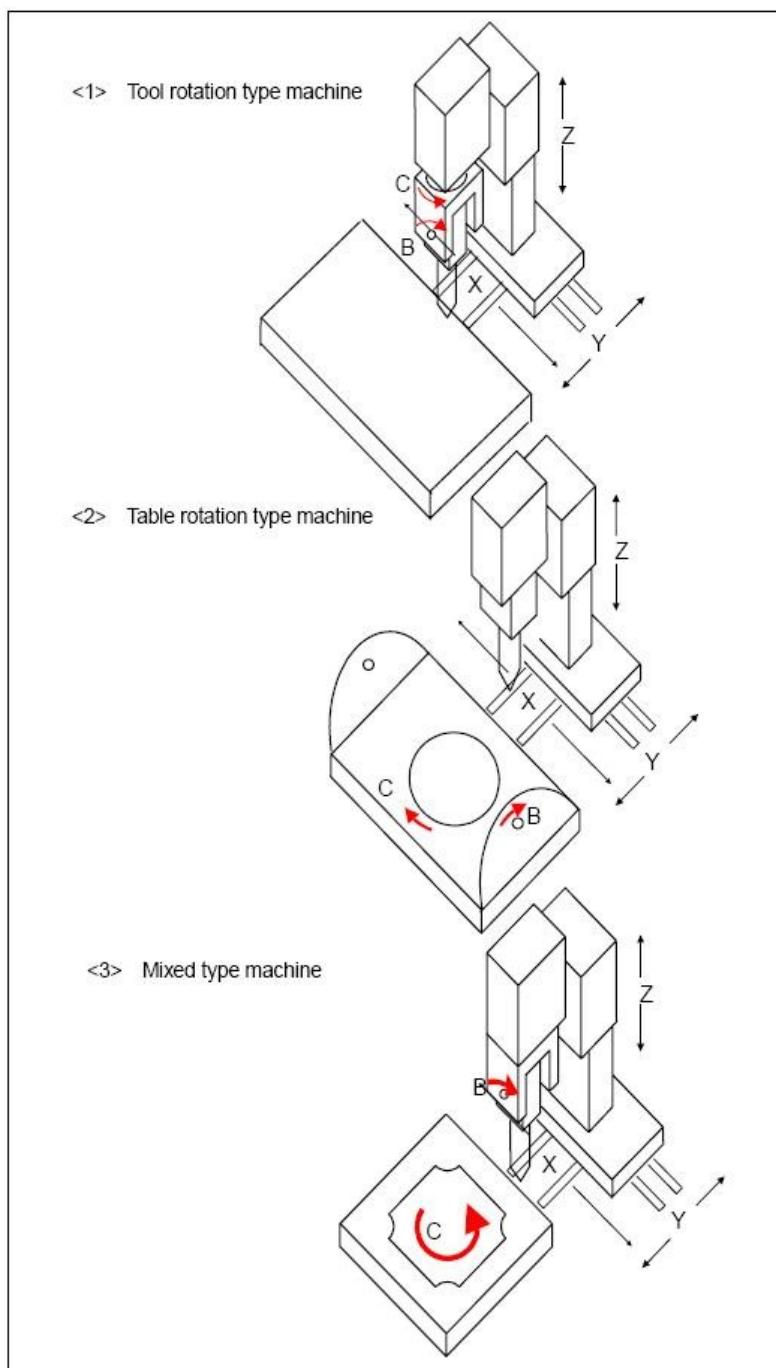


Figure 1.1: Three types of 5-axis machines

The CNC supports also variations of above machine types where the rotary axis are inclined.

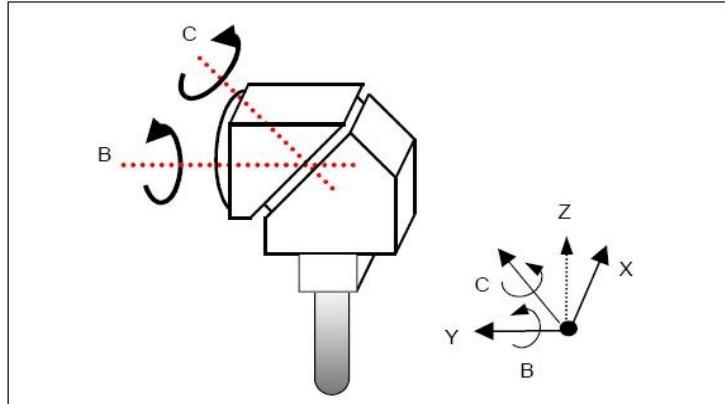


Figure 1.2: Tool rotation type machine with inclined rotary axis

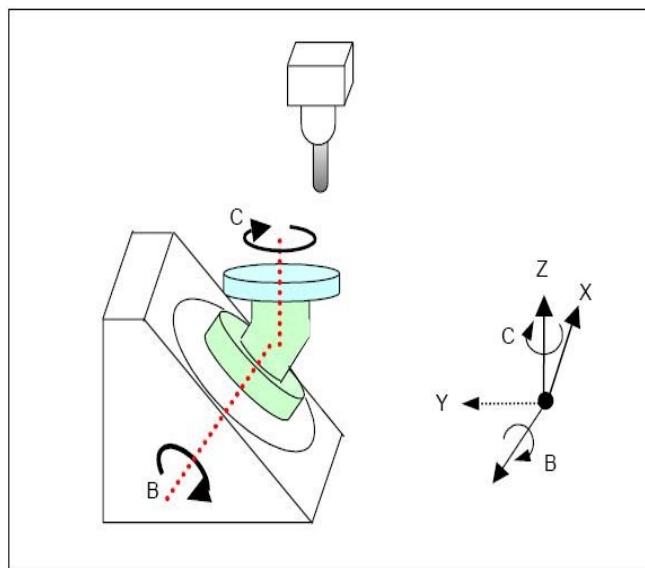


Figure 1.3: Table rotation type machine with inclined rotary axis

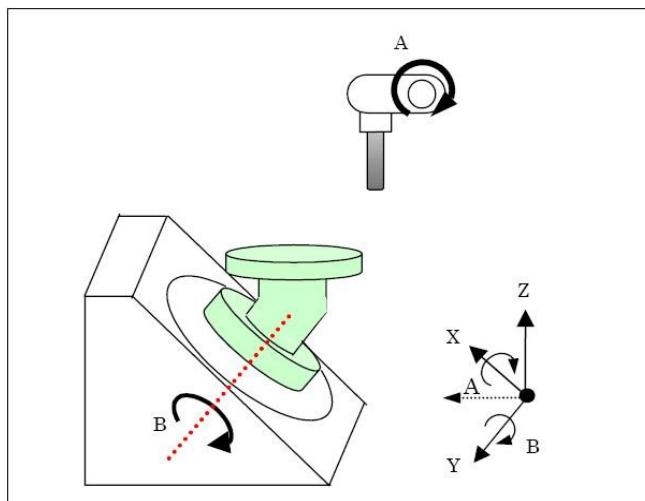


Figure 1.4: Mixed type machine with inclined rotary axis at the table

II SIMULTANEOUS 5-AXIS MACHINING

1 Overview

Two function are provided for simultaneous five axes programming:

- Tool Center Point Control
- Cutting point control
- Nano-Smoothing for 5-axis machining
- NURBS for 5-axis machining

2 Tool Center Point Control and Cutting Point Control

2.1 Overview

2.1.1 Tool Center Point Control

On a 5-axis machine, this function performs tool length compensation constantly, even in the middle of a block, and exerts control so that the tool center point moves along the specified path. (See Figure 2.1.)

This function is intended to perform machining on such 5-axis machines having rotary axes that turn a tool or table as well as three orthogonal axes (X-, Y-, and Z-axes) by accomplishing tool length compensation while changing the attitude of the tool. It enables the tool center point to move along the specified path even if the tool's direction changes with respect to the workpiece.

A coordinate system used for programming the tool center point control is called the programming coordinate system.

A coordinate system fixed on the table can be used as the programming coordinate system, which makes CAM programming easy.

A workpiece coordinate system fixed on a machine coordinate system can be employed as the programming coordinate system as well. On a machine of mixed type or table rotation type, cutter compensation for 5-axis machining is programmed in the workpiece coordinate system. So, to use cutter compensation for 5-axis machining and tool center point control at the same time, the workpiece coordinate system must be used as the programming coordinate system.

In any case, the cutting speed can be controlled easily because the tool center point moves at a specified speed with respect to the table (workpiece).

The commands that can be issued during tool center point control are positioning (G00), linear interpolation (G01), circular interpolation (G02, G03), and helical interpolation (G02, G03).

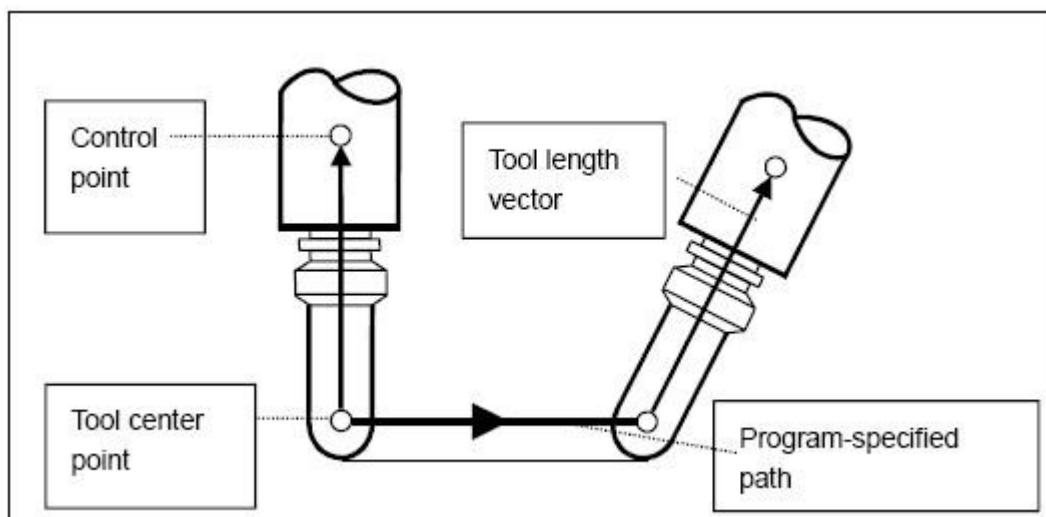


Figure 2.1: Path of tool center point

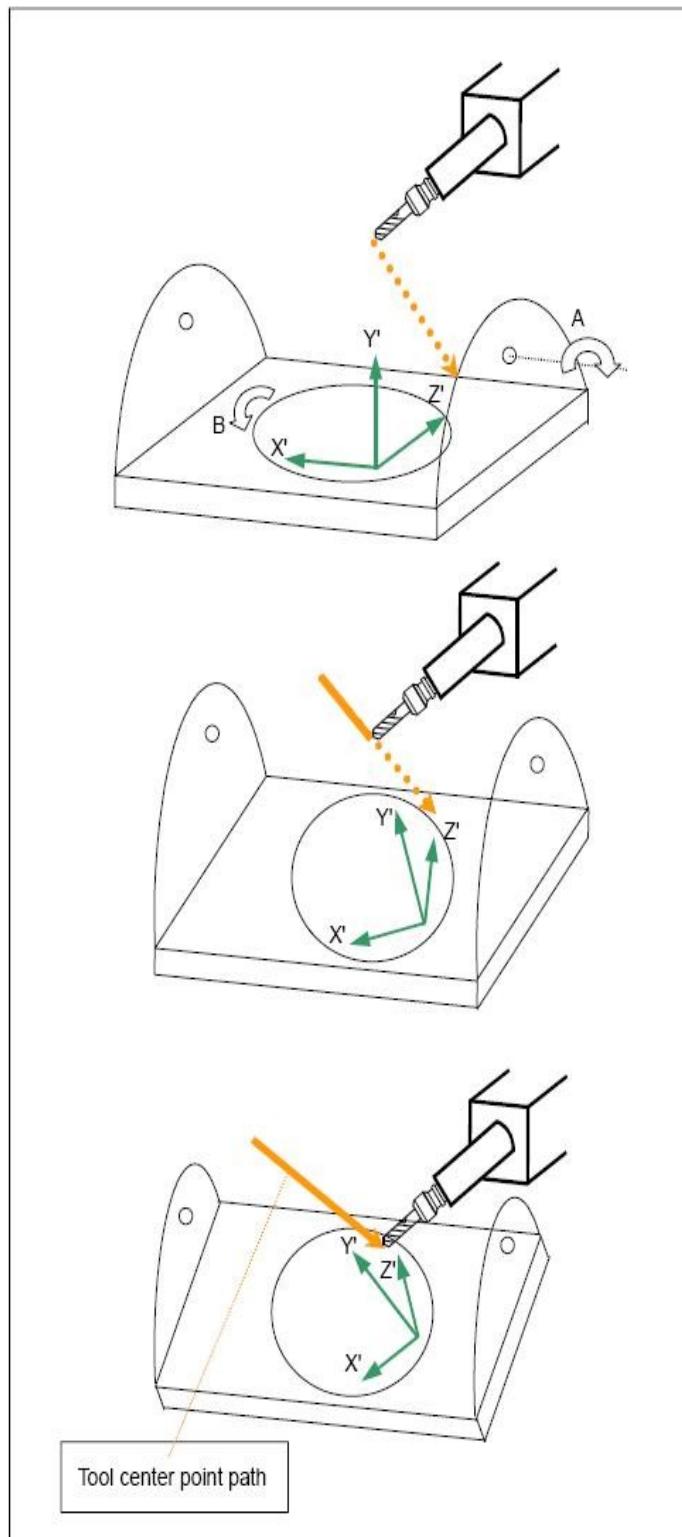


Figure 2.2: Path of TCP

When a coordinate system fixed on the table is used as the programming coordinate system, programming can be performed without worrying about the rotation of the table because the programming coordinate system does not move with respect to the table, although the position and direction of the workpiece fixed on the table change due to its rotation. When a straight line is specified, the tool center point moves along a straight path with respect to the workpiece as instructed. (See Figure 2.2.)

By setting the relevant parameter, the workpiece coordinate system can also be employed as the programming coordinate system. In this case, as the table turns, the position and direction of the workpiece fixed on the table change with respect to the programming coordinate system. It is therefore necessary to take into account the rotation of the table when specifying the end point. In this case, too, when a straight line is specified, the tool

center point moves along a straight path with respect to the workpiece as instructed. Figure 2.3 illustrates how linear interpolation is accomplished with a mixed type machine, showing the relationship between the case when a table-fixed coordinate system is used as the programming coordinate system and the case when the workpiece coordinate system is used. If linear interpolation is specified in this function mode, speed control is exerted in such a way that the tool center point moves at a specified speed with respect to the workpiece.

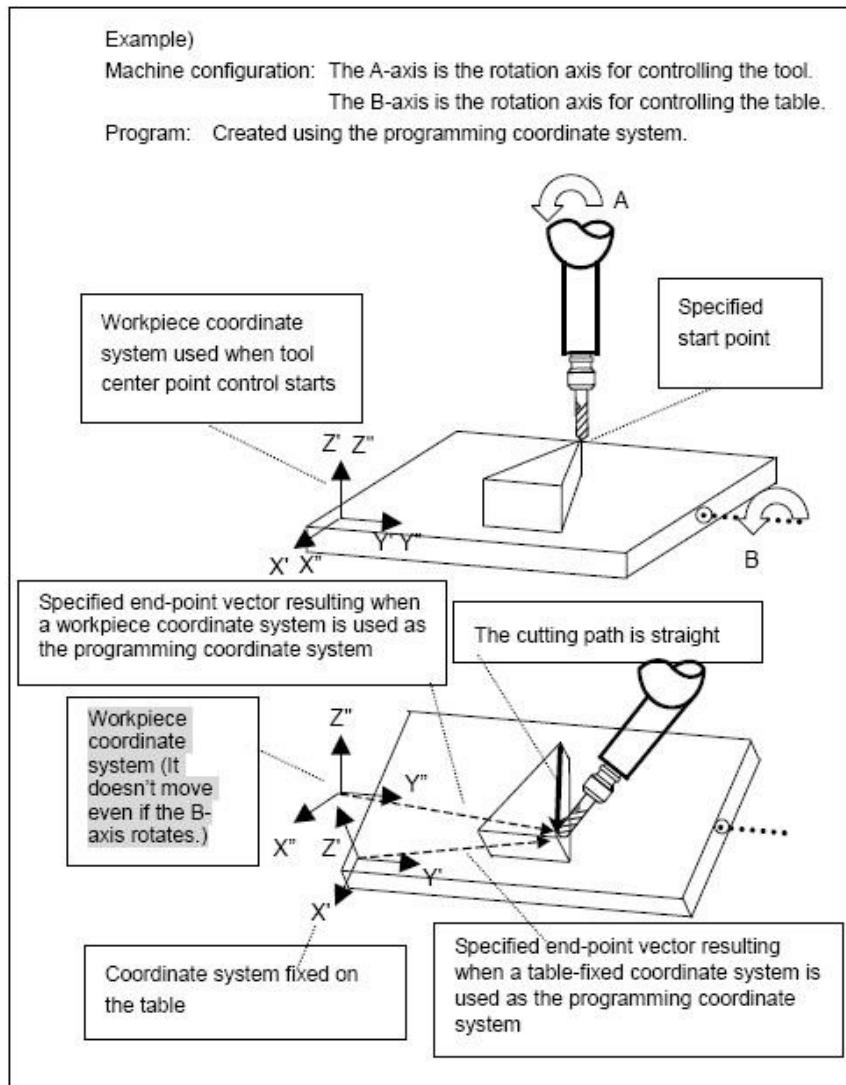


Figure 2.3: Linear interpolation with a mixed type machine

Even if the rotary axis that controls the tool does not intersect the one that controls the table, this function can still be used.

There are two types, as described below, one of which is used depending on how the direction of the tool axis is specified.

Type 1

The block end point of the rotary axes is specified (e.g. A, B, C). The CNC performs tool length compensation by the specified amount in the tool axis direction that is calculated from the specified position of the rotary axes and exerts control so that the tip of the tool moves along the specified path.

Type 2

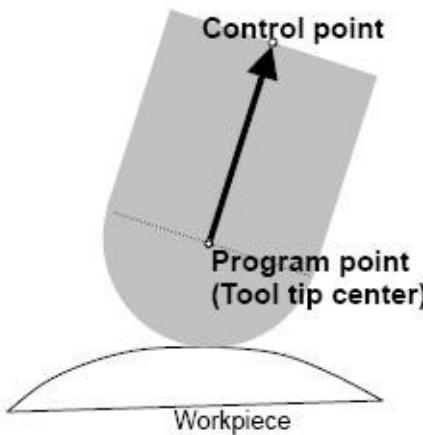
The direction of the tool axis (I, J, K) at the block end point, as seen from the coordinate system fixed on the table, is specified, instead of the position of the rotary axes. The CNC calculates an end point of the rotary axes where the tool will face the specified direction, performs tool length compensation by the

specified amount in the tool axis direction that is calculated from the position of the rotary axes, and exerts control so that the tip of the tool moves along the specified path.

2.1.2 Cutting Point Control

While Tool tip center is commanded with Tool center point control, Cutting point can be commanded with Cutting Point Command. With this feature, a tool with corner-R can be used.

Tool Center Point Control



Cutting Point Command

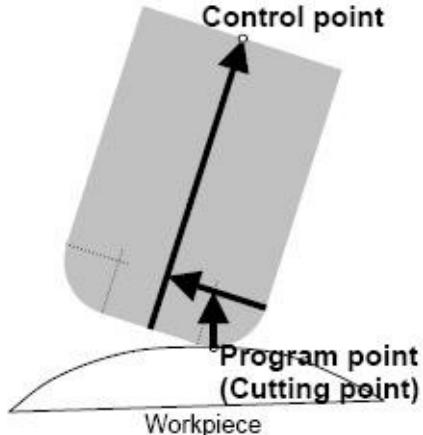


Figure 2.4: Tool Center Point Control and Cutting Point Control

There are two types of cutting point control commands, that are equivalent to the types of tool center point command.

NOTE

Cutting point control command can be used in machining center systems with enabled Tool offset memory C and Tool Center point Control.

2.2 Programming

2.2.1 Overview

G43.4 ... H_ ;	Starts tool center point control type 1
G43.5 ... H_ ;	Starts tool center point control type 2
G43.8 ... H_ ;	Starts cutting point control type 1
G43.9 ... H_ ;	Starts cutting point control type 2
G49 ;	Cancels tool center point or cutting point control

2.2.2 Tool center point control Type 1

Positioning and linear interpolation

G43.4 IP_ α_ β_ H_ P_ ;	Starts tool center point control
IP_ α_ β_ N5_ ;	Positioning / linear interpolation
IP_	In the case of an absolute programming, the coordinate value of the end point of the tool tip movement . In the case of an incremental programming, the amounts of the tool tip movement
α, β	In the case of an absolute programming, the coordinate value of the end point of the rotary axes. In the case of an incremental programming, the amount of the rotary axis movement.
H	Tool offset number
P	Selection for conventional control (P0) or tool posture control (P1). Here, the parameter TPC (No. 19604#0) can also select the way of controls, when P is not commanded.
N5_	In case of absolute programming, the coordinate values for axes that are not controlled by 5-axis transformation. In the case of an incremental programming, the amounts of the movement of non 5-axis machining control axis

CAUTION

1. Maximum number of non 5-axis machining control axes is 2. Specifying more than 2 causes alarm PS5421.
2. For 31i-A5 the total number of commanded axes in one block (IP + α + β + N5) must not exceed 5.

Movement to the position specified by the G43.4 block (startup) does not constitute tool center point control. Only tool length compensation is performed.

As for rotary axes, either table rotation axes or tool rotation axes are specified.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along a straight line with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

Circular Interpolation

G43.4 IP_ H_ P_ ;	Starts tool center point control
--------------------------	----------------------------------

G17 { G02 } IP - { I_J_K_ } R_ α _ β _ F_ ;
G18 { G02 } IP - { I_J_K_ } R_ α _ β _ F_ ;
G19 { G02 } IP - { I_J_K_ } R_ α _ β _ F_ ;

G17, G18, G19 X-Y , Z-X, Y-Z plane of table coordinate system

G02 , G03 Clockwise (CW) , Counterclockwise (CCW) circular interpolation

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
 In the case of an incremental programming, the amount of the tool tip movement
 (This pertains only to two axes on the plane.)

I_J_K_ Specify the distance between the start point in the rotary axis position of the block start point and the center of the arc, as seen from the programming coordinate system.

R Arc radius **R > 0**: The center angle of the arc is less than 180°. **R < 0** : The center angle of the arc is more than 180°.

α, β In the case of an absolute programming, the coordinate value of the end point of the rotary axes. In the case of an incremental programming, the amount of the rotary axis movement

F Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)

H Tool offset number

P Selection for conventional control (P0) or tool posture control (P1). Here, the parameter TPC (No. **19604#0**) can also select the way of controls, when P is not commanded.

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed. While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along an arc with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

CAUTION

Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

Helical interpolation

G43.4 IP_H_;

G17 { G02 } IP - { I_J_K_ R_ } α _ β _ γ _ F_ ;

Starts tool center point control

Circular interpolation on X-Y plane

G18 { G02 } IP - { I_J_K_ R_ } α _ β _ γ _ F_ ;

Circular interpolation on Z-X plane

G19 { G02 } IP - { I_J_K_ R_ } α _ β _ γ _ F_ ;

Circular interpolation on Y-Z plane

G17, G18, G19 X-Y , Z-X, Y-Z plane of table coordinate system

G02 , G03 Clockwise (CW) , Counterclockwise (CCW) circular interpolation

X_Y_Z_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement
(This pertains only to two axes on the plane.)

I_J_K_ Specify the distance between the start point in the rotary axis position of the block start point and the center of the arc, as seen from the programming coordinate system.

R Arc radius **R > 0**: The center angle of the arc is less than 180°. **R < 0** : The center angle of the arc is more than 180°.

α, β In the case of an absolute programming, the coordinate value of the end point of the rotary axes. In the case of an incremental programming, the amount of the rotary axis movement

γ In the case of an absolute programming, the coordinate values of the end point of the tool tip movement. In the case of an incremental programming, the amounts of the tool tip movement. (This pertains only to one of the linear axes subject to tool center point control for 5-axis machining which does not exist on the plane. Linear interpolation is performed simultaneously with circular interpolation.)

F Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)

H Tool offset number

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

Because the specified speed is usually the speed in the tangent direction of the arc, the speed of the linear axis,

$$F = \frac{\text{Length of the linear axis}}{\text{Length of the arc}}$$

when seen from the table coordinate system, is:

Depending on parameter HTG (No.1403#5), the specified speed varies as shown in the following table.

HTG (No. 1403#5)	
0	1
Tangential speed of arc	Synthetic speed of the linear axis speed and tangential speed

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves helically with respect to the table (workpiece). The end of the tool center point comes to the point

specified on the programming coordinate system.

CAUTION

Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

2.2.3 Tool Center Point Control Type 2

ATTENTION

Do not specify rotary axis commands in tool center point command type 2!

Positioning and linear interpolation

G43.5 IP_ H_ Q_ P_;

Starts tool center point control

IP_ N5_ I_ J_ K_ ;

Linear interpolation

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement.

In the case of an incremental programming, the amounts of the tool tip movement

I, J, K Tool axis direction at the block end point as seen from the programming coordinate system

H Tool offset number

Q Inclination angle of the tool (in degrees)

P Selection for conventional control (P0) or tool posture control (P1). Here, the parameter TPC (No. **19604#0**) can also select the way of controls, when P is not commanded.

N5_ In case of absolute programming, the coordinate values for axes that are not controlled by 5-axis transformation. In the case of an incremental programming, the amounts of the movement of non 5-axis machining control axis

CAUTION

3. Maximum number of non 5-axis machining control axes is 2. Specifying more than 2 causes alarm PS5421.
4. For 31i-A5 the total number of commanded axes in one block (IP + α + β + N5) must not exceed 5.

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).

With a tool rotation type machine, I, J, K can be specified using the G43.5 block. In the case of a table rotation type or mixed type machine, however, these cannot be specified. Specifying them with a table rotation type or mixed type machine causes alarm PS5421.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along a straight line with respect to the table (workpiece). The end of the tool center point comes to

the point specified on the programming coordinate system

CAUTION

If one or two of the I, J, and K values are omitted, the omitted value or values are considered to be 0.

In a block in which I, J, and K are all omitted, the compensation vector of the preceding block is used.

This block can be used only when the programming coordinate system is fixed on the table (when the WKP parameter (No.19696#5) is set to 0). Specifying G43.5 when the WKP parameter (No.19696#5) is set to 1 causes alarm PS5459.

Type 2 cannot be used when there is only one rotary axis or when any hypothetical axis is used. Specifying G43.5 in such a case causes alarm PS5459.5 When using the rotary axis rollover function or the rotary axis control function, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees.

Circular Interpolation

G43.5 IP_ H_ Q_ P_;

Starts tool center point control

**G17{ G02
G03} IP_ I_ J_ K_ R_ F_;**

**G18{ G02
G03} IP_ I_ J_ K_ R_ F_;**

**G19{ G02
G03} IP_ I_ J_ K_ R_ F_;**

G17, G18, G19 X-Y , Z-X, Y-Z plane of table coordinate system

G02 , G03 Clockwise (CW) , Counterclockwise (CCW) circular interpolation

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)

I_ J_ K_ Tool axis direction at the block end point as seen from the programming coordinate system.

R Arc radius **R > 0**: The center angle of the arc is less than 180°. **R < 0** : The center angle of the arc is more than 180°.

α, β In the case of an absolute programming, the coordinate value of the end point of the rotary axes. In the case of an incremental programming, the amount of the rotary axis movement

F Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)

H Tool offset number

Q Inclination angle of the tool (in degree)

P Selection for conventional control (P0) or tool posture control (P1). Here, the parameter TPC (No. 19604#0) can also select the way of controls, when P is not commanded.

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).

With a tool rotation type machine, I, J, K can be specified using the G43.5 block. In the case of a table rotation type or mixed type machine, however, these cannot be specified. Specifying them with a table rotation type or mixed type machine causes alarm PS5421.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along an arc with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

CAUTION

Only arc radius R can be specified. (The distance from the start point to the center of the arc cannot be specified using I, J, and K.)

A round circle (the start point and end point are the same) cannot be specified. Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

See the **CAUTION** box for tool center point control (type 2).

Helical interpolation

G43.5 IP_H_;

Starts tool center point control

**G17{ G02
G03 } IP_I_J_K_R_γ_F_;**

**G18{ G02
G03 } IP_I_J_K_R_γ_F_;**

**G19{ G02
G03 } IP_I_J_K_R_γ_F_;**

G17, G18, G19 X-Y, Z-X, Y-Z plane of table coordinate system

G02 , G03 Clockwise (CW) , Counterclockwise (CCW) circular interpolation

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)

I_J_K_ Tool axis direction at the block end point as seen from the programming coordinate system.

R Arc radius **R > 0**: The center angle of the arc is less than 180°. **R < 0** : The center angle of the arc is more than 180°.

α, β In the case of an absolute programming, the coordinate value of the end point of the rotary axes. In the case of an incremental programming, the amount of the rotary axis movement

γ In the case of an absolute programming, the coordinate values of the end point of the tool tip movement. In the case of an incremental programming, the amounts of the tool tip movement. (This pertains only to one of the linear axes subject to tool center point control for 5-axis machining which does not exist on the plane. Linear interpolation is performed simultaneously with circular interpolation.)

F Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)

H	Tool offset number
Q	Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

Because the specified speed is usually the speed in the tangent direction of the arc, the speed of the linear axis,

$$F = \frac{\text{Length of the linear axis}}{\text{Length of the arc}}$$

when seen from the table coordinate system, is:

Depending on parameter HTG (No. 1403#5), the specified speed varies as shown in the following table.

HTG (No. 1403#5)	
0	1
Tangential speed of arc	Synthetic speed of the linear axis speed and tangential speed

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).

With a tool rotation type machine, I, J, K can be specified using the G43.5 block. In the case of a table rotation type or mixed type machine, however, these cannot be specified. Specifying them with a table rotation type or mixed type machine causes alarm PS5421.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves helically with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

CAUTION

Only arc radius R can be specified. (The distance from the start point to the center of the arc cannot be specified using I, J, and K.)

A round circle cannot be specified.

Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

See the **CAUTION** box for tool center point control (type 2).

2.2.4 Cutting point control type 1

Positioning and linear interpolation

G43.8 IP_ α_ β_ H_ D_, L2 I_ J_ K_ P_; Starts cutting point control

IP_ α_ β_ ,L2 I_ J_ K_ ;

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement.

In the case of an incremental programming, the amounts of the tool tip movement

α, β In the case of an absolute programming, the coordinate value of the end point of the rotary axes.

In the case of an incremental programming, the amount of the rotary axis movement

H Tool offset number (Tool length)

D Tool offset number (tool radius and corner radius)

L2 I, J, K Direction perpendicular to cutting surface.

Don't specify anything else than I, J, K after L2.

P Selection for conventional control (P0) or tool posture control (P1). Here, the parameter TPC (No. 19604#0) can also select the way of controls, when P is not commanded.

The block end point of the rotation axes is specified (e.g. A, B, C).

The CNC performs tool length compensation by the specified amount in the tool axis direction that is calculated from the specified position of the rotation axes and controls so that the tip of the tool moves along the specified path.

2.2.5 Cutting point control type 2

Positioning and linear interpolation

G43.9 IP_ H_ D_, L2 I_ J_ K_ P_; Starts cutting point control

IP_ I_ J_ K_ ,L2 I_ J_ K_ ;

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement.

In the case of an incremental programming, the amounts of the tool tip movement

I, J, K Vector of the tool direction.

H Tool offset number (Tool length)

D Tool offset number (tool radius and corner radius)

L2 I, J, K Direction perpendicular to cutting surface.

Don't specify anything else than I, J, K after L2.

P Selection for conventional control (P0) or tool posture control (P1). Here, the parameter TPC (No. 19604#0) can also select the way of controls, when P is not commanded.

The direction of the tool axis (I, J, K) at the block end point, as seen from the coordinate system fixed on the

table, is specified,
instead of the position of the rotation axes.

The CNC calculates an end point of the rotation axes where the tool will face the specified direction, performs tool length compensation by the specified amount in the tool axis direction that is calculated from the position of the rotation axes, and controls so that the tip of the tool moves along the specified path.

2.2.6 Tool center point control cancellation

G49 IP_ α _ β ;

Cancels Tool center point control

IP_ In the case of an absolute programming, the coordinate value of the end point of the tool tip movement.

In the case of an incremental programming, the amounts of the tool tip movement

α , β In the case of an absolute programming, the coordinate value of the end point of the rotary axes.

In the case of an incremental programming, the amount of the rotary axis movement

The cancellation block for tool center point control is the one that controls buffering.

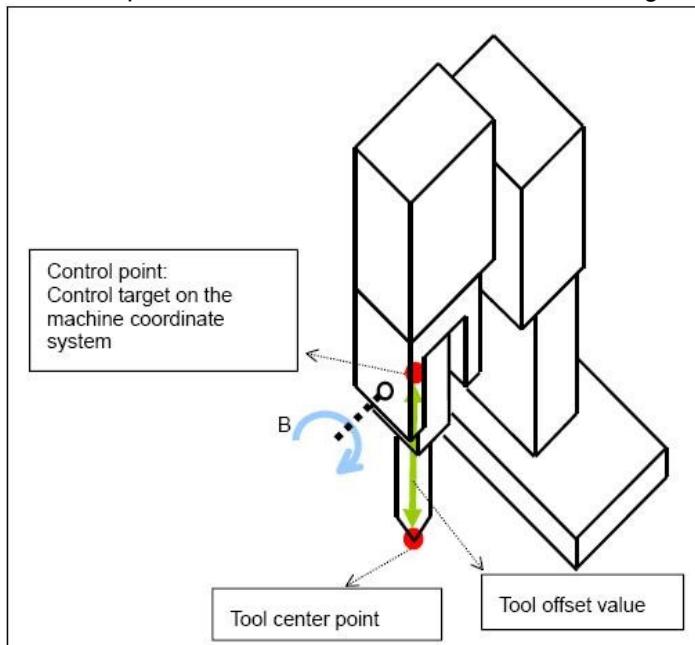


Figure 2.5: Control point and tool center point

CAUTION

The G49 command must be executed in G00 or G01 mode.

2.3 Explanations

2.3.1 Tool center point command

During tool center point control, the command specifies the location of each block end point as seen from the programming coordinate system.

The program specifies the tool center point.

As for the rotary axis, the command specifies the coordinate values of each block end point in the case of type 1 or the tool direction at each block end point in the case of type 2.

The feedrate is specified by the tangential speed relative to the workpiece (the tool's relative speed as opposed to the workpiece), which is represented by F.

NOTE

Tool center point control requires either the AI contour control I or AI contour control II option. In addition, be sure to specify the following parameters:

- Parameter LRP (No.1401#1)=1 : Linear rapid traverse
- Parameter FRP (No.19501#5)=1 : Acceleration/deceleration before interpolation is used for rapid traverse.
- Parameter (No.1671) : Acceleration before interpolation for rapid traverse.
- Parameter (No.1672) : Change time for bell-shaped acceleration before interpolation for rapid traverse.
- Parameter (No.1660) : Maximum permissible acceleration for acceleration/deceleration before interpolation.

If they are not specified, alarm PS5420 is generated.

2.3.2 Modal Commands where tool center point control can be specified

Tool center point control can be specified in the modal G code states listed below.

In a modal state other than the following modal G codes, specifying tool center point control results in alarm PS5421:

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Canned cycle cancel (G80)
- Constant surface speed control cancel (G97)
- Canned cycle: return to initial level (G98)
- Canned cycle: return to R point level (G99)

Milling

- Coordinate system rotation start or 3-dimensional coordinate conversion mode on (G69)
- Feed per minute (G94)
- Polar coordinate interpolation mode cancel (G113)

Turning

- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))

2.3.1 Commands that can be specified during tool center point control

The commands that can be specified during tool center point control are linear interpolation (G01), positioning (G00), circular interpolation (G02, G03), and helical interpolation (G02, G03).

When linear interpolation (G01) is specified during tool center point control, speed control is exerted so that the tool center point moves at the specified speed.

The circular interpolation command (G02, G03) controls the tangential speed of the arc path along which the tool center point moves.

The helical interpolation command (G02, G03) controls the tangential speed of the arc path along which the tool center point moves or a synthetic speed including that of the helical axis. (This is dependent on the setting of parameter HTG (No.1403#5).) As the actual speed, the speed at the control point is shown.

The G codes that can be specified in the tool center point control mode are listed below.

Specifying a G code other than these codes results in alarm PS5421.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- Exact stop (G09)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Stored stroke check function (G22/G23)
- Cutter or tool nose radius compensation: preserve vector (G38)
- Cutter or tool nose radius compensation: corner circular interpolation (G39)
- Cutter compensation: cancel (G40)
- Cutter or tool nose radius compensation / Three-dimensional cutter compensation (G41/G42)
- Cutter compensation for 5-axis machining (G41.2/G42.2/G41.4/G42.4/G41.5/G42.5)
- Tool length compensation cancel (G49)
- Scaling (G50/G51)
- Programmable mirror image (G50.1/G51.1)
- Exact stop mode (G61)
- Automatic corner override mode (G62)
- Tapping mode (G63)
- Cutting mode(G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Figure copy (G72.1/G72.2)
- Absolute programming (G90)
- Incremental programming (G91)

- Inverse time feed (G93)

Milling

- Tool offset increase (G45)
- Tool offset decrease (G46)
- Tool offset double increase (G47)
- Tool offset double decrease (G48)

2.3.2 Inverse Time Feed (G93)

For linear interpolation:

$$FRN = \frac{1}{(time[min])} = \frac{\text{feedrate}}{\text{distance}}$$

When inverse time feed is specified under Tool center point control, "distance" becomes the amount of the movement of the tool center point. However, when the rotation axis moves without the tool center point moving to a work, "distance" becomes the amount of the movement of the rotation axis.

2.3.3 Rotary axis commands

If a command is specified during tool center point control that prohibits the tool center point from moving with respect to the workpiece, the maximum cutting speed (parameter No.1430) is assumed as the feedrate of the rotary axis when parameter RFC (No.19696#6) is 0, and the speed specified by F is assumed when parameter RFC (No.19696#6) is 1. The rotary axis command cannot be specified during tool center point control of type 2. Specifying the command with type 2 causes alarm PS5421.

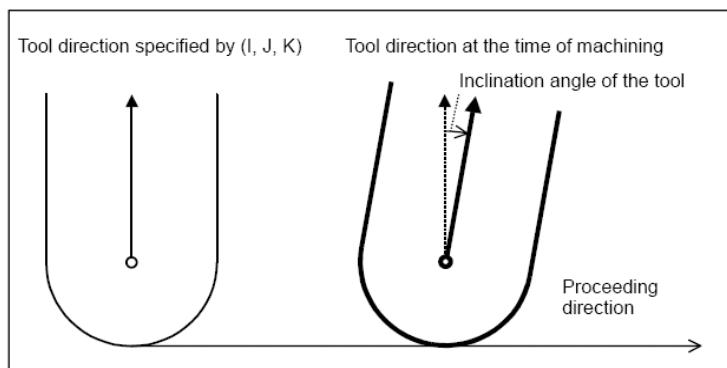
(See also 2.3.7)

2.3.4 Inclination angle of the tool

In the case of tool center point control of type 2, the inclination angle of the tool can be specified using address Q of G43.5. The inclination angle of the tool represents how inclined the tool direction is toward the proceeding direction with respect to the direction specified by (I, J, K) at the time of machining on the plane produced by the tool direction specified by (I, J, K) and the proceeding direction on the programming coordinate system. (See the figure below.)

Overviewly, the normal direction on the machining plane is specified by (I, J, K). If the tool direction needs to be inclined toward the proceeding direction with respect to the normal direction at the time of machining, perform compensation using Q.

If the direction specified by (I, J, K) matches the direction in which the tool moves at the time of machining, commanding Q is unnecessary.



Example

G43.5 I_ J_ K_ H_ Q2.0	Inclines the tool by two degrees toward the proceeding direction at the time of machining
------------------------	---

2.3.5 Table coordinate system as programming coordinate system

The programming coordinate system is used for tool center point control and 3D Cutter Compensation. When the G43.4 or G43.5 command is specified with parameter WKP (No. 19696#5) set to 0, the workpiece coordinate system that is fixed on the table at that point of time becomes the programming coordinate system. Thereafter, the programming coordinate system rotates as the table turns around.

It does not rotate with the tool head.

X, Y, and Z mentioned hereinafter are assumed to be commanded on the programming coordinate system.

In case that INZ (Parameter No. 19754#5) = 0, if the G43.4 or G43.5 command is specified or when the rotary axis of the table has moved in a block preceding G43.4 or G43.5, the angle of the table's rotary axis represents the initial state of the programming coordinate system.

NOTE:

In case that the program is made assuming that the table coordinate system is fixed to the table when the rotary table position is 0, if tool center point control or 3-dimensional cutter compensation is commanded several times, the table rotary axis position must be 0 every time when these function is started. (INZ = 0)

In the case of type 2, the tool direction seen from the coordinate system that is fixed on the table is specified by I, J, K.. In the descriptions that follow, the table-fixed coordinate system is represented by X', Y', and Z'.

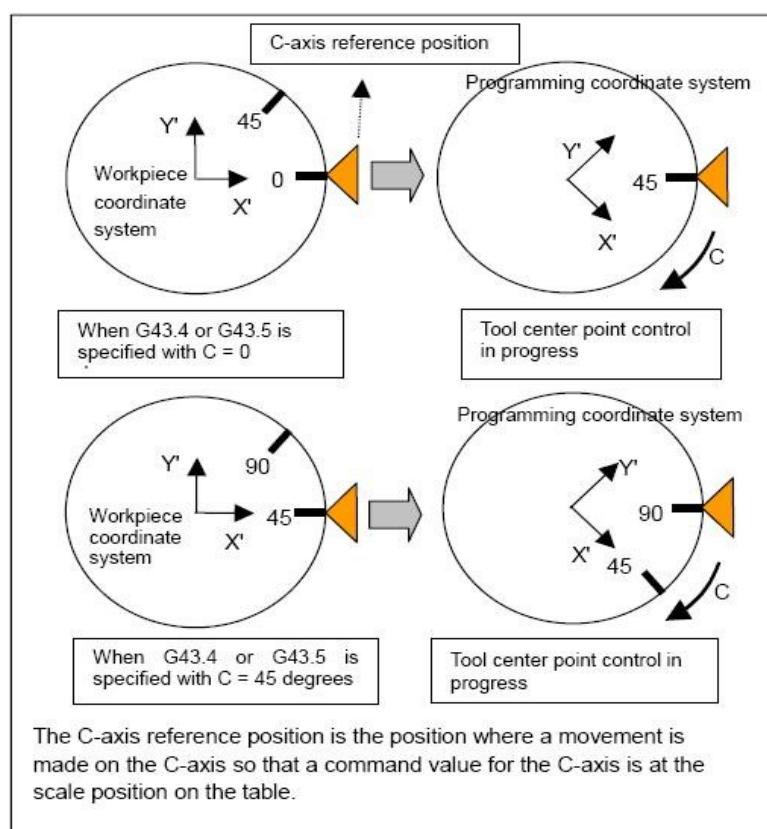


Figure 2.6: Programming coordinate system fixed on the table

In case that INZ (Parameter No. 19754#5) = 1, the table coordinate system is fixed to the table in the state that the table rotary axis position is 0 regardless of the rotary axis position at the start block of tool center point control

and 3D Cutter Compensation.

Assume that C-axis is the rotary axis around Z-axis.

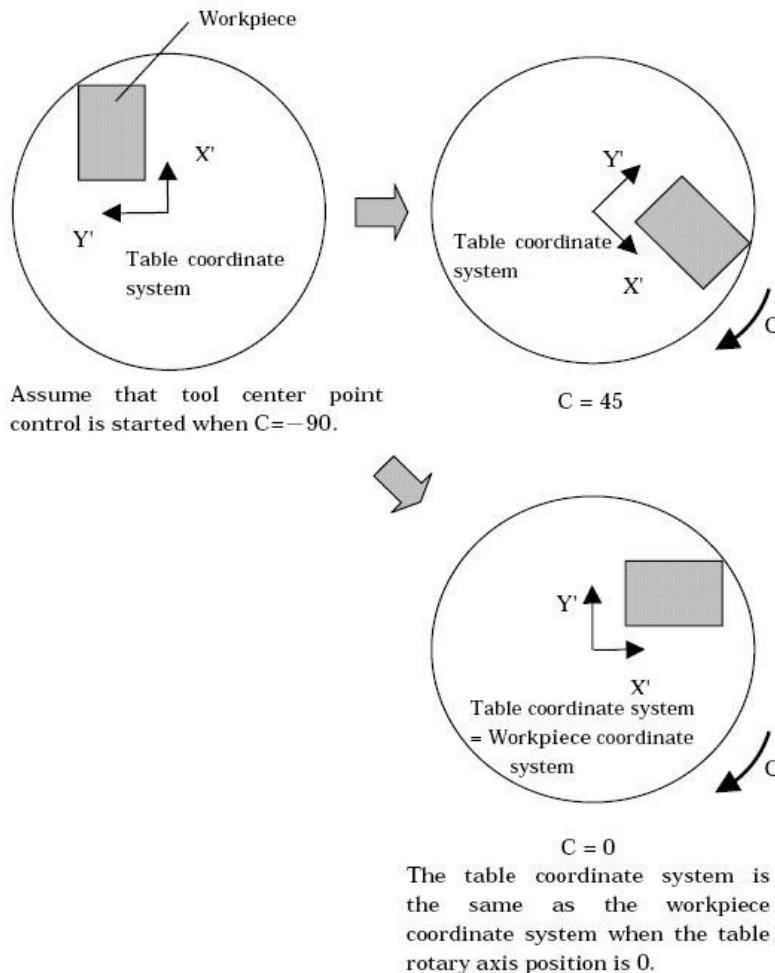


Illustration 1: Example for INZ = 1

If tool center point control or 3-dimensional cutter compensation is started when the table rotary axis position is not 0, the table coordinate system is not the same as the workpiece coordinate system at the start block.

If the axis motion command is specified at the start block, the axis moves in the workpiece coordinate system.

The axis moves in the table coordinate system from next block.

2.3.6 Workpiece coordinate system as programming coordinate system

When the G43.4 command is specified with parameter WKP (No.19696#5) set to 1, the workpiece coordinate system that is in use at that point of time becomes the programming coordinate system.

In this case, the programming coordinate system does not rotate as the table turns around but remains fixed on the workpiece coordinate system.

Hereinafter, when X, Y, and Z are commanded, the tool moves along a straight line toward the table (workpiece). For X, Y, Z, specify the end point location after the rotation of the table, as seen from the programming coordinate system.

Tool center point control type 2 cannot be used. Specifying G43.5 when parameter WKP (No.19696#5) set to 1 causes alarm PS5459.

In the descriptions that follow, the coordinate values of the workpiece coordinate system used as the programming coordinate system are represented by X'', Y'', and Z''.

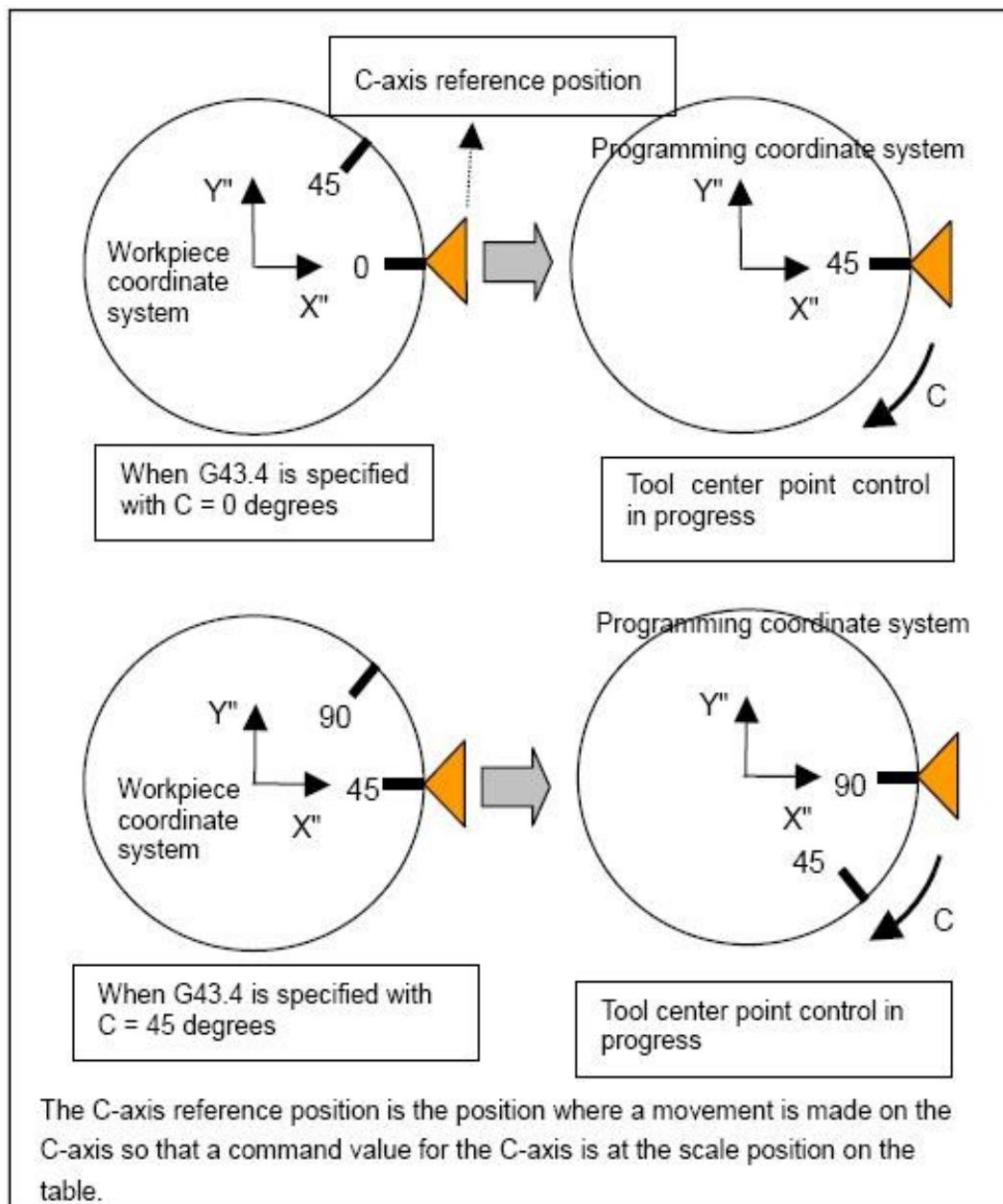


Figure 2.7: Programming coordinate system identical to workpiece coordinate system

2.3.7 Notes for circular and helical interpolation on workpiece coordinate system

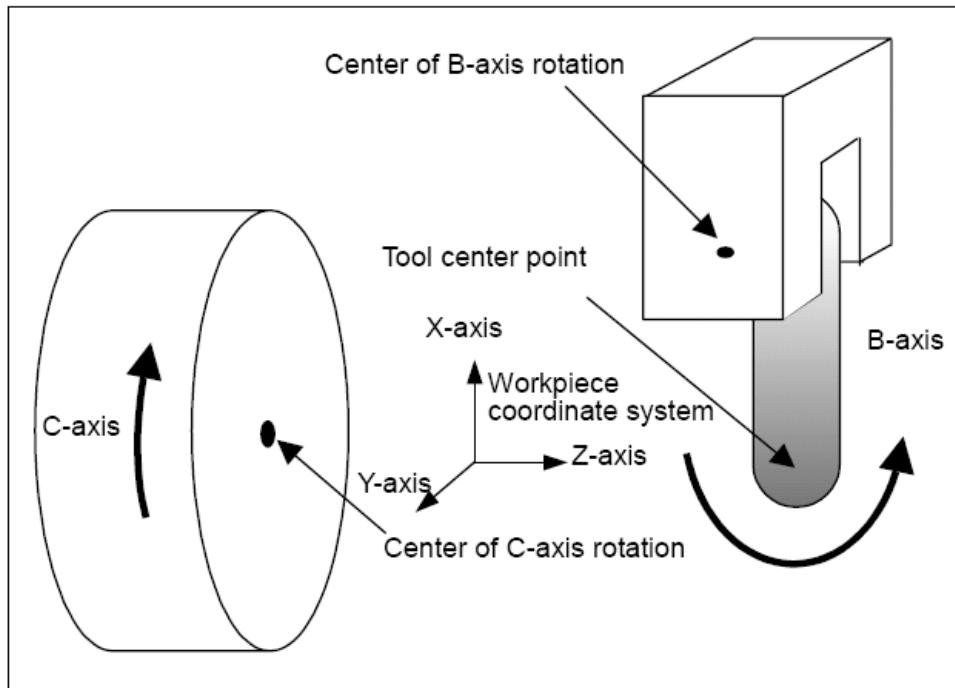
- The start point, end point, and center of an arc change as the table rotation axis rotates.
- I, J, K commands the vector of the block start point to the center of the arc from the start point in the rotary axis position.
- Note the following:
 1. Only a table rotation axis normal to a selected plane can be rotated during circular interpolation.
 2. During circular interpolation, those table rotation axes not normal to a selected plane must continue to be at the same position as when tool center point control is started.

If 1. or 2. is not satisfied, the alarm PS5421 is issued.

No restriction is imposed on rotation on a tool rotation axis.

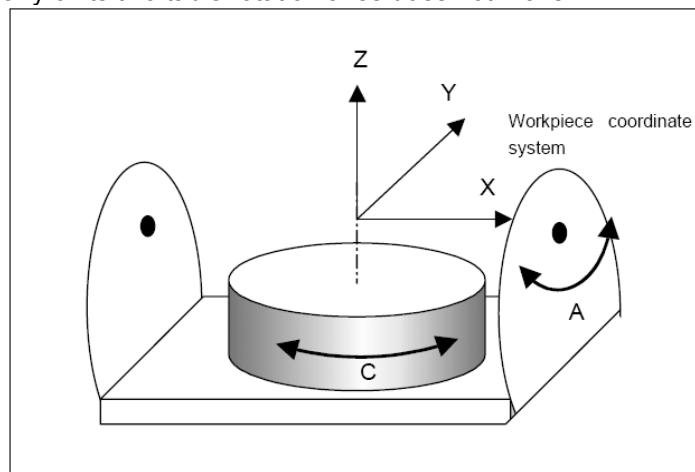
2.3.8 Examples

Description is based on mixed type machine configuration as shown in below picture.



<p>...</p> <p>(G01 C90. ;)</p> <p>G43.4 H1 ;</p> <p>G17 G02 IP IR B10. C20. ;</p> <p>...</p>	<p>After the G43.4 command, the X-Y plane is selected using the G17 command and circular interpolation is performed by rotating the C-axis (table rotation axis) (including those cases where the C-axis moves before the G43.4 command).</p> <p>→ This case corresponds to 2.3.7 and allows circular interpolation.</p> <p><i>IP:</i> Coordinates of the end point <i>IR:</i> Arc radius</p>
<p>...</p> <p>G43.4 H1 ;</p> <p>G18 G02 IP IR ;</p> <p>...</p>	<p>After the G43.4 command, the Z-X plane is selected using the G18 command and circular interpolation is performed without rotating the C-axis (including those cases where the C-axis moves before the G43.4 command).</p> <p>→ This case corresponds to 2.3.7 and allows circular interpolation.</p> <p>The same is also true when the G19 command is used.</p>
<p>...</p> <p>G43.4 H1 ;</p> <p>G18 G02 IP IR C20. ;</p> <p>...</p>	<p>After the G43.4 command, the Z-X plane is selected using the G18 command and the C-axis is rotated during circular interpolation.</p> <p>→ Alarm (violation of 2.3.7)</p> <p>The same is also true when the G19 command is used.</p>
<p>...</p> <p>G43.4 H1 ;</p> <p>G01 C10. ;</p> <p>G18 G02 IP IR ;</p> <p>...</p>	<p>After the G43.4 command, the Z-X plane is selected using the G18 command and circular interpolation is performed after rotating the C-axis.</p> <p>→ Alarm (violation of 2.3.7)</p> <p>The same is also true when the G19 command is used.</p>

Description is based on table rotation type machine. A table rotation type machine can be considered equivalent to a mixed type machine if any of its two table rotation axes does not move.



<pre>... G01 A90. (C10.) ; G43.4 H1 ; G17 G02 IP IR ; ...</pre>	<p>The master axis (A-axis) moves before the G43.4 command and, after the G43.4 command, circular interpolation is performed without rotating the A- or C-axis (including those cases where the C-axis moves before the G43.4 command).</p> <p>→ This case corresponds to 2.3.7 and allows circular interpolation.</p>
<pre>... G01 A90. ; G43.4 H1 ; G01 C10. ; G17 G02 IP IR ; ...</pre> <pre>... G01 A90. ; G43.4 H1 ; G17 G02 IP IR C10. ; ...</pre>	<p>The master axis (A-axis) moves before the G43.4 command and, after the G43.4 command, circular interpolation is performed using the G17 (X-Y plane) command by rotating the C-axis, or the C-axis is rotated during circular interpolation.</p> <p>→ Alarm (violation of 2.3.7)</p>
<pre>... G43.4 H1 ; G01 A10. ; G17 G02 IP IR C10. ; ...</pre>	<p>After the G43.4 command, the A-axis is moved and circular interpolation is performed using the G17 command (X-Y plane).</p> <p>→ Alarm (violation of 2.3.7)</p>
<pre>... G01 A90. ; G43.4 H1 ; G19 G02 IP IR A10. ; ...</pre>	<p>The G43.4 command is executed after moving the A-axis and circular interpolation is performed while rotating the A-axis using the G19 (Y-Z plane) command.</p> <p>→ This case corresponds to 2.3.7 and allows circular interpolation.</p>
<pre>... G43.4 H1 ; G01 C10. ; G19 G02 IP IR ; ...</pre>	<p>After the G43.4 command, the C-axis is rotated and circular interpolation is performed using the G19 (Y-Z plane) command.</p> <p>→ Alarm (violation of 2.3.7)</p>
<pre>... G01 A90. C10. ; G43.4 H1 ; G18 G02 IP IR; ...</pre>	<p>The G43.4 command is executed after moving the A- and C-axes, and circular interpolation is performed using the G18 (Z-X plane) command without moving any rotary axis.</p> <p>→ This case corresponds to 2.3.7 and allows circular interpolation.</p>
<pre>... G43.4 H1 ; G01 A10. (C10.) G18 G02 IP IR; ...</pre>	<p>After the G43.4 command, circular interpolation is performed using the G18 command (Z-X plane) by moving any of the rotary axes.</p> <p>→ Alarm (violation of 2.3.7)</p>

2.3.9 Moving distance of rotary axes

If the moving distance of the rotary axis is long compared to that of the linear axis, the rotary axis moves faster so that the tool center point moves at the specified speed, possibly resulting in the tool center point traveling on an inadequate path.

In such a case, it is possible to exert control to slow down the speed and ensure that the tool center point travels on the specified path, by setting parameter CRS (No.**19746#6**) to 1. When using this control, specify in parameter (No.**19751**) (for rapid traverse) and parameter (No.**19752**) (for cutting feed) the extent of deviation of the path at which the speed is to be slowed down (maximum allowable deviation of the path). When 0 is specified, the least input increment is regarded as the maximum allowable deviation of the path.

2.3.10 Tool Behavior at startup and cancellation

When tool center point control is started (G43.4/G43.5) or canceled (G49), the tool moves by a tool-offset value. Compensation vector calculation is performed only at the end of a block.

2.3.11 Current display during tool center point control

During tool center point control, the position of the control point (rotation center of the tool rotation axis) is displayed as the machine coordinate.

When parameter WKP (No.**19696#5**) is 0, whether to use absolute or relative coordinates can be selected using parameter DET (No.**19608#2**).

If parameter DET (No.**19608#2**) is 0, the position of the tool center point on the programming coordinate system is displayed.

If parameter DET (No.**19608#2**) is 1, the position of the tool center point in the workpiece coordinate system is displayed.

2.3.12 Tool offset

If tool offsets are used based on tool numbers, tool center point control is carried out using the tool length compensation value corresponding to the relevant tool number (T code).

If tool life management is used, tool center point control is carried out using the tool length compensation value corresponding to the tool in use.

2.3.13 Tool Posture Control for G00 / G01

Tool Posture Control of Tool Center Point control for 5-axis machining controls the two rotary axes so that the tool posture satisfies the following conditions in real time. (Figure 2.8)

- The actual vector of the tool posture (tool direction) lies on the plane, which is made by the tool length compensation vectors at the start point and the end point of a block in real time.
- The angle between the actual vector of the tool posture and the tool length compensation vector at the start point is proportional to the length between the actual tool center point and the start tool center point of a block.

Commanding P0 at the start block of Tool Center Point control (G43.4/G43.5) makes the conventional control active. And, commanding P1 at the start block of Tool Center Point control (G43.4/G43.5) makes the tool posture control active. Additionally, when the parameter TPC (No.**19604#0**) is set to 1, the posture control is active at the start block of Tool Center Point control (G43.4/G43.5) which has no P address. The control way (Conventional control or Tool posture control) selected at the start block Tool Center Point control (G43.4/G43.5) is active until G49 is commanded.

The tool posture control is available for Rapid traverse (G00: linear type) and Liner interpolation (G01). Other notes, cautions and restrictions are the same as Tool Center Point control for 5-axis machining.

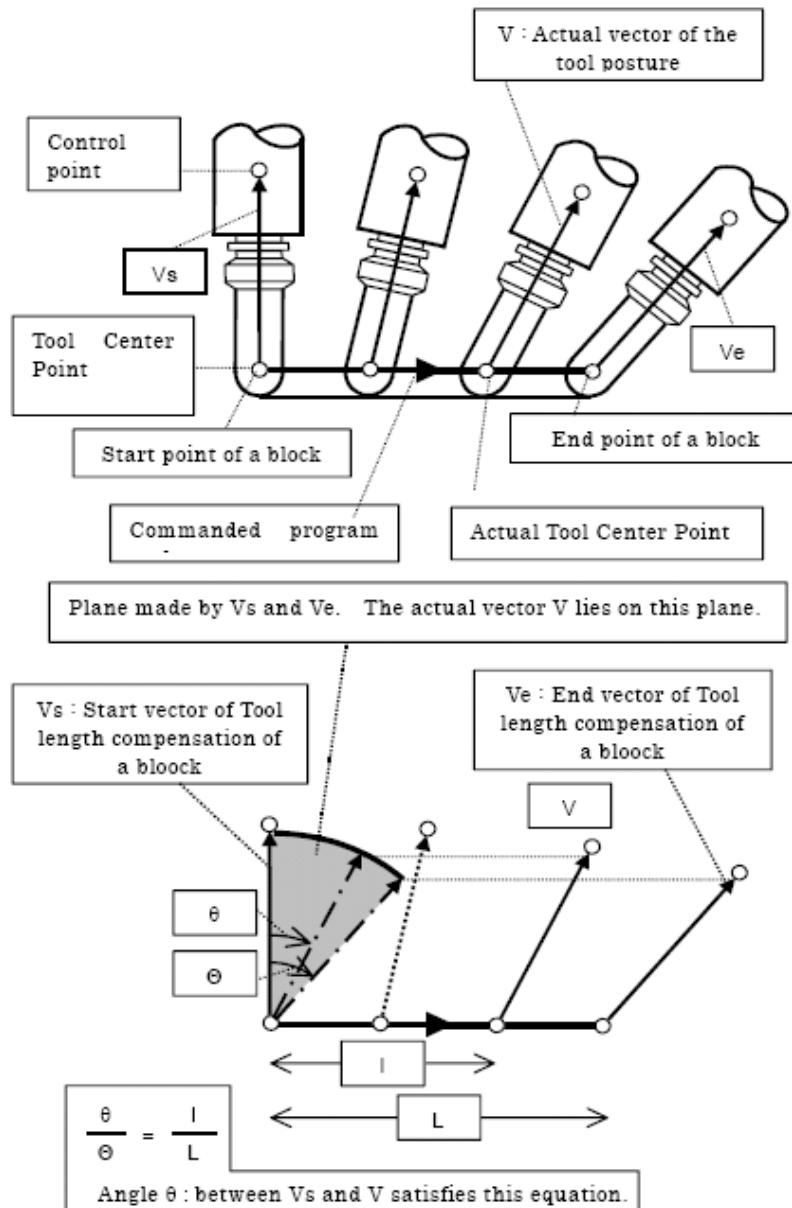


Figure 2.8: Detailed tool posture control

2.3.14 Tool Posture Control for Circular Interpolation (G02 / G03)

Here, three directions are defined as follows.

- Tool direction: Direction from tool center point to control point (tool posture)
- Center direction: Direction from tool center to circular center
- Travel direction: Direction of tool traveling on interpolation plane, normal to center direction

And angles between tool direction and other directions are defined as follows.

α_s : Angle between tool direction and center direction at block start position

β_s : Angle between tool direction and travel direction at block start position

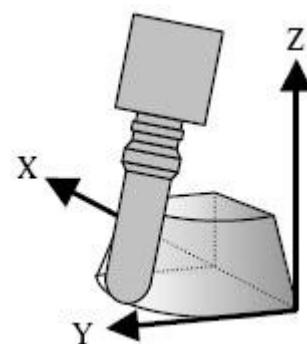


Figure 2.9: Example

α_e : Angle between tool direction and center direction at block end position

β_e : Angle between tool direction and travel direction at block end position

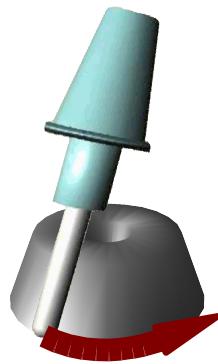
α_t : Momentary angle between tool direction and center direction

β_t : Momentary angle between tool direction and travel direction

Momentary tool direction is controlled as follows by commanding circular/helical interpolation command (G02/G03) in tool posture control (Figure 2.10).

$$\alpha_t = \alpha_s + (\alpha_e - \alpha_s) \cdot t$$

$$\beta_t = \beta_s + (\beta_e - \beta_s) \cdot t$$



Here, t shows movement ratio of tool center point (ratio of momentary amount of movement to all amount of block). $t = 0$ at block start position and $t = 1$ at block end.

With a command corresponding to $\alpha_s = \alpha_e$, $\beta_s = \beta_e = 90\text{deg}$, tool direction tool posture moves on a cone side face. Even a command not corresponding to such angles, tool direction is also controlled according to the above algorithm.

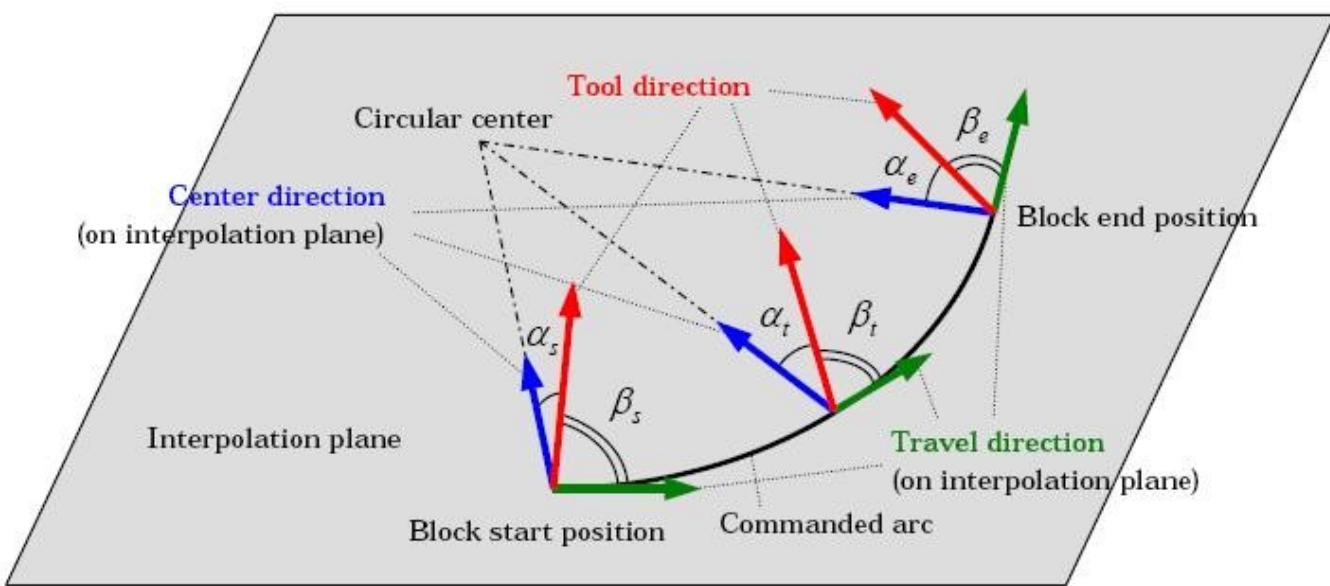


Figure 2.10: Tool posture control fro G02 / G03

NOTE

1. It is impossible to command so that tool direction reverses to interpolation plane defined by G17/G18/G19 command. With such a command, an alarm P/S0432 (UNAVAILABLE POSTURE IN TPC) is generated.
2. A rotation axis might rotate greatly when tool posture becomes near the singular posture. There is a function to avoid such a movement (related parameter No.19696#3, No.19738, No.19739) for positioning and linear interpolation. But similar function for circular/helical interpolation is not provided.
3. This function cannot be used with three dimensional circular interpolation.

2.3.15 ‘Tool side rotary axis’ and ‘Work-piece side rotary axis’

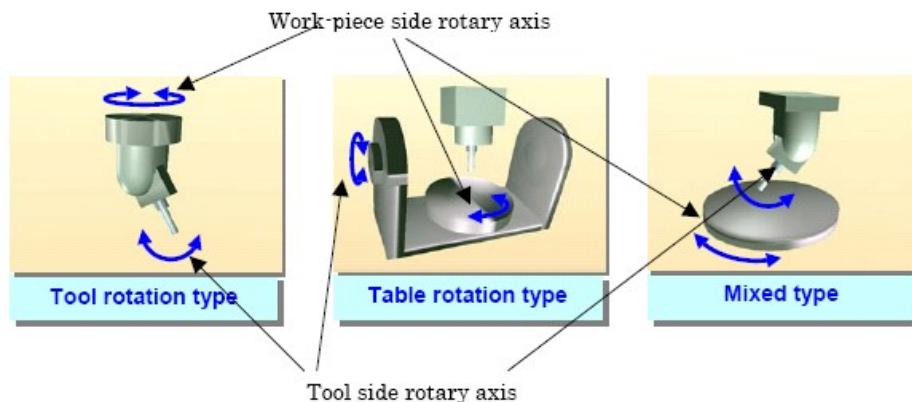
In this paragraph, ‘Tool side rotary axis’ and ‘Work-piece side rotary axis’ which are used in the next paragraph are explained.

It is required to take notice of ‘Singularity’ in the case of Tool Center Point Control Type II.

On a 5-axis machine, one rotary axis tilts the tool against the work-piece. This rotary axis is called 'Tool-side rotary axis', and the other is called 'Work-piece side rotary axis'.

According to the mechanical unit type, they are as follows:

Mechanical unit type (No. 19680)	Tool side rotary axis	Work-piece side rotary axis
Tool rotation type (2)	Slave axis	Master axis
Table rotation type (12)	Master axis	Slave axis
Mixed type (21)	Tool rotation axis	Table rotation type



2.3.16 Singularity, Singular position, Singular posture

Example:

On a tool rotation type machine which has C axis (master axis around Z axis) and B axis (slave axis around Y axis) and the reference tool direction in Z axis direction, the singular position is $B=0, +/-180, \dots, \text{deg}$. Then, the tool posture is the singular posture at $C=\text{any-deg.}$.

When $B=0$ (singular position), the tool posture is a singular posture at $C=\text{anydeg.}$, as shown in Fig.4 for example.

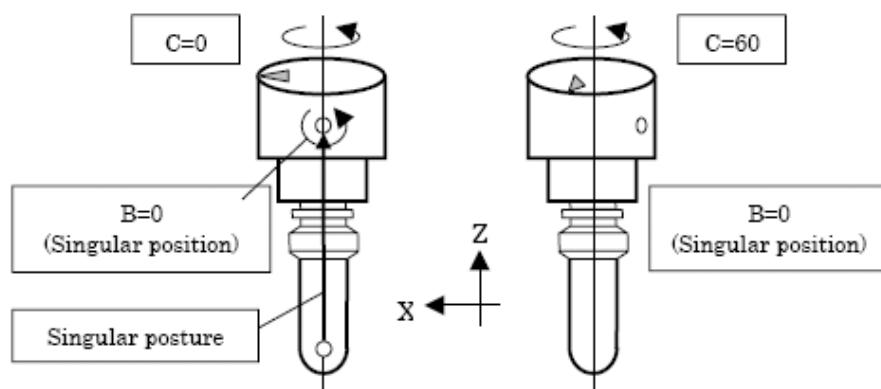


Figure 2.11: Singular position and singular posture

In the case that the reference tool axis direction is tilted (Parameter No. 19698, No. 19699) or in the case that the rotary axis is inclined (Parameter No. 19682, No. 19683, No. 19687, No. 19688), there are machines which have no singular position and singular posture.

2.3.17 Tool posture in the near of singular posture

When Tool center point control is used on a machine with singular position, in the case that the tool posture is near the singular posture while a block is executed, the movement of the Work-piece side rotary axis may

become large and the movement of the machine may become unstable.

In such a case, if the tool posture at the block end is changed so that the tool posture passes the singular posture, the movement of the machine will become stable, the machining time will be reduced, and the smoothness of the machining surface will be improved.

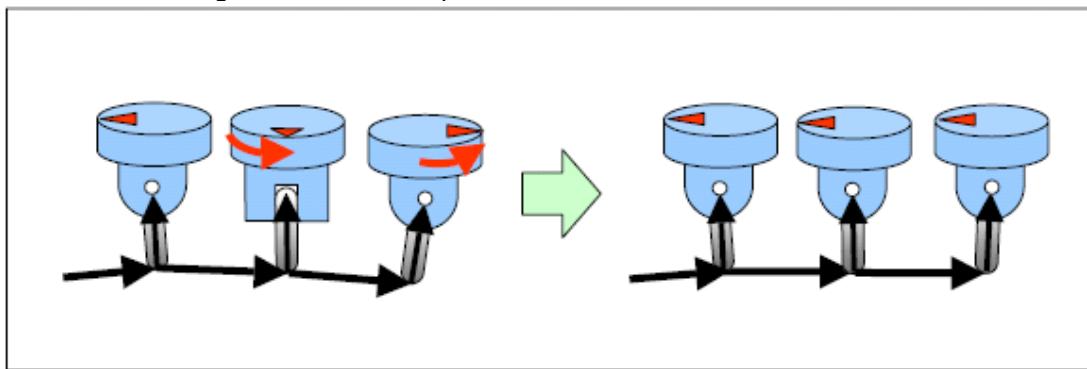


Figure 2.12: Change of tool posture at the end of block

C-axis rotates very rapidly and largely near Singular posture.

So, the rapid and large C-axis rotation should be avoided by traveling just through Singular posture. To travel the singular posture, the end point is modified within the tolerance.

In the case that the tool posture is commanded with I,J,K (G43.5 : Tool center point control type II), Tool posture control has the following function.

- It checks if the tool posture can be near Singular posture in each block.
- When the tool posture can be near Singular posture in the block, it changes

the tool posture (the rotary axis position) at the end of the block so that the tool posture passes the singular posture.

When the difference between a tool posture during a block and the singular posture is less than the parameter No.19738, the tool posture is regarded as 'near singular posture'. If the parameter No.19738 is 0, the tool posture at the end of a block is not changed.

And, as the result of the change of the tool posture at the end of a block, if the Work-piece side rotary axis position is changed larger than the parameter No.19739, the change is not done.

Moreover the change is not done when the tool posture at the start or end of the block is the singular posture, or when the tool posture becomes the singular posture during execution of the block.

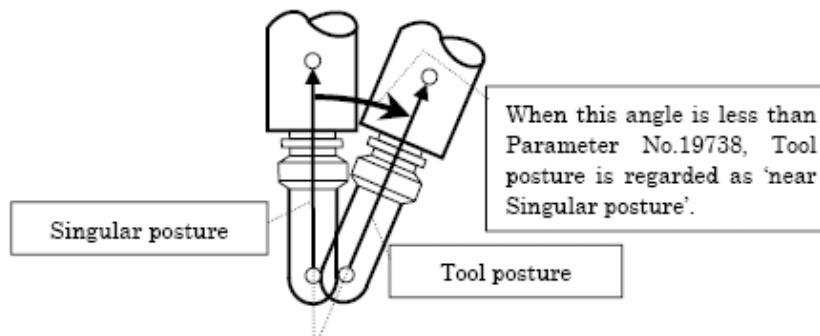


Figure 2.13: Tool posture in the near of singular posture

When the tool posture can be near Singular posture in a block, the rotary axes positions at the block end, which are calculated by the tool posture at the block end with I,J,K, are changed as follows:

Tool side rotary axis : Reversed from the singular angle.

Ex. : When the axis position was 60deg at the block end and the singular angle (position) was 20deg, the

axis position at the block end is changed to -20deg(=2*20 -60).

Work-piece side rotary axis :Same with the rotary axis at the start of the block

Ex. : When the axis position was 30deg at the block end and -150deg at the block start, the axis position at the block end is changed to -150deg.

As the result, the tool posture passes the singular posture during the block.

If the following difference between the position before the change and the position after the change of Work-piece side rotary axis is larger than the parameter No.19739, the change is not done.

|<Position before the change> - <Position after the change +/-180deg>|

And, in accordance with the parameter NPC (No.**19696#3**), the program is done without the change or the alarm P/S5421 occurs.

NOTE

In the case that the vectors of Tool length compensation at start and end of a block are the same (same direction or reverse direction), Tool Posture Control does not work and Conventional control works, because the plane made by those vectors are not fixed.

Other notes are the same as Tool Center Point control for 5-axis machining.

2.3.18 Angle of rotary axis for command type 2

Movement range is not specified

When I, J, K, Q for type 2, specify the direction of the tool more than two pairs of "computed angles" of the rotary axes usually exist.

The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the specified tool axis direction.

The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.

The following descriptions assume that there is no movement range specification (parameter No.19741 - No.19744 = 0).

Output judgment conditions for tool rotation type and table rotation type machine

The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller.

↓
↓ When the master axis moving angle is the same
↓

The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.

↓
↓ When the slave axis moving angle is the same
↓

The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓
↓ When the master axis angle is equally near to 0 degree
↓

The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Output judgment conditions for mixed type machine

The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller.

↓
↓ When the table moving angle is the same

The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.

↓
↓ When the tool moving angle is the same

The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓
↓ When the table angle is equally near to 0 degree

The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgment condition is called "movement judgment". When parameter PRI (No. **19608#5**) is 1, the movement judgments for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgment" process is explained below.

When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."

Usually, two pairs of "basic computed angles" exist. For example, assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:

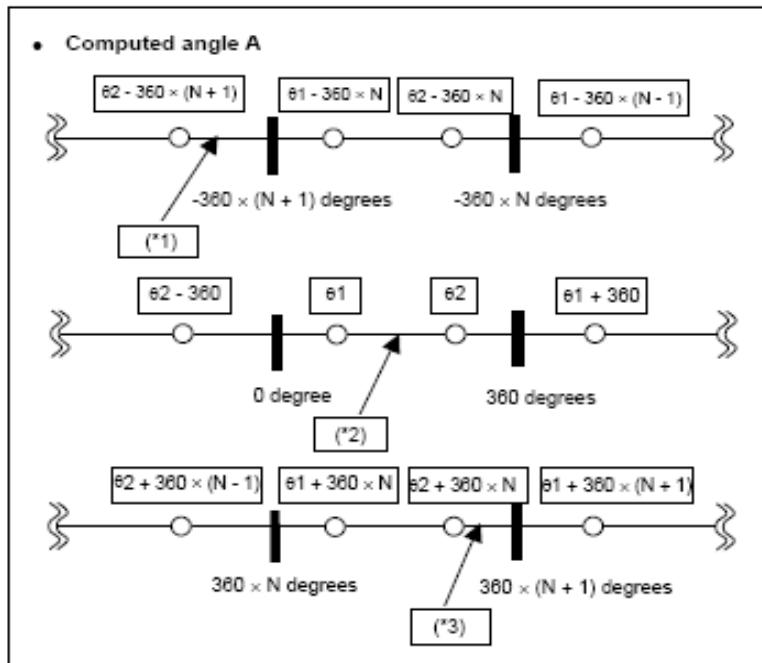
(A θ_1 degree; B φ_1 degree)

(A θ_2 degrees; B φ_2 degrees) where $\theta_1 \leq \theta_2$.

The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees $\times N$ or "basic computed angle" - 360 degrees $\times N$.

The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.

Based on the PA angle, the "movement judgment" process is done as follows (when parameter PRI (No. **19608#5**) is 0)

**Figure 2.14: Movement judgment**

When the PA angle is (*1):

The output angle is: (A $\theta_2 - 360 \times (N + 1)$ degrees; B φ_2 degrees). Namely, $\theta_2 - 360 \times (N + 1)$ degrees is adopted that is nearer to the computed angle of A, and φ_2 , which is the same group as θ_2 , is adopted as the computed angle of B.

When the PA angle is (*2):

The output angle is: (A θ_1 degrees; B φ_1 degrees). Namely, θ_1 degrees are adopted that is nearer to the computed angle of A, and φ_1 , which is the same group as θ_1 , is adopted as the computed angle of B.

When the PA angle is (*3):

The output angle is: (A $\theta + 360 \times N$ degrees; B φ_2 degrees). Namely, $\theta + 360 \times N$ degrees is adopted that is nearer to the computed angle of A, and φ_2 , which is the same group as θ_2 , is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgment" is made for rotary axis B (slave) according to the "output judgment conditions."

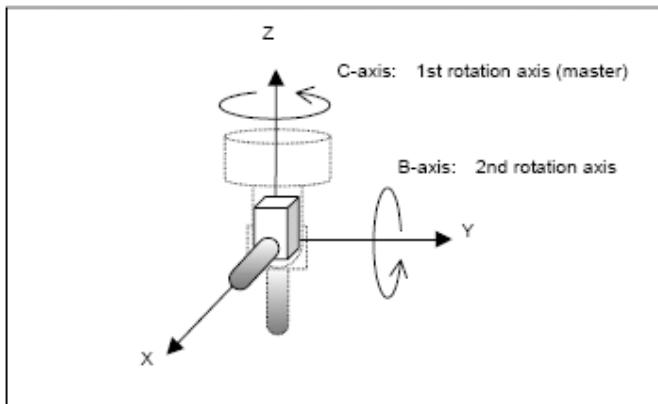
If the "output angle" of rotary axis A is determined by the "movement judgment" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis

B.

Similarly, if the "output angle" of rotary axis B is determined by the "movement judgment" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

The "output angle" is explained below using a tool rotation type machine as an example.

This example illustrates a machine having a "BC type tool axis Z."

**Figure 2.15: BC tool axis Z**

The following two pairs of "computed basic angles" exist that direct the tool axis toward the + X-axis direction.
 (B 90 degrees; C 180 degrees)
 (B 270 degrees; C 0 degree)

When the current rotary axis angles are (B -70 degrees; C 30 degrees). The "output angles" are (B -90 degrees; C 0 degree). 0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

When the current rotary axis angles are (B 80 degrees; C 500 degrees). The "output angles" are (B 90 degrees; C 540 degrees). 540 degrees (180 degrees, 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

When the current rotary axis angles are (B 60 degrees; C 90 degrees). The "output angles" are (B 90 degrees; C 180 degrees).

Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

When the current rotary axis angles are (B 180 degrees; C 90 degrees)

The "output angles" are (B 270 degrees; C 0 degree).

Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree. That is, the pair is adopted whose C axis angle is 0 degree and whose B axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.

In that case, the master axis does not move from the current angle. An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

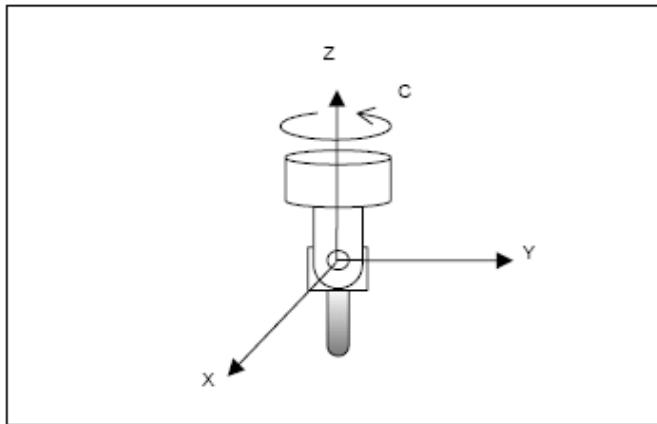


Figure 2.16: BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

2.3.19 Movement range is specified

If the upper and lower limits of the movement range of the rotary axis are specified using parameters No.19741 to No.19744, the rotary axis will move only within the specified range when the direction is specified using I, J, K, Q command for type 2 control.

Although the procedure for determining the angles is the same as that used "when the movement range is not specified," the "output angles" need to be selected from **those computed angles that are within the specified movement range for both axes**.

Output judgment conditions for tool rotation type and table rotation type machine

1. Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller represents the "output angles."

↓

↓ When the master axis moving angle is the same

↓

2. The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.

↓

↓ When the slave axis moving angle is the same

↓

3. The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓

↓ When the master axis angle is equally near to 0 degree

↓

4. The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Output judgment conditions for mixed type machine

Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose table (second rotary axis) moving angle is smaller represents the "output angles."

↓
↓ When the table moving angle is the same

↓
The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.

↓
↓ When the tool moving angle is the same

↓
The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓
↓ When the table angle is equally near to 0 degree

↓
The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

When parameter PRI (No. **19608#5**) is 1, the movement judgments for the first rotary axis and second rotary axis are made in reverse order.

CAUTION

1. If the lower limit of the movement range is larger than the upper limit, alarm PS5459 occurs when G43.5 is specified.
2. If no "computed angle" is found within the movement range because the range is too small, alarm PS5459 occurs.
3. If 0 is set for both parameters that specify the upper and lower limits of the movement range, the tool operates assuming that there is no range specification.
4. When the rotary axis rollover function or rotary axis control function is used (in which case, set parameter No. **1260** (amount of movement per rotation of the rotary axis) to 360 degrees), the tool does not move beyond 0 degree (360 degrees) (does not take the shortcut) if the movement range is set between 0 and 360 degrees. Also, do not specify a negative value or a value larger than 360 degrees for the movement range.

An example of the "movement judgment" process is given below.

Assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:

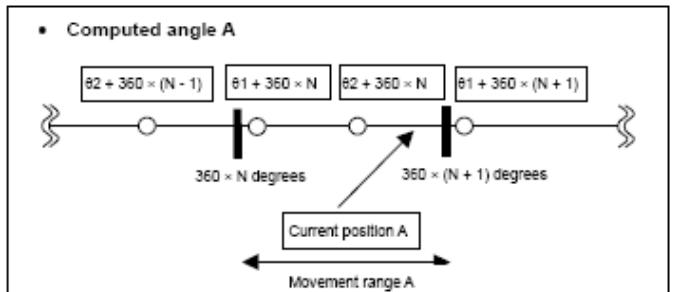
(A 01 degree; B φ1 degree)

(A 02 degrees; B φ2 degrees) where $01 \leq 02$.

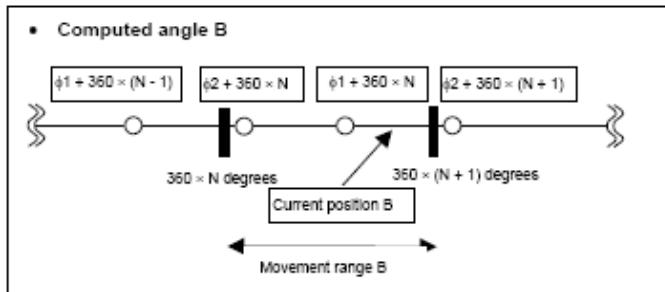
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees $\times N$ or "basic computed angle" - 360 degrees $\times N$.

Assume that the current positions and movement ranges of rotary axis

A (master) and rotary axis B (slave) are as shown in the following figure.



"Computed angle of rotary axis A and its current position and movement range"



"Computed angle of rotary axis B and its current position and movement range"

When the two axes have a positional relationship as shown in the figure, the output angle of rotary axis A is $(\theta_2 + 360 \times N)$ degrees and that of rotary axis B is $(\varphi_2 + 360 \times N)$ degrees (when parameter PRI (No. 19608#5) is set to 0).

More concretely, from the computed angles obtained for rotary axis A, the nearest angle within the movement range, i.e. $\theta_2 + 360 \times N$ degrees, is first adopted. Then, from the computed angles obtained for rotary axis B, the angle belonging to the same group as θ_2 , i.e. $\varphi_2 + 360 \times N$, is adopted.

Note that, in this example, the output angles and moving direction differ depending on whether the movement range is specified or not (0 to 360 degrees), even if N is set to 0 and coordinates are rounded to 0 to 360 degrees.

Namely, if the movement range is not specified, $\theta_1 + 360$ degrees nearest to the current position is adopted as the computed angle for rotary axis A and, from the computed angles belonging to the same group as θ_1 , φ_1 degrees nearest to the current position is adopted as the computed angle for rotary axis B. Rotary axis A moves in the plus direction. As its coordinate is rounded to 360 degrees, rotary axis A reaches θ_1 degrees while moving in the plus direction.

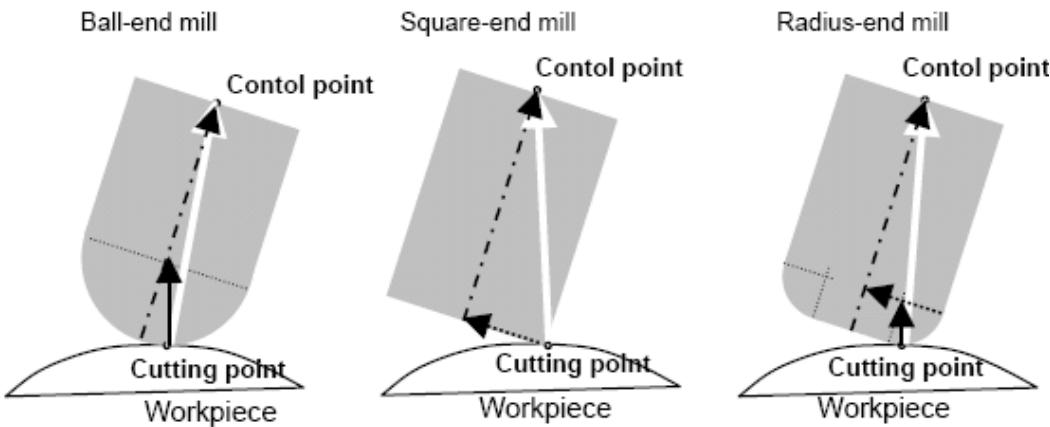
By contrast, when the movement range is set to 0 to 360 degrees, the output angles are (A θ_2 degrees; B φ_2 degrees). Neither rotary axis A nor B moves in a way that it exceeds 0 degree (360 degrees).

2.3.20 Cutting point command

During the mode of Cutting point command, the Control point (Machine position) for each kind of tool, Ball-end mill, and Square-end mill and Radius-end mill (Tool with Corner-R) is calculated from the commanded cutting point, according to Tool length offset commanded by H, Tool radius offset and Corner-R commanded by D.

And the tool moves with the calculated Control point, as follows :

Here, the Corner-R offset data is the offset data set on the offset screen (refer to chapter 1.2). When Corner-R offset data is equal to Tool radius offset, the tool is regarded as a Ball-end mill. And, when Corner-R is equal to 0, the tool is regarded as a Square-end mill.



	Tool offset vector
	Tool lengthoffset The Direction is commanded by (α, β) or (I, J, K) . The length is commanded by H.
	Tool radiusoffset The Direction is perpendicular to Tool length offset on the plane made by Tool length offset and Corner-R offset. The length is commanded by D.
	Corner-R offset The Direction is perpendicular to the cutting surface, commanded by (I, J, K) (following ",L2"). The length is commanded by D.

In the block of G43.8/G43.9, when a part of (I, J, K) following ",L2" is not commanded, the omitted command is regarded as 0, and when all of (I, J, K) following ",L2" are not commanded , the tool direction is regarded as the standard tool direction, specified by the parameters No. 19697, 19698 and 19699.

After the block of G43.8/G43.9, when a part of (I, J, K) following ",L2" is not commanded, the omitted command is regarded as 0, and when all of (I, J, K) following ",L2" are not commanded , the direction is regarded as the direction of the previous block. When all of (I, J, K) following ",L2" is 0, an alarm (PS5464) occurs. The number of effective digits below the decimal point of (I, J, K) following ",L2" is the same as the Increment System.

N10 G90 G00 X0.0 Y0.0 Z0.0 C0.0 ;	Move to starting point
N20 G43.8 H1 D1 , L2 I-1..0 J0.0 K10.0 ;	Cutting point command start
N30 G01 X20.0 Y0.0 Z0.0 A0.0 C0.0 F1800., L2 I-1..0 J0.0 K10.0 ;	Instruct the F code before ",L2"
N40 G01 X20.0 Y20.0 Z0.0 A10.0 C5.0 ;	Direction of (I, J, K) following ",L2" is the same as the previous block.
N50 G01 X0.0 Y20.0 Z0.0 A20.0 C7.0 ,L2 I0.0 J-1.0 K1.0 ;	
...	
N200 G49	Cutting point cancel

2.3.21 Singular posture

In the case of Square-end mill or Radius-end mill, the tool is instability when the tool direction is perpendicular to cutting surface because the coordinate corresponding to the contact point of the tool with the cutting surface is not decided to one point. This posture is called singular posture. In Cutting Point Command, the tool in singular posture can be prevented from being unstable by setting the parameter (No.11262). Angle (0.0~90.0(deg)) is set to the parameter (No.11262). The tool posture is considered to be near singular posture if the angle from the direction of tool length offset to the perpendicular to cutting surface is smaller than the parameter (No.11262). The singular posture does not exist when the parameter (No.11262)=0.0 is set. In the case of near singular posture,

"Vector from the program point to Tool center point" is replaced by "Vector from the cutting point to Tool center point" immediately before becoming near singular posture (Fig. 1.1.1 (c)), and the instability peculiar to the singular posture is avoided. At this time, the cutting point shift from the program point(Fig. 1.1.1 (d)).

Don't set a too big value in the parameter (No.11262), because the larger the parameter (No.11262) is, the more greatly the cutting point shift from the program point.

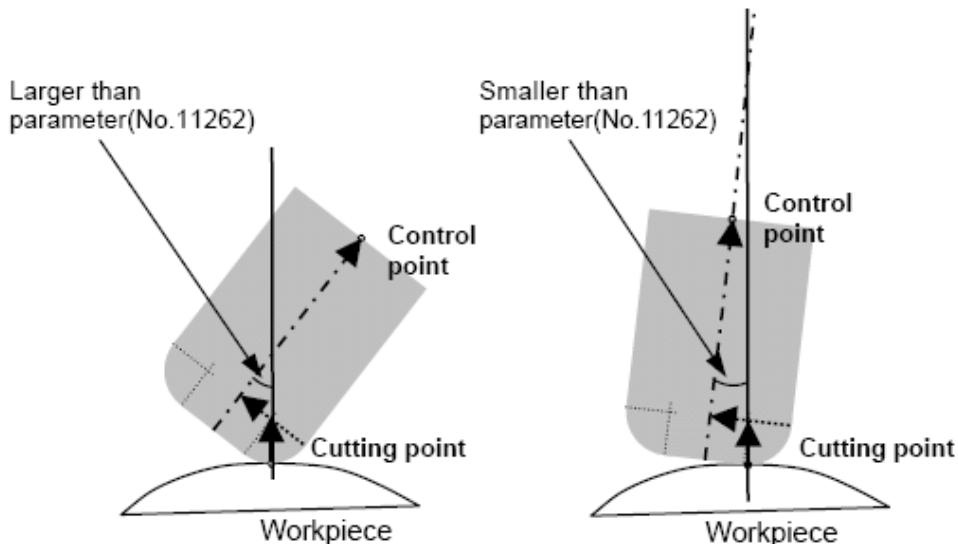


Figure 2.17: near singular posture (right)

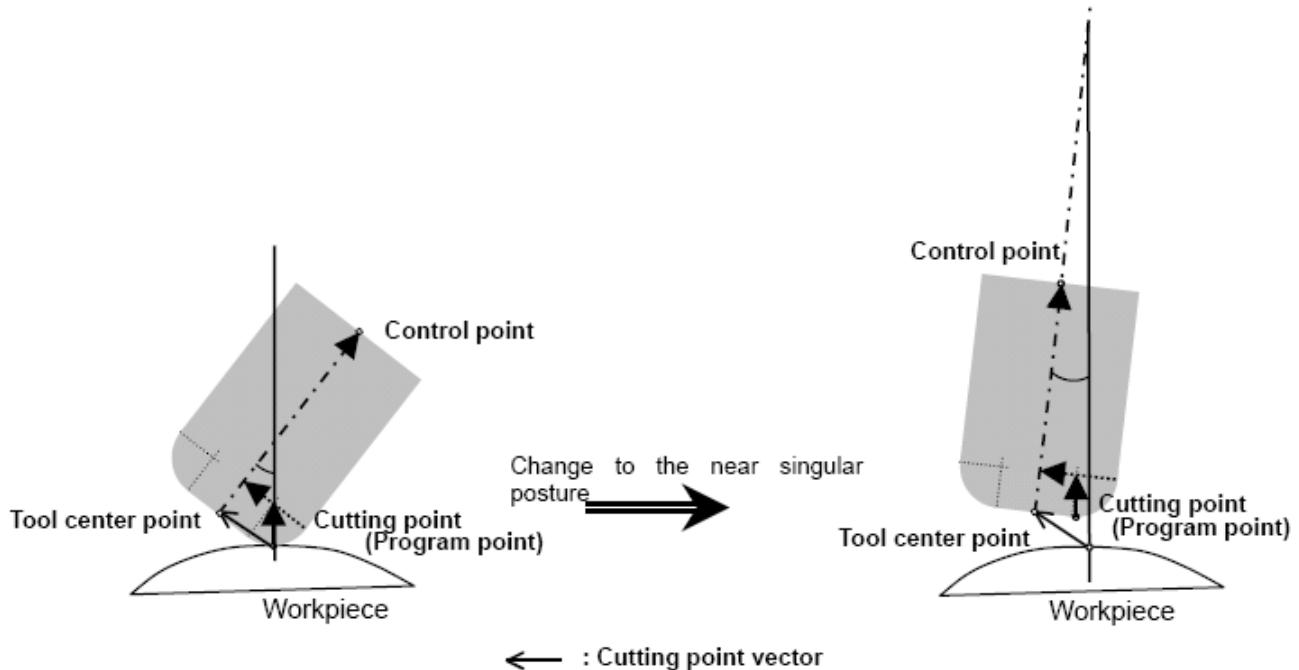


Figure 2.18: In the case of near singular posture, "Vector from the program point to Tool center point" is replaced by "Vector from the cutting point to Tool center point" immediately before becoming near singular posture

Restrictions

- Cutting point control is available for G00 and G01 mode.
- In the case of tool rotation type machine configuration, if automatic operation is stopped and manual intervention is performed on the rotation axis, the alarm PS5464 occurs.
- Other restrictions is the same as Tool center point control.

2.4 Programming Examples

2.4.1 Tool rotation type machine

Explanations are given below assuming a machine configuration in which a tool rotation axis that turns around the Y-axis is located beneath another tool rotation axis that turns around the Z-axis. (See Fig. 21.1 (j).)

If linear interpolation is specified for the X-, Y-, and Z-axes when a workpiece coordinate system is used as the programming coordinate system, control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the tool rotates.

Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

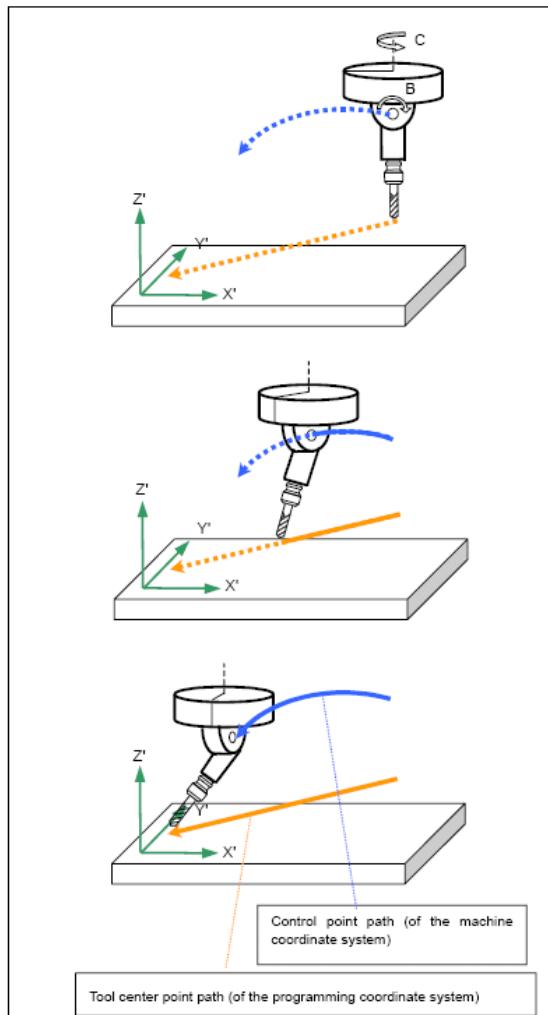
In the case of a machine having two tool rotation axes, the table does not rotate with respect to the workpiece coordinate system even if the rotary axes move. Therefore, the programming coordinate system always matches the workpiece coordinate system, regardless of whether parameter WKP (No. 19696#5) is set to 0 or 1.

TCP type 1

O100 (Sample program 1) ;	
N1 G00 G90 B0 C0 ;	
N2 G55 ;	Prepares the programming coordinate system.
N3 G43.4 H01 ;	Starts tool center point control. H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;	Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 C60.0 B45.0 F500 ;	Linear interpolation
N6 G49;	Cancels tool center point control.
N7 M30;	

TCP type 2

O100 (Sample program 1) ;	
N1 G00 G90 B0 C0 ;	
N2 G55 ;	Prepares the programming coordinate system.
N3 G43.5 H01 ;	Starts tool center point control. H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;	Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 I1.0 J1.732 K2.0 F500 ;	Linear interpolation
N6 G49;	Cancels tool center point control.
N7 M30;	



2.4.2 Table rotation type machine

Explanations are given below assuming a machine configuration (turnnion) in which a rotation table that turns around the Y-axis is located above another table rotation axis that turns around the X-axis.

(See Figure 2.19)

If linear interpolation is specified for the X-, Y-, and Z-axes on the programming coordinate system and if the rotary axis that moves the rotation table is specified (in the case of type 1) or the tool direction is specified (in the case of type 2), control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the rotation table rotates.

Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

TCP type 1

When type 1 is selected and the coordinate system fixed on the table is used as the programming coordinate system (Parameter WKP (No.19696#5) = 0):

O200 (Sample program 2) ;	
N1 G00 G90 B0 C0 ;	Prepares the programming coordinate system.
N2 G55 ;	
N3 G43.4 H01 ;	Starts tool center point control. H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;	Moves to the start point.
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ;	Linear interpolation
N6 G49;	Cancels tool center point control.
N7 M30;	

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system (Parameter WKP (No.**19696#5**) = 1):

O201 (Sample program 2-1) ;	
N1 G00 G90 B0 C0 ;	Prepares the programming coordinate system.
N2 G55 ;	
N3 G43.4 H01 ;	Starts tool center point control. H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;	Moves to the start point.
N5 G01 X7.574 Y47.247 Z83.052 C60.0 B45.0 F500 ;	Linear interpolation
N6 G49;	Cancels tool center point control.
N7 M30;	

TCP type 2

For type 2 (when the coordinate system fixed on the table is used as the programming coordinate system (only when parameter WKP (No.**19696#5**) is set to 0)):

O202 (Sample program 2-2) ;	
N1 G00 G90 B0 C0 ;	Prepares the programming coordinate system.
N2 G55 ;	
N3 G43.5 H01 ;	Starts tool center point control. H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;	Moves to the start point.
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ;	Linear interpolation
N6 G49;	Cancels tool center point control.
N7 M30;	

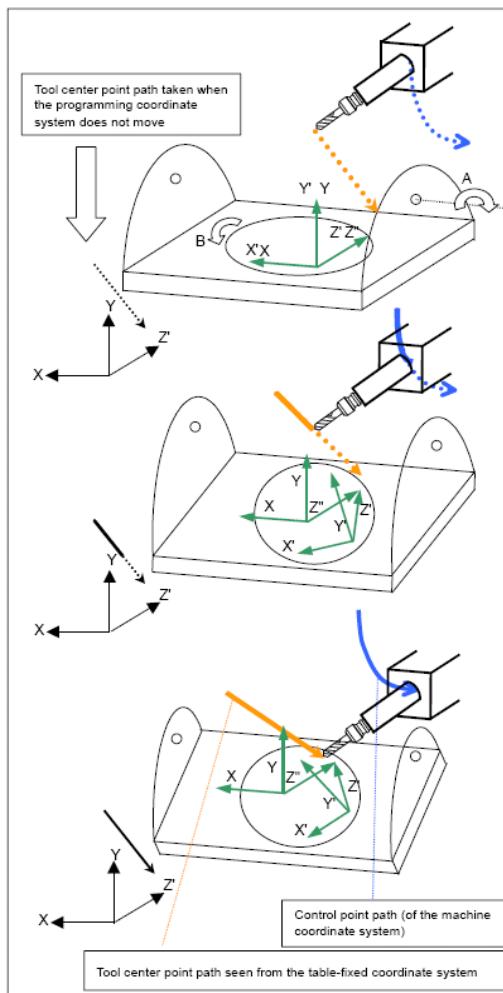


Figure 2.19: Example for table type machine

2.4.3 Mixed type machine

Explanations are given below assuming a mixed type machine configuration that has one table rotation axis (which turns around the X-axis) and one tool rotation axis (which turns around the Y-axis). (See Figure 2.20)

If linear interpolation is specified for the X-, Y-, and Z-axes on the programming coordinate system and if the rotary axis that moves the rotation table and the tool rotation axis are specified (in the case of type 1) or the tool direction is specified (in the case of type 2), control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the rotation table and tool rotate.

Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

TCP type 1

When type 1 is selected and the coordinate system fixed on the table is used as the programming coordinate system (Parameter WKP (No.19696#5) = 0):

O300 (Sample program 3) :

N1 G00 G90 B0 C0 ;

N2 G55 ;

N3 G43.4 H01 ;

N4 G00 X200.0 Y150.0 Z20.0 ;

N5 G01 X5.0 Y5.0 Z5.0 **A60.0 B45.0** F500 ;

Prepares the programming coordinate system.

Starts tool center point control. H01 is the tool compensation number.

Moves to the start point.

Linear interpolation

N6 G49; Cancels tool center point control.

N7 M30;

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system (Parameter WKP (No. **19696#5**) = 1):

O301 (Sample program 3-1) :

N1 G00 G90 B0 C0 ;

N2 G55 ;

N3 G43.4 H01 ;

N4 G00 X200.0 Y150.0 Z20.0 ;

N5 G01 X5.0 Y48.170 Z-79.772 A60.0 B45.0 F500;

N6 G49;

N7 M30;

Prepares the programming coordinate system.

Starts tool center point control. H01 is the tool compensation number.

Moves to the start point.

Linear interpolation

Cancels tool center point control.

TCP type 2

For type 2 (when the coordinate system fixed on the table is used as the programming coordinate system (only when parameter WKP (No. **19696#5**) is set to 0)):

O302 (Sample program 3-2) :

N1 G00 G90 B0 C0 ;

N2 G55 ;

N3 G43.5 H01 ;

N4 G00 X200.0 Y150.0 Z20.0 ;

N5 G01 X5.0 Y5.0 Z5.0 I2.0 J1.732 K1.0 F500 ;

N6 G49;

N7 M30;

Prepares the programming coordinate system.
Starts tool center point control. H01 is the tool compensation number.

Moves to the start point.

Linear interpolation

Cancels tool center point control.

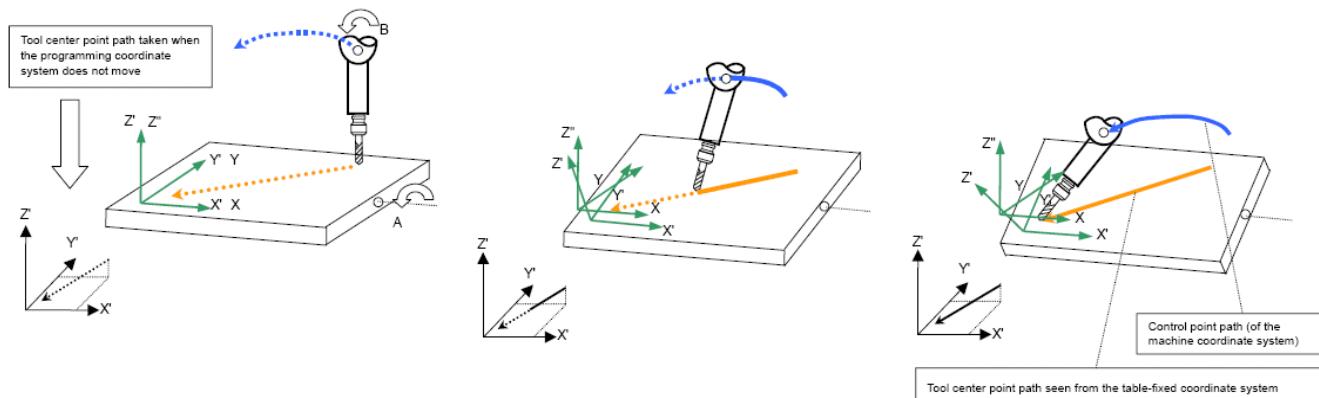


Figure 2.20: Example for mixed type machine

2.5 Limitations

Hypothetical axis of a table rotation axis	When a table rotation axis is set as a hypothetical axis, tool center point control is performed on the assumption that the table rotation axis is at 0 degrees.
Deceleration at a corner	During tool center point control, the control point may move on a curved line even if a straight line is specified. Some commands may cause the tool center point to make a sharp turn. For this reason, the tool may be decelerated if a small value

	<p>is set as the permissible speed difference (parameter No. 1783) or the permissible acceleration (parameter No. 1660 or 1737) for a corner.</p>				
Look-ahead pre-interpolation Acceleration/deceleration	<p>When using tool center point control, use look-ahead acceleration/deceleration before interpolation at the same time. If look-ahead acceleration/deceleration before interpolation is not used, alarm PS5420 is issued. For details, see "NOTE" in "Commands that can be specified during tool center point control" in "Explanation."</p>				
Cutter compensation for 5-axis machining	<p>When tool center point control is exercised together with cutter compensation for 5-axis machining on a machine of mixed type or table rotation type, specify a value in the workpiece coordinate system by setting the parameter WKP (No. 19696) to 1.</p> <p>In that case, when specifying cutter compensation for 5-axis machining before tool center point control, specify the cancellation of cutter compensation for 5-axis machining after canceling tool center point control (example 1). When specifying tool center point control before cutter compensation for 5-axis machining, specify the cancellation of tool center point control after canceling three-dimensional cutter compensation (example 2).</p> <table border="1" style="margin-left: auto; margin-right: auto;"> <tr> <td style="text-align: center; padding: 5px;">Example 1</td> <td style="text-align: center; padding: 5px;">Example 2</td> </tr> <tr> <td style="text-align: center; padding: 5px;">G41.2 D1 : G43.4 H1 : G49 : G40</td> <td style="text-align: center; padding: 5px;">G43.4 H1 : G41.2 D1 : G40 : G49</td> </tr> </table> <p>When specifying cutter compensation for 5-axis machining first, the block for canceling tool center point control controls buffering. Note that, in a block preceding the G49 block, the compensation vector for cutter compensation for 5-axis machining is directed toward the vertical direction of movement.</p>	Example 1	Example 2	G41.2 D1 : G43.4 H1 : G49 : G40	G43.4 H1 : G41.2 D1 : G40 : G49
Example 1	Example 2				
G41.2 D1 : G43.4 H1 : G49 : G40	G43.4 H1 : G41.2 D1 : G40 : G49				
Parallel axis control	<p>When exerting tool center point control together with parallel axis control, make sure that the master and slave axes are properly aligned and keep the parking signal off.</p>				
Programmable mirror image	<p>Note the following points when making a programmable mirror image:</p> <p>In the case of tool center point control of type 1 Mirroring the linear axis alone does not create a mirror image for the rotary axis. To make the direction of the tool symmetrical, it is necessary to make a mirror image for the rotary axis as well.</p> <p>In the case of tool center point control of type 2 When parameter MIR (No.19608#6) is set to 0, mirroring the linear axis alone does not create a mirror image for I, J, K. Note that the inclination direction specified by Q is the movement direction of the tool center point after mirroring. When parameter MIR (No.19608#6) is set to 1, mirroring the linear axis automatically creates a mirror image for I, J, K. A mirror image cannot be put on the rotary axis directly.</p>				

3 Cutter Compensation for 5-axis machining

For machines having multiple rotary axes for freely controlling the orientation of a tool axis, this function calculates a tool vector from the positions of these rotary axes. The function then calculates a compensation vector in a plane (compensation plane) perpendicular to the tool vector and performs three-dimensional cutter compensation.

3.1 Cutter Compensation in Tool Rotation Type Machine

In a 5-axis machine having two tool rotation axes as shown in Figure 3.1, this function can perform cutter compensation.

Shown below is a 5-axis machine that has tool rotation axis B on the Y-axis and tool rotation axis C on the Z-axis.

This machine configuration is used as a sample configuration in the following explanation unless otherwise noted.

Cutter compensation in tool rotation machines is classified into two types according to the way of machining: tool side offset and leading edge offset.

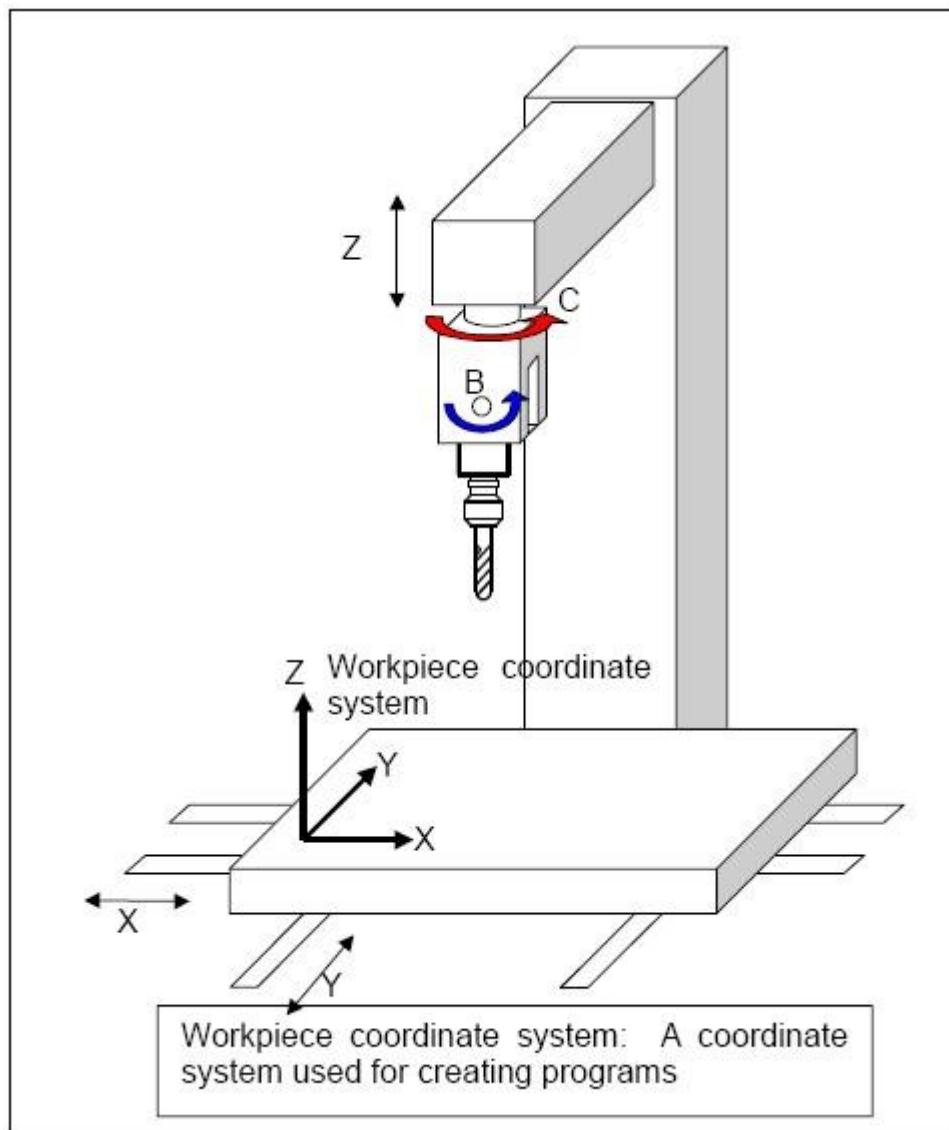


Figure 3.1: Tool rotation type machine

3.1.1 Tool side offset

This type of cutter compensation performs three-dimensional compensation in a plane (compensation plane)

perpendicular to the tool vector.

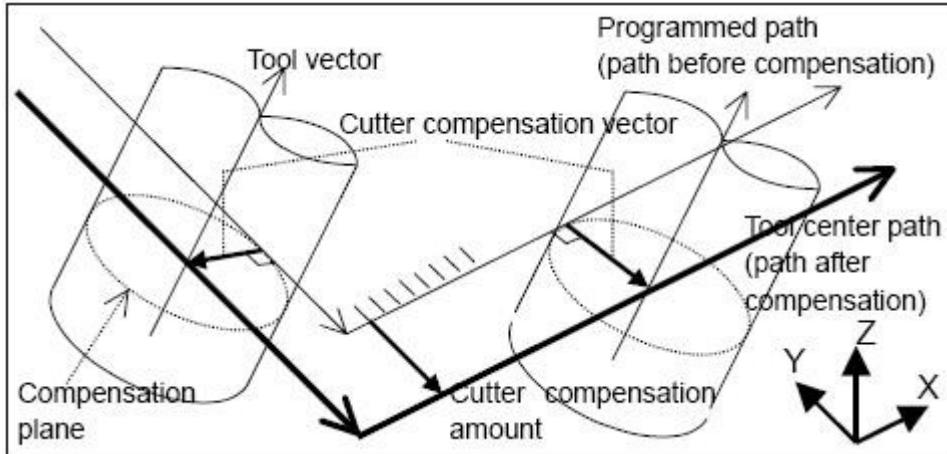


Figure 3.2: Tool side offset

Tool side offset Type 2

G41.2 (or G42.2) IP_D_ ;	Start cutter compensation
IP_I_J_K_ ;	
G41.2	Cutter compensation left (group 07)
G42.2	Cutter compensation right (group 07)
IP_	Value specified for axis movement (including rotary axis)
D_	Code specifying the cutter compensation amount (1 to 3 digits)

NOTE

1. If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2. In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3. If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4. If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. **1260** (mount of travel per rotation about the rotation axis).
5. Only tool side offset provides type 2 commands. Leading edge offset, described later, does not provide type 2 commands.

Tool side offset type 1

G41.6 (or G42.6) IP_I_J_K_D_Q_;	Start cutter compensation
IP_I_J_K_ ;	
G41.6	Cutter compensation left (group 07)
G42.6	Cutter compensation right (group 07)
IP_	Value specified for axis movement (including rotary axis)

D_	Code specifying the cutter compensation amount (1 to 3 digits)
Q_	Tool's angle of gradient (in degrees)
I_J_K_	Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (workpiece coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

NOTE

When a movement perpendicular to the next movement (bit 1 (SUV) of parameter No. **5003** is set to 1) is specified as the operation performed at the time of startup or cancellation, a move command such as X_Y_Z_ must not be specified in the G41.2 and G42.2 block.

Cancelling tool side offset

G40 IP_ ;	Cancellation of cutter compensation (group 07)
IP_	Value specified for axis movement (including rotary axis)

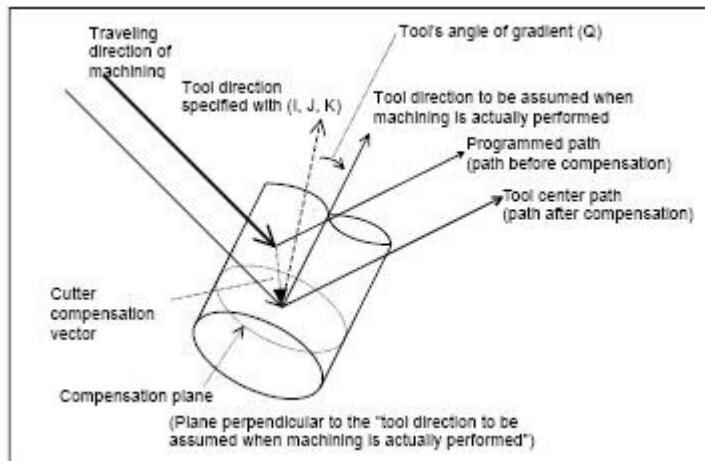
3.1.2 Explanation

Tools angle of gradient in type 2

For type 2 of cutter compensation for 5-axis machining, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Figure 3.3.)

Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.

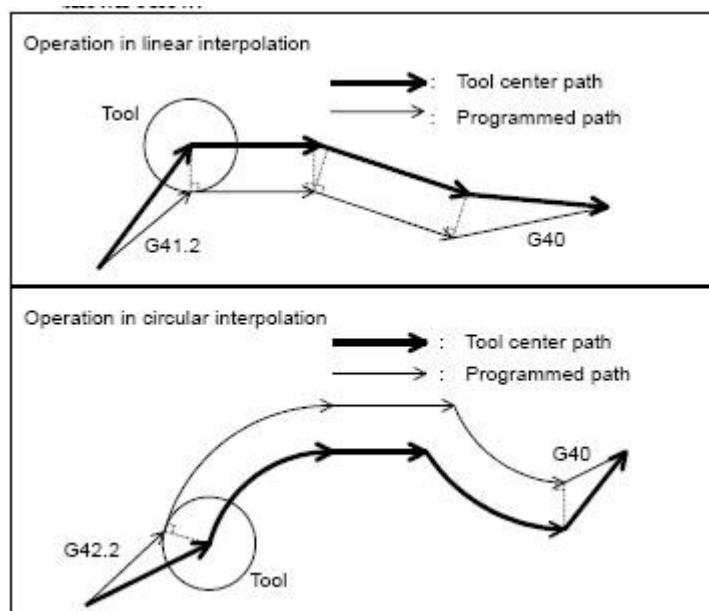
If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary

**Figure 3.3: Tool's angle of gradient in type 2****(Example)****G41.6 I_ J_ K_ H_ Q2.0**

Command this to perform machining by inclining the tool's traveling direction twice

Operation at startup and cancellation**1. Type A**

The tool is moved in the same way as for cutter compensation as shown below.

**Figure 3.4: Operation at startup and cancellation (type A)****2. Type B**

The tool is moved in the same way as for cutter compensation as shown below.

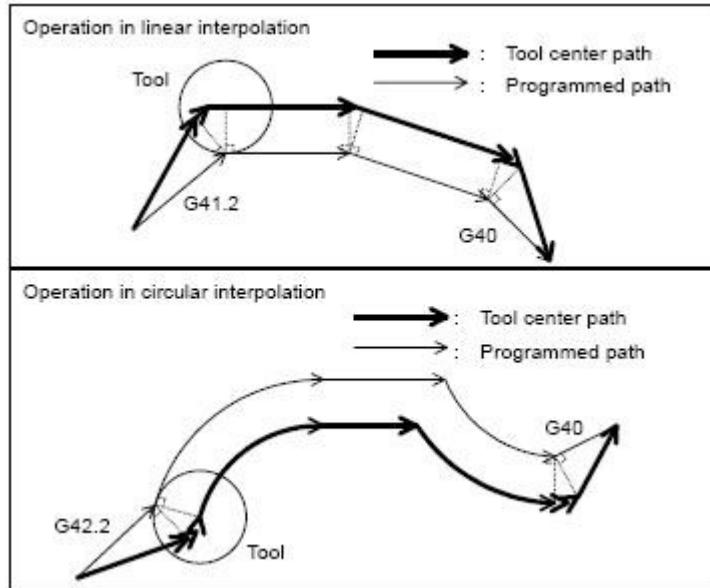


Figure 3.5: Operation at startup and cancellation (type B)

3. Movement perpendicular to the next movement

When G41.2, G42.2, or G40 is specified, a block that moves the tool linearly by the amount of cutter compensation in a direction perpendicular to the movement direction of the next block is inserted as shown below.

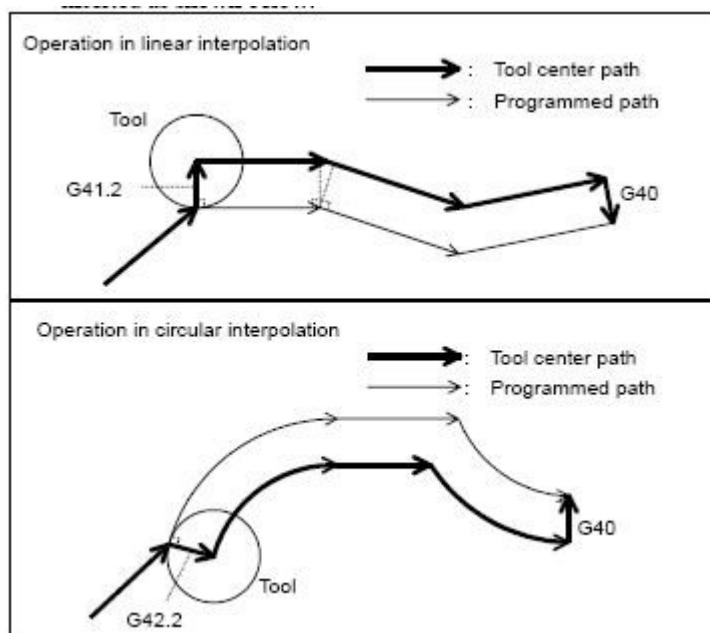


Figure 3.6: Operation at startup and cancellation (In a direction perpendicular to the next movement)

NOTE

When the movement direction is perpendicular to the next movement (bit 1 (SUV) of parameter No. **5003** is set to 1), the following conditions must always be satisfied at startup and cancellation:

1. A block specifying G40, G41.2, or G42.2 must be in the G00 or G01 mode.
2. A block specifying G40, G41.2, or G42.2 must contain no move command.
3. The block next to a block specifying G41.2 or G42.2 must contain move command G00, G01, G02, or G03.

Operation during compensation

Operations such as change of the offset direction and offset value, retention of a vector, and interference checks are performed in the same way as for cutter compensation. However, G39 (corner rounding) cannot be specified. So, note the following:

- (1) When the tool center path goes outside the programmed path at a corner, a linear movement takes place at the corner without inserting an arc. When the tool center path goes inside the programmed path, nothing is inserted.

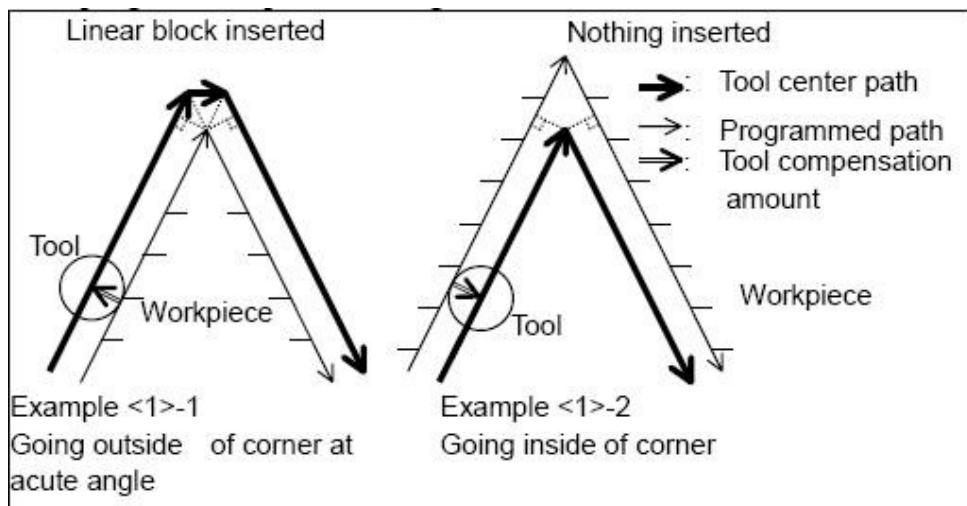


Figure 3.7: Operation during compensation <1>-1 and <1>-2

In the above examples, the terms "inside" and "outside" denote how the tool center path is positioned with respect to the programmed path. In the figure below, example <1>-3 shows the same relationship between the tool center path and programmed path as example <1>-1 and indicates that the tool center path is outside the programmed path; example <1>-4 shows the same relationship as example <1>-2 and indicates that the tool center path is inside the programmed path.

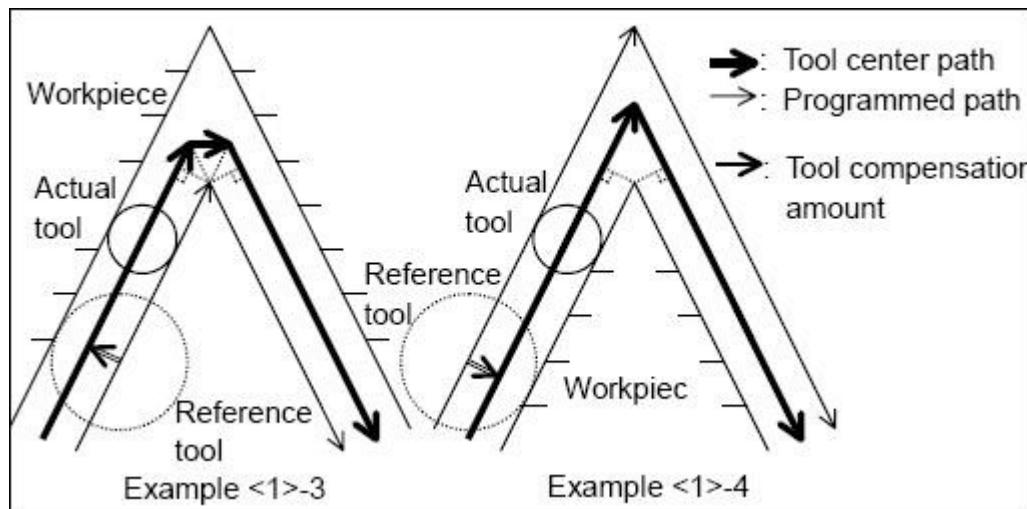
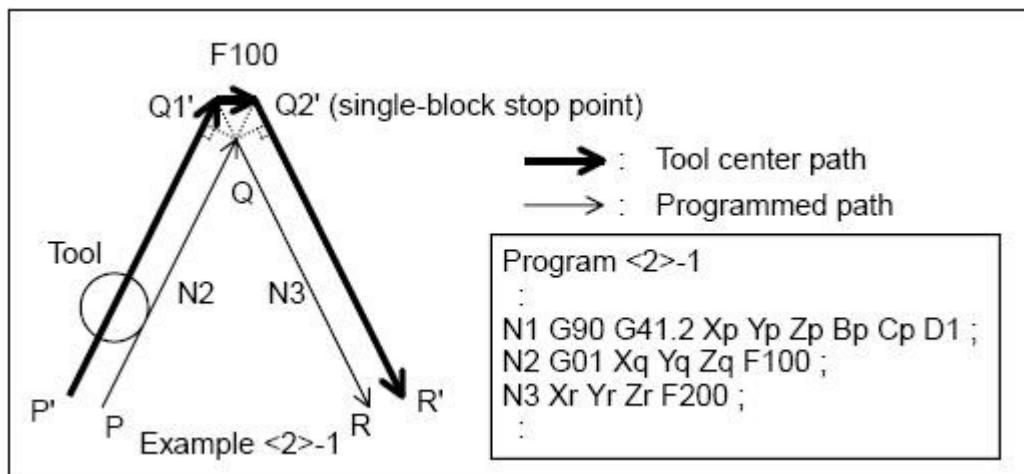


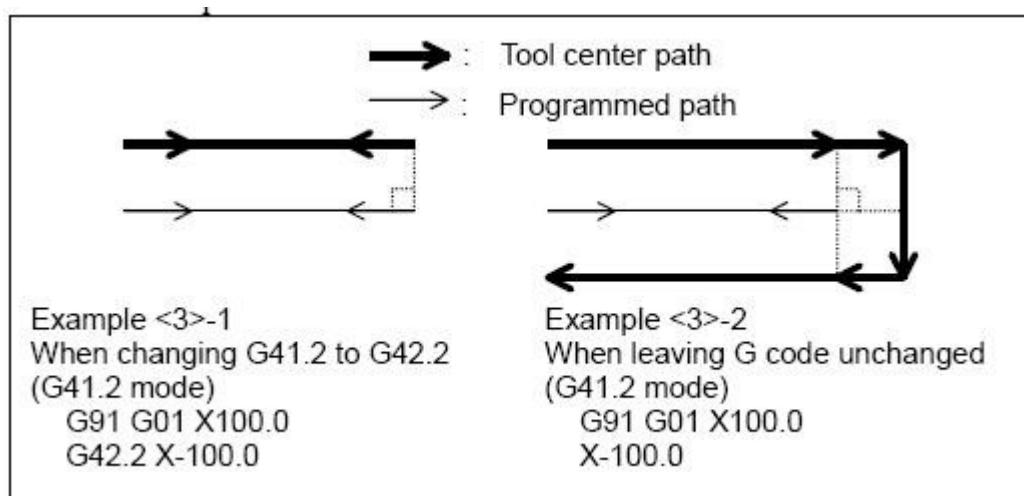
Figure 3.8: Operation during compensation <1>-3 and <1>-4

- (2) When the tool moves at a corner, the feedrate of the previous block is used if the corner is positioned before a single-block stop point; if the corner is after a single-block stop point, the feedrate of the next block is used.

**Figure 3.9: Operation during compensation (2)**

In the above example, the single-block stop point of N2 is Q2', so the feedrates along paths P'-Q1' and Q1'-Q2' are both F100.

- (3) When a command that makes the tool retrace the path of the previous block is specified, the tool path can match the locus of the previous block by changing the G code to change the offset direction. If the G code is left unchanged, the operation shown in example (3) - 2 results:

**Figure 3.10: Operation during compensation <3>**

Interference check when the compensation plane changes

An interference check is made when the compensation plane (a plane perpendicular to the tool vector) has changed.

Example:

If the following program is executed, an alarm PS0041 (over-cutting due to cutter or tool nose radius compensation) is issued at N4:

```

O100 F3000 ;
N1 G90 G00 X0 Y0 Z0 A-46 C180 ;
N2 G41.2 D1 ;
N3 G01 X100 ;
N4 Y-200 Z-200 ;
N5 A45 ;
N6 Y-400 Z0 ;
N7 X0 ;
N8 Y-200 Z-200 ;
N9 A-46 ;
N10 Y0 Z0 ;
N11 G40 ;
M30 ;

```

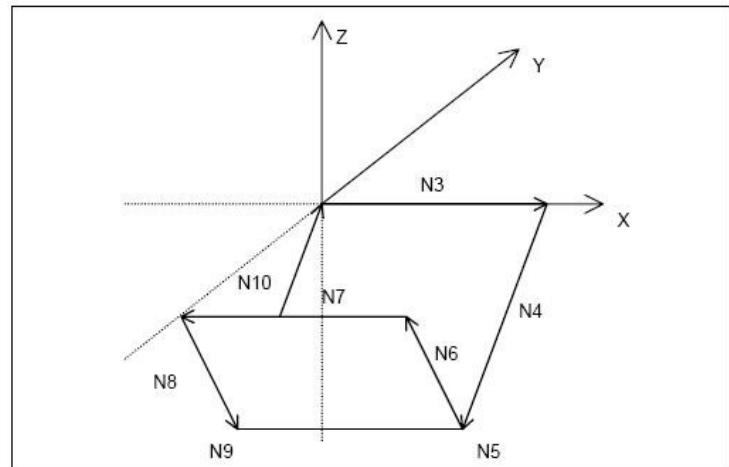


Figure 3.11: Conceptual diagram

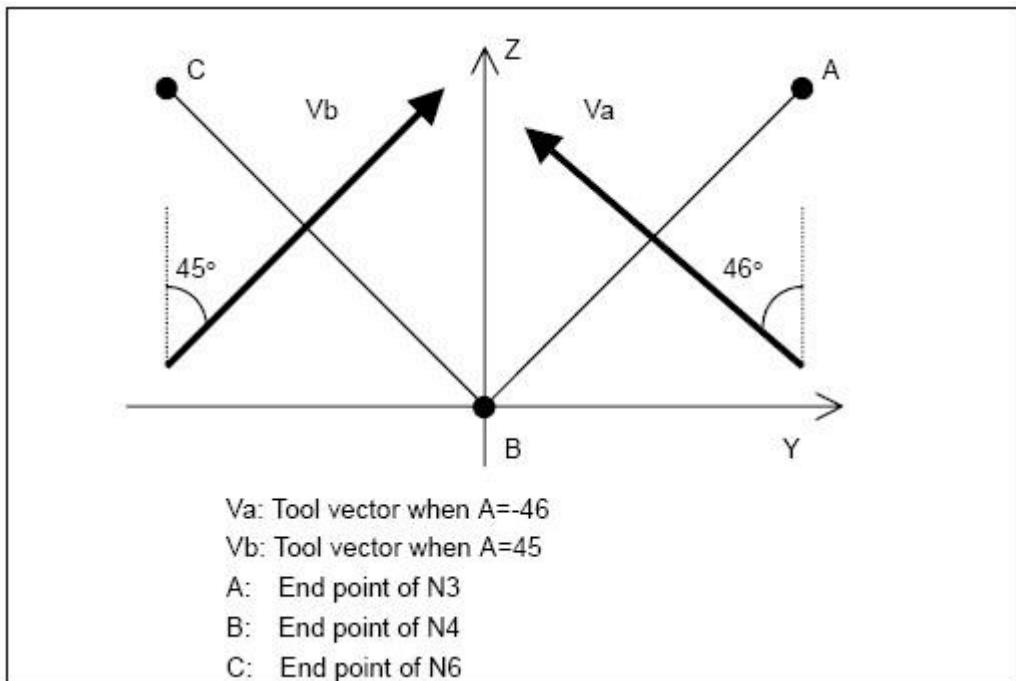


Figure 3.12: Tool vector

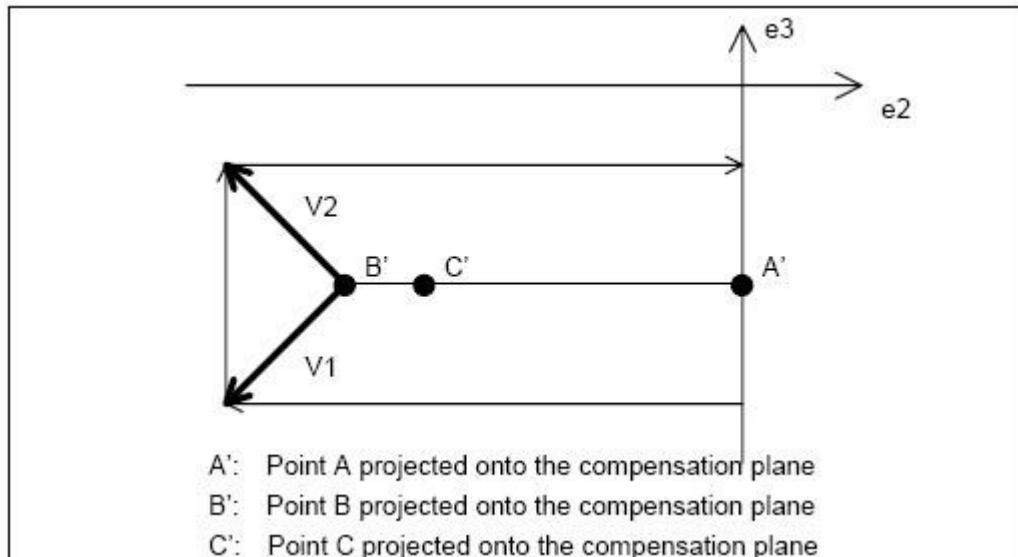


Figure 3.13: Compensation vector at the end point (point B) of N4 (in the compensation plane)

The movement direction of A'B' is opposite to that of B'C', so two compensation vectors V1 and V2 are produced at point B' (the end point of N4). In such a case, there is a possibility of over-cutting, so an alarm PS0041 is issued at N4.

1. Conditions for issuing the interference alarm

Suppose that a move command for a rotary axis makes the tool vector change significantly from one block to another. In this case, an interference alarm is assumed because compensation vectors are regarded as being generated in wrong directions when the path angle difference in the compensation plane is large, even though the angle difference between the directions of compensation vectors to be generated by those blocks is small.

Here, the compensation plane is perpendicular to the tool direction (V_a in the figure below) of the first of the two blocks. Specifically, the conditions listed below are used for issuing the alarm.

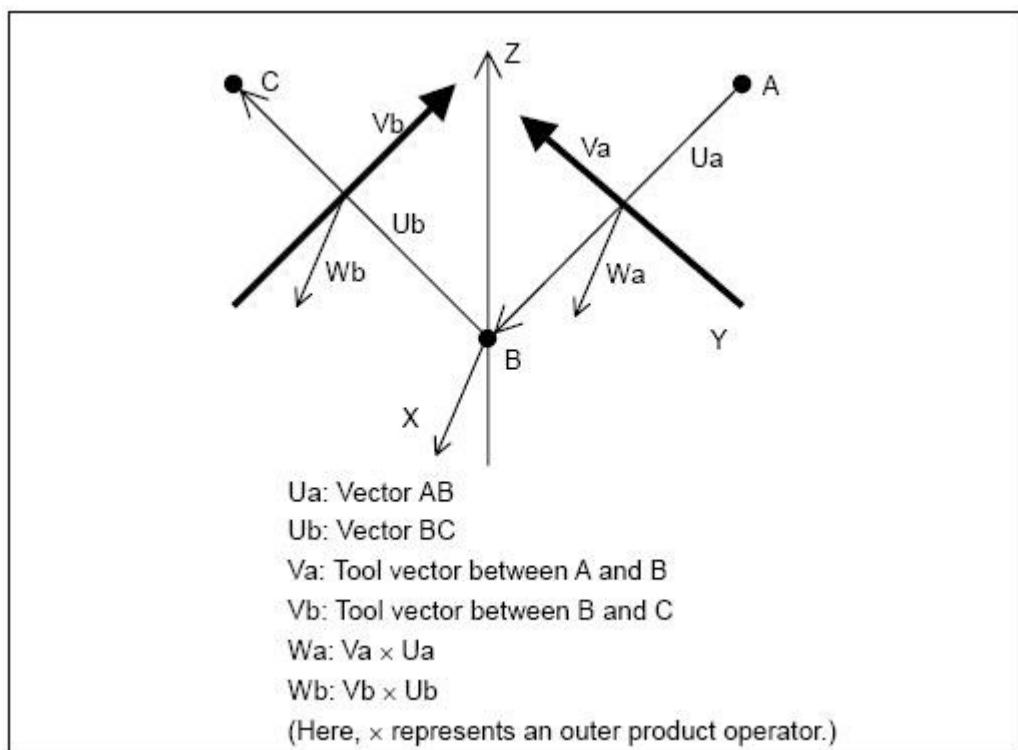


Figure 3.14: Conceptual diagram

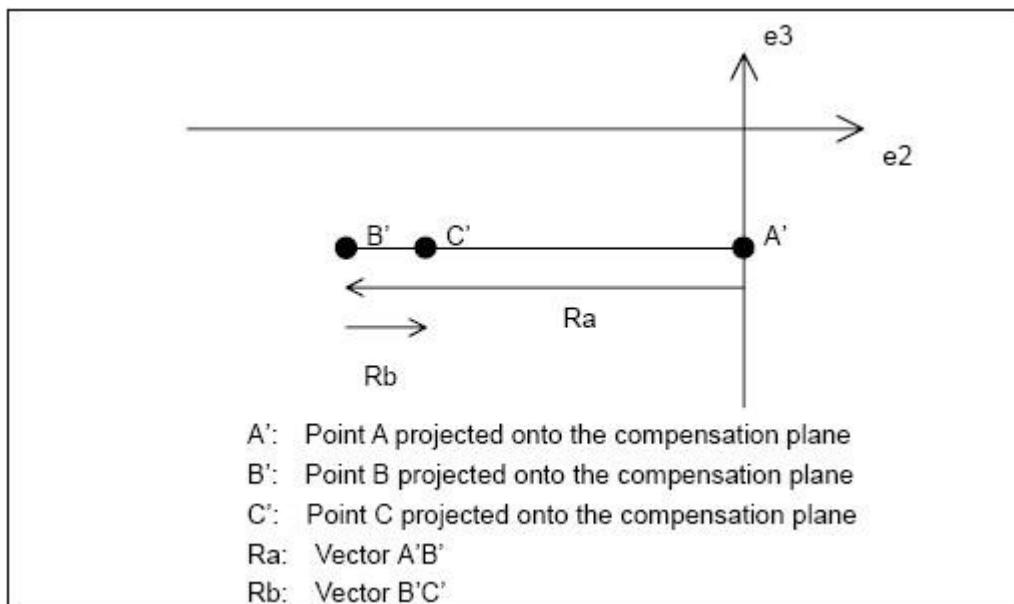


Figure 3.15: Programmed path before and after the end point (point B) of N4 (in the compensation plane)

When all the following conditions are satisfied, an alarm PS0041) is issued:

- (1) The tool vector changes significantly.

α : Angle for determination set in parameter No. **19635** (The default is 45° .)

$(V_a, V_b) \leq \cos(\alpha)$ (where, (V_a, V_b) means an inner product.)

- (2) The difference between the directions of the compensation vectors to be generated is small.

W_a : Direction of a compensation vector to be generated by block AB

W_b : Direction of a compensation vector to be generated by block BC

$$W_a = V_a \times U_a$$

$$W_b = V_b \times U_b$$

$$(W_a, W_b) \geq 0$$

- (3) The path angle difference in the compensation plane is large. $(R_a, R_b) < 0$

- (2) Suppressing the issue of the alarm with a Q command

By inserting a Q command into a block that resulted in the alarm, the issue of the alarm can be suppressed.

- (1) Q1 command

By inserting a Q1 command, a perpendicular vector is generated.

Example: N4 Y-200 Z-200 Q1

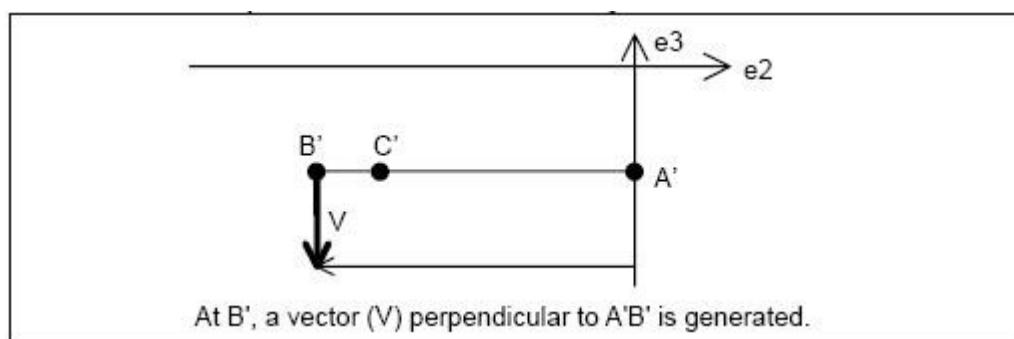


Figure 3.16: Q1 command

A perpendicular vector can also be generated by specifying G41.2 or G42.1 in the next block as follows:

Example: N6 G41.2 Y-400 Z0

(2) Q2 command

With a program specifying a linear-to-linear connection, up to two compensation vectors are generated. In this case, the second vector is deleted by inserting a Q2 command. The Q2 command has no effect on circular interpolation.

Example: N4 Y-200 Z-200 Q2

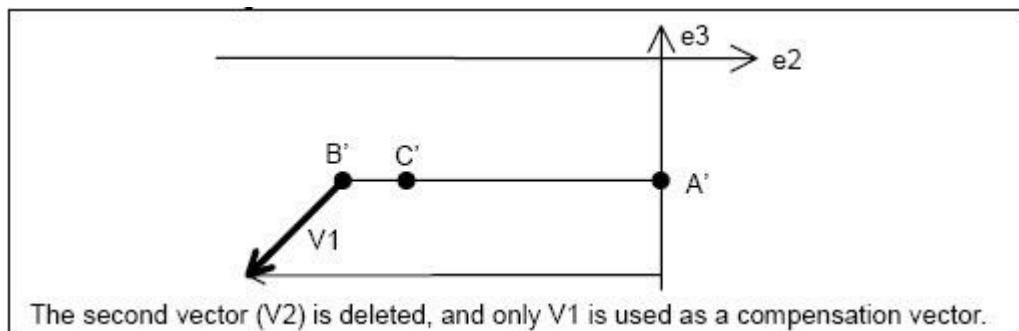


Figure 3.17: Q2 command

(3) Q3 command

By inserting a Q3 command, the issue of the alarm can be suppressed.

Example: N4 Y-200 Z-200 Q3

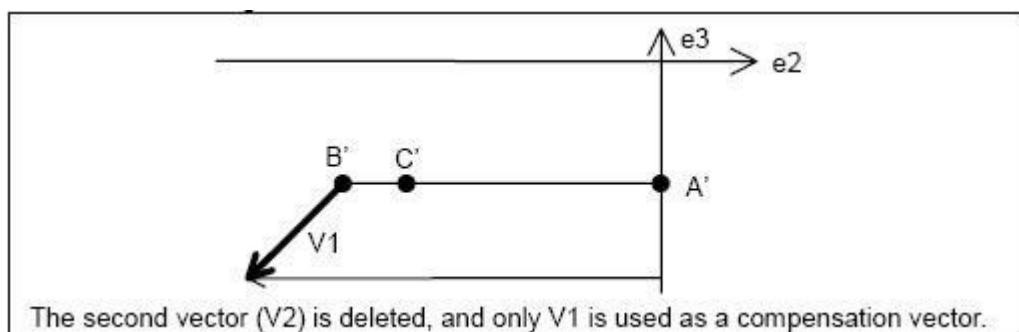


Figure 3.18: Q3 command

Others

When the tool movement changes linear to circular (helical), circular (helical) to linear, or circular (helical) to circular (helical), the start, end, and center points of a circular (helical) movement are projected on the compensation plane that is perpendicular to the tool axis, and a compensation vector is calculated in the plane. The obtained vector is added to the originally specified position to create a position to be specified. Then, the tool is moved linearly or circularly (helically) to the created position.

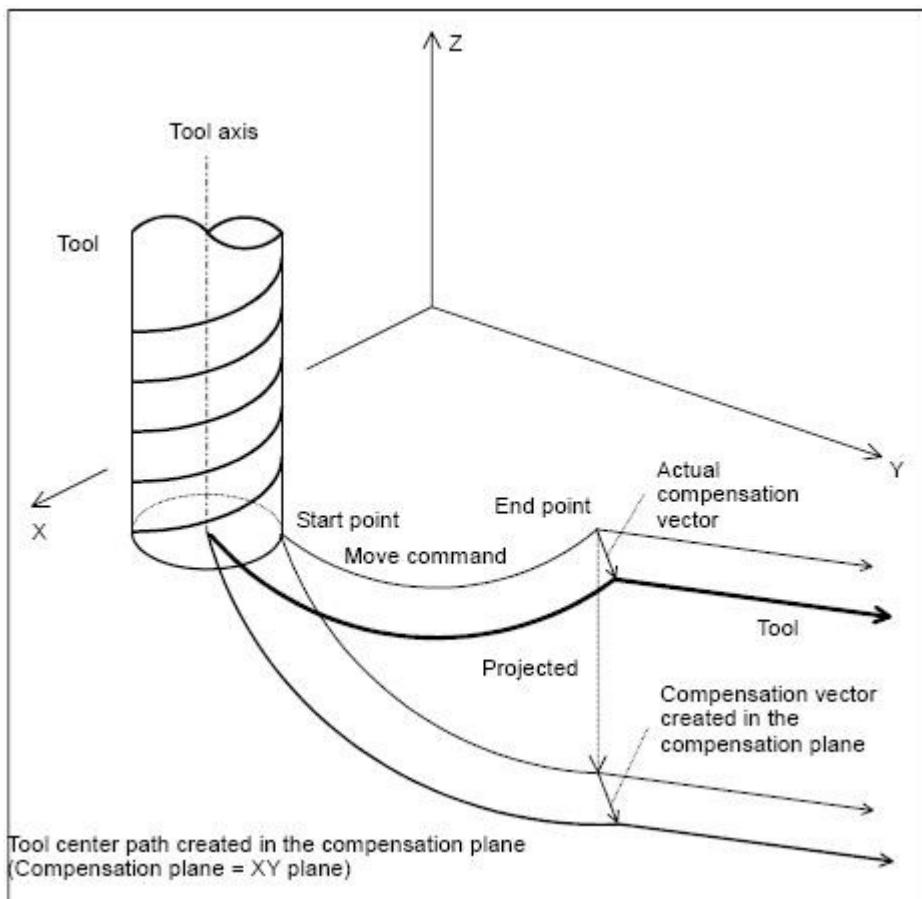


Figure 3.19: Operation during compensation

Angle of the rotary axis for type 2 (when the movement range is not specified)

Refer to chapter chapter II-2.3.17.

Angle of the rotary axis for type 2 (when the movement range is specified)

Refer to chapter chapter II-2.3.17.

3.1.3 Leading edge offset

Leading edge offset is a type of cutter compensation used when a workpiece is machined with the edge of a tool. The tool is automatically shifted by the amount of cutter compensation on the line where a plane formed by a tool vector and tool movement direction meets a plane perpendicular to the tool axis direction.

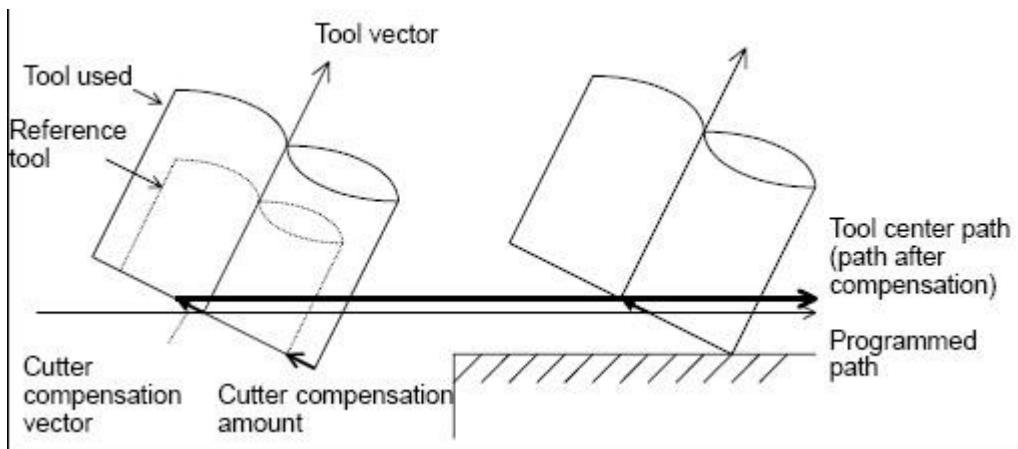


Figure 3.20: Leading edge offset

Programming

G41.3 D_ ;	Leading edge offset
...	
G40 ;	Cancelling leading edge

NOTE

1. G41.3 can be specified only in the G00 and G01 modes. In a block containing G41.3 or G40, only addresses D, O, and N can be specified.
2. The block that follows a block containing a G41.3 command must contain a move command. In the block after G41.3, however, a tool movement in the same direction as the tool axis direction or the opposite direction cannot be specified.
3. No modal G code that belongs to the same group as G00 and G01 can be specified in the G41.3 mode. If such a modal G code is specified, alarm PS5460 is issued.
4. Leading edge offset does not provide type 2 commands. It does not allow the tool direction to be specified with the I, J, and K commands.

3.1.4 Explanation (Leading edge offset)

Operation at startup and cancellation

The operation performed at leading edge offset startup and cancellation does not vary. When G41.3 is specified, the tool is moved by the amount of compensation (V_c) in the plane formed by the movement vector (VM) of the block after the G41.3 block and the tool vector (VT) obtained at the time of G41.3 specification. The tool movement is perpendicular to the tool vector. When G40 is specified, the tool is moved to cancel V_c . The following illustrates how the compensation is performed:

<1> When the tool vector is inclined in the tool movement direction

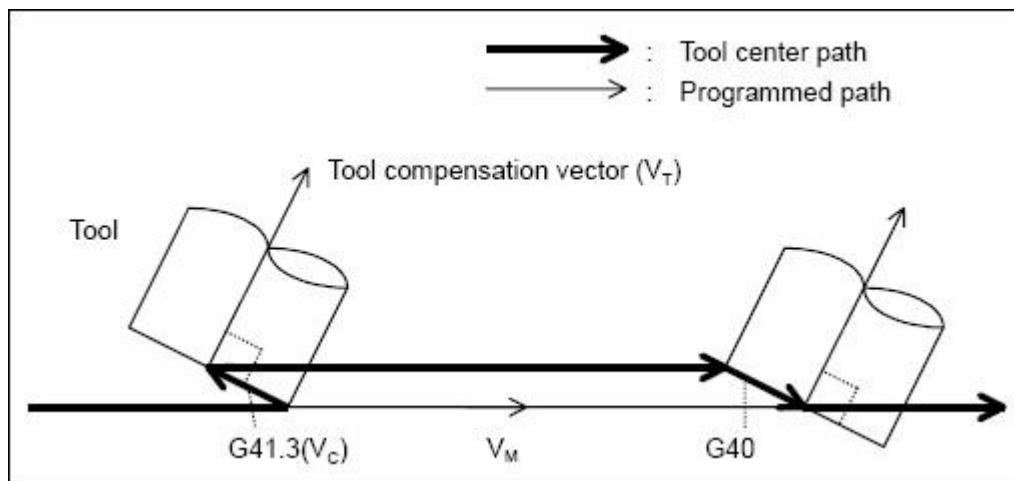


Figure 3.21: When the tool vector is inclined in the tool movement direction

<2> When the tool vector is inclined in the direction opposite to the tool movement direction

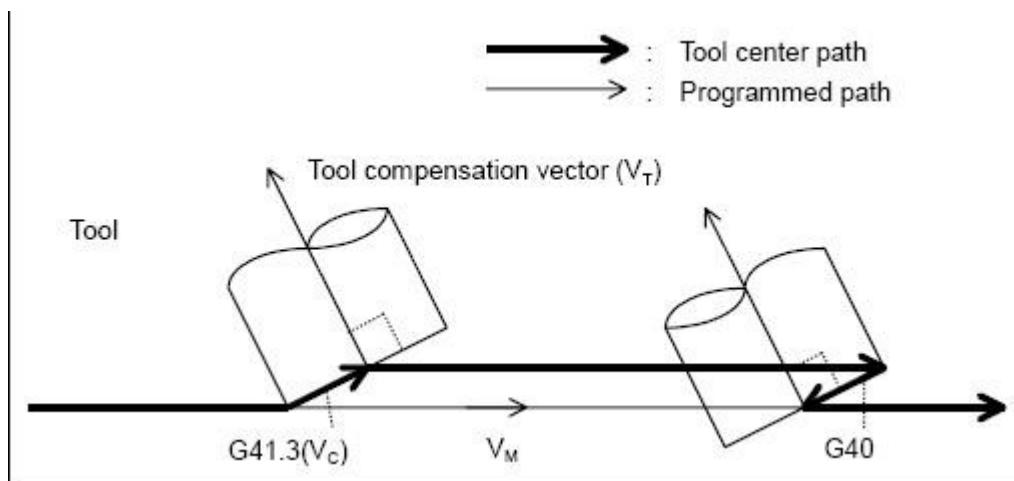


Figure 3.22: When the tool vector is inclined in the direction opposite to the tool movement direction

Operation during compensation

The tool center moves so that a compensation vector (V_C) perpendicular to the tool vector (V_T) is created in the plane formed by the tool vector (V_T) at the end point of each block and the movement vector (V_M) of the next block.

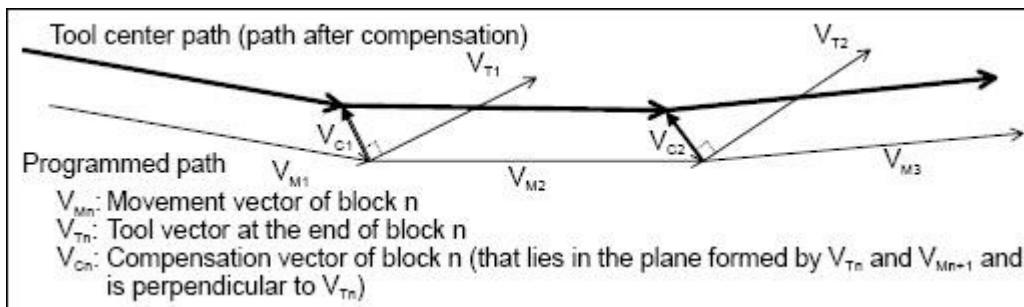


Figure 3.23: Operation during compensation

If a G code or M code that suppresses buffering is specified in the compensation mode, however, the compensation vector created immediately before the specification is maintained.

When a block specifying no movement (including a block containing a move command for a rotary axis only) is specified in the compensation mode, the movement vector of the block after the block specifying no movement is used to create a compensation vector as shown below.

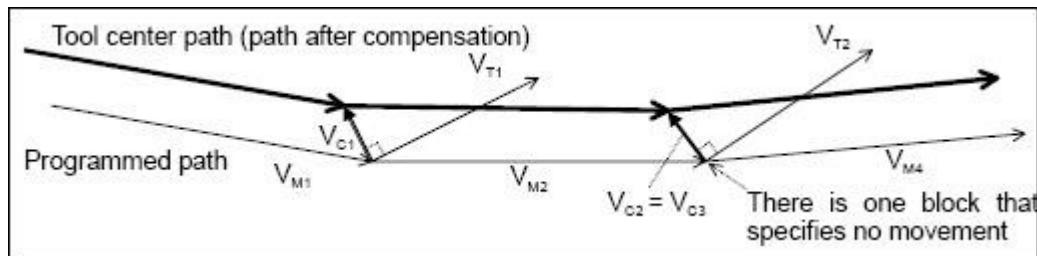


Figure 3.24: When there is one block that specifies no movement

If block 3 specifies no movement, the compensation vector of block 2 (VC2) is created in a plane formed by the movement vector of block 4 (VM4) and the tool vector (VT2) at the end of block 2. VC2 is perpendicular to VT2.

CAUTION

If two or more successive blocks specify no movement, the previously created compensation vector is maintained. However, such specification should be avoided.

Block immediately before the offset cancel command (G40)

In the block immediately before the offset cancel command (G40), a compensation vector is created from the movement vector of that block and the tool vector at the end point of the block as shown below.

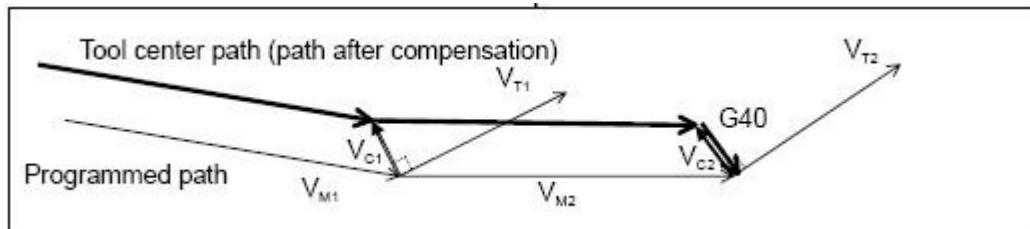


Figure 3.25: Block immediately before G40

The compensation vector (VC2) of block 2 is created in a plane formed by the tool vector (VT2) at the end point of block 2 and the movement vector (VM2) of block 2. VC2 is perpendicular to VT2.

Compensation performed when θ is approximately 0° , 90° , or 180°

When the included angle θ between VMn+1 and VTn is regarded as 0° , 180° , or 90° , the compensation vector is created in a different way. So, when creating a program, note the following points:

1. Setting a variation range for determining θ to be 0° , 180° , or 90°

When the included angle (θ) between the tool vector (VTn) and movement vector (VMn+1) becomes approximately 0° , 180° , or 90° , the system regards θ as 0° , 180° , or 90° , respectively, then creates a compensation vector which is different from the normal compensation vector. The variation range used for determining θ to be 0° , 180° , and 90° is set in parameter No. 19631. For example, let the angle set in this parameter be $\Delta\theta$. Then, the system regards θ as follows:

1. If $0 \leq \theta \leq \Delta\theta$, θ is regarded as 0° .

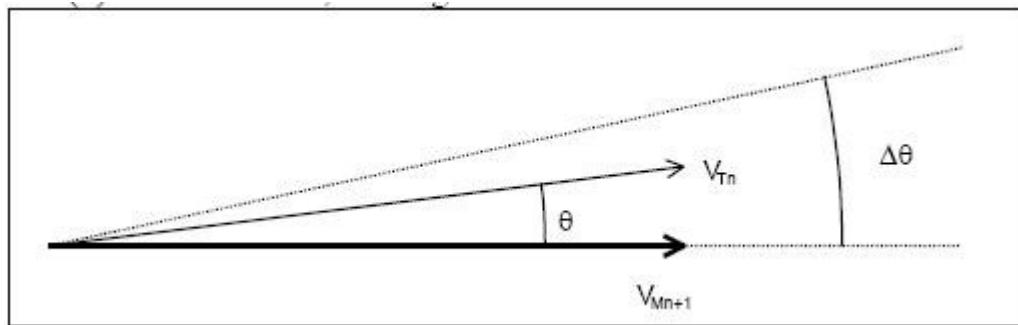


Figure 3.26: Determination of $\theta = 0^\circ$

2. If $(180 - \Delta\theta) \leq \theta \leq 180$, θ is regarded as 180° .

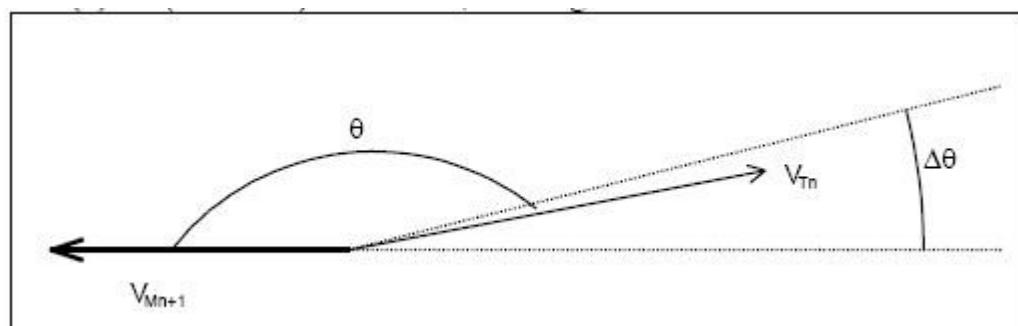


Figure 3.27: Determination of $\theta = 180^\circ$

3. If $(90 - \Delta\theta) \leq \theta \leq (90 + \Delta\theta)$, θ is regarded as 90° .

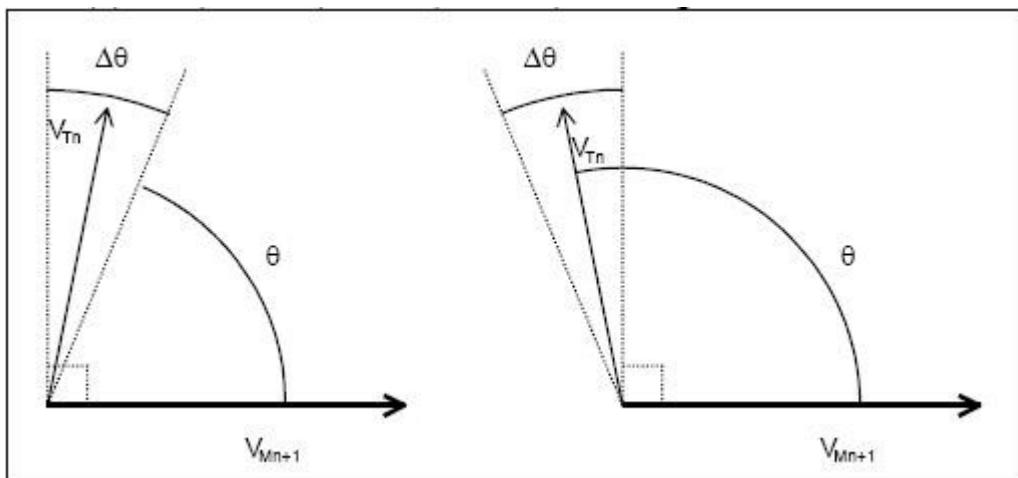


Figure 3.28: Determination of $\theta = 90^\circ$

- 4.

2. Compensation vector when θ is regarded as 0° or 180°

At startup (when G41.3 is specified), alarm PS5408 is issued. This means that the tool vector of a block and the movement vector of the next block must not point in the same direction or in opposite directions at startup.

At other than startup, the previously created compensation vector is maintained without change. If the included angles between VT2 and VM3, VT3 and VM4, and VT4 and VM5 are regarded as 0° , compensation vector VC1 of block 1 is maintained as compensation vectors VC2, VC3, and VC4 of blocks 2, 3, and 4, respectively.

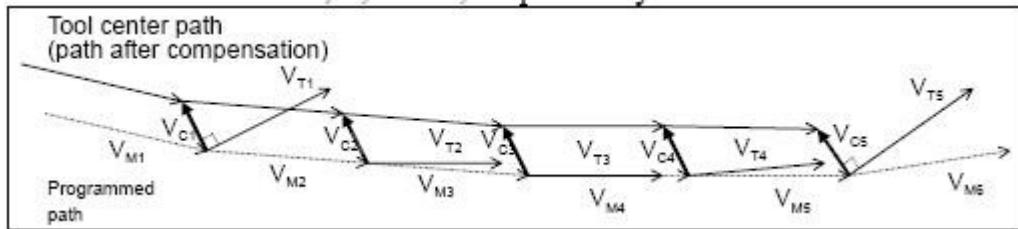


Figure 3.29: When $\theta = 0^\circ$ is determined

If the included angles between V_{T2} and V_{M3} , V_{T3} and V_{M4} , and V_{T4} and V_{M5} are regarded as 180° , compensation vector V_{C1} of block 1 is maintained as compensation vectors V_{C2} , V_{C3} , and V_{C4} of blocks 2, 3, and 4, respectively.

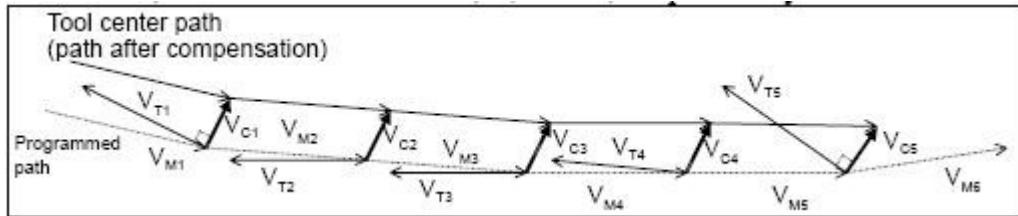


Figure 3.30: When $\theta = 180^\circ$ is determined

3. Compensation vector when θ is regarded as 90°

If the previous compensation vector (V_{Cn-1}) points in the opposite direction ($(V_{Mn} \times V_{Tn-1}) \times V_{Tn-1}$ direction) to V_{Mn} with respect to V_{Tn-1} , the current compensation vector (V_{Cn}) is created so that it also points in the $(V_{Mn+1} \times V_{Tn}) \times V_{Tn}$ direction.

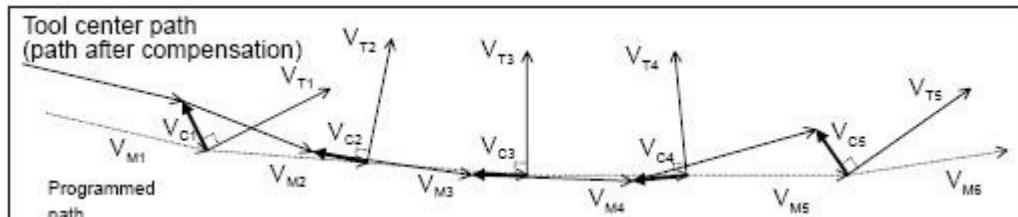


Figure 3.31: When $\theta = 90^\circ$ is determined 1

If the previous compensation vector (V_{Cn-1}) points in the same direction ($-(V_{Mn} \cdot V_{Tn-1}) \cdot V_{Tn-1}$ direction) as V_{Mn} with respect to V_{Tn-1} , the current compensation vector (V_{Cn}) is created so that it also points in the $-(V_{Mn+1} \cdot V_{Tn}) \cdot V_{Tn}$ direction.

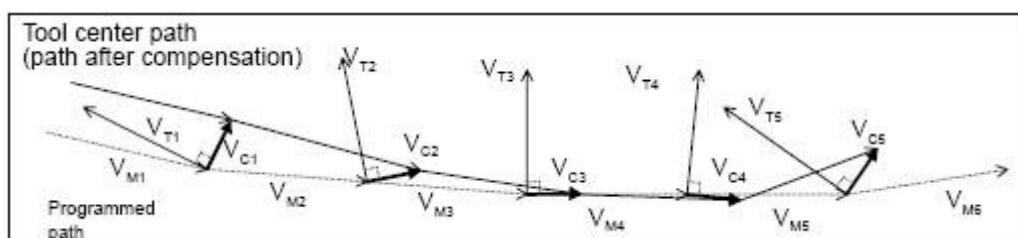


Figure 3.32: When $\theta = 90^\circ$ is determined 2

3.1.5 Tool tip position (cutting point) command

For machines having a rotary axis for rotating a tool, this function performs cutter compensation for 5-axis machining at the tool tip position if a programmed point is specified with a pivot point.

When this function is used, the programmed point (pivot point) is converted into a tool tip position (cutting point) and a vector of cutter compensation for 5-axis machining is calculated for the position obtained by the conversion. Then, the programmed point (pivot point) is compensated for with the vector of cutter compensation for 5-axis machining.

If the tool side offset (G41.2/G42.2) of cutter compensation for 5-axis machining is performed, the operation of this function is as follows:

- (1) If parameter No. **19632** is 0
The vector of cutter compensation for 5-axis machining is calculated at the programmed point (pivot point).
- (2) If parameter No. **19632** is not 0 (this function)
The vector of cutter compensation for 5-axis machining is calculated at the tool tip position (cutting point).

Operation explanation

This function calculates a vector at the tool tip position for the cutter compensation function for 5-axis machining as described below.

- (1) Convert the programmed coordinates from a programmed point (pivot point) to a tool tip position (cutting point). Parameter No. **19632** is used to store the distance from the programmed point (pivot point) to the tool tip position (cutting point).
- (2) Calculate a vector of cutter compensation for 5-axis machining at the tool tip position (cutting point).
- (3) Add the cutter compensation vector to the programmed point (pivot point).

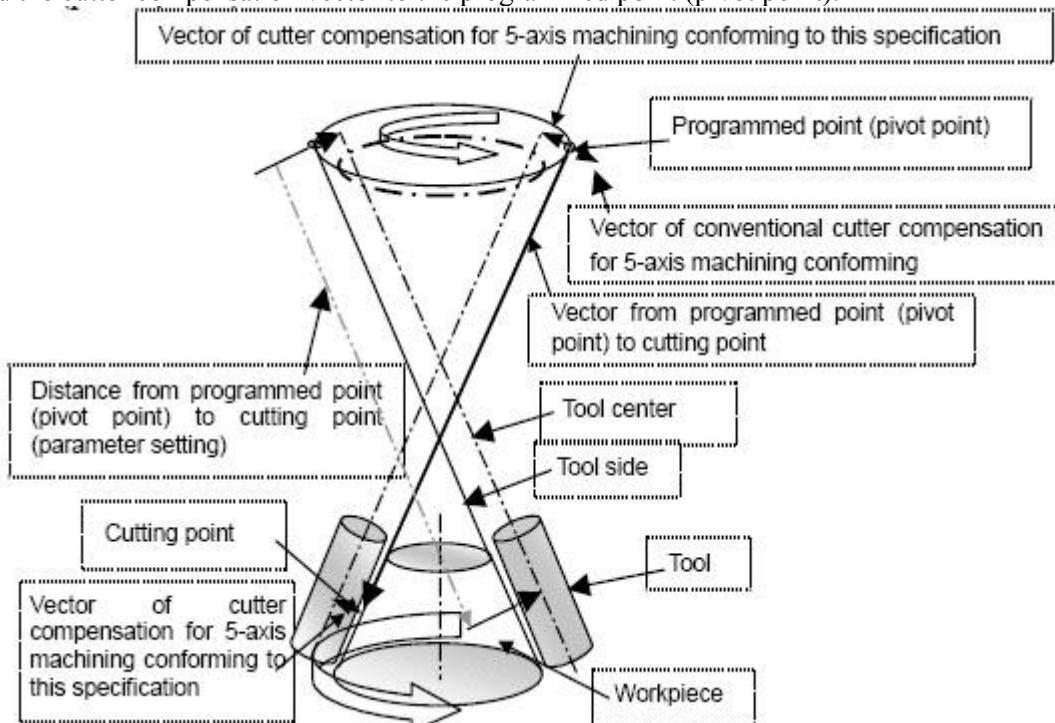


Figure 3.33: Basic operation (for G42.2)

Operation example

For a machine configuration in which the tool axis direction is along the Z-axis and the rotary axes are the B and C axes (Figure 3.34)

LC: Parameter (No. **19632**) specifying the distance from the programmed point (pivot point) to the tool tip position (cutting point)

b: Specified B-axis value, c: Specified C-axis value

$Q = (Q_x, Q_y, Q_z)$: Programmed point (pivot point)

P, R: Programmed points (pivot points) in the preceding and succeeding blocks

QT = (QT_x, QT_y, QT_z): Tool position (tool tip position (cutting point)) resulting from conversion

PT, RT: Tool positions (tool tip positions (cutting positions)) in the preceding and succeeding blocks

Then,

- (1) Convert programmed points (pivot points) P, Q, and R to tool tip positions (cutting points) PT, QT, and RT.

$$QT_x = LC \cdot \sin(b) \cdot \cos(c) + Q_x$$

$$QT_y = LC \cdot \sin(b) \cdot \sin(c) + Q_y$$

$$QT_z = LC \cdot \cos(b) + Q_z$$
 (The same applies to PT and RT.)
- (2) Calculate vector VD of cutter compensation for 5-axis machining from tool tip positions (cutting points) PT, QT, and RT and tool gradient VT.
- (3) Add cutter vector VD to programmed point (pivot point) Q and set the result as the end point position.

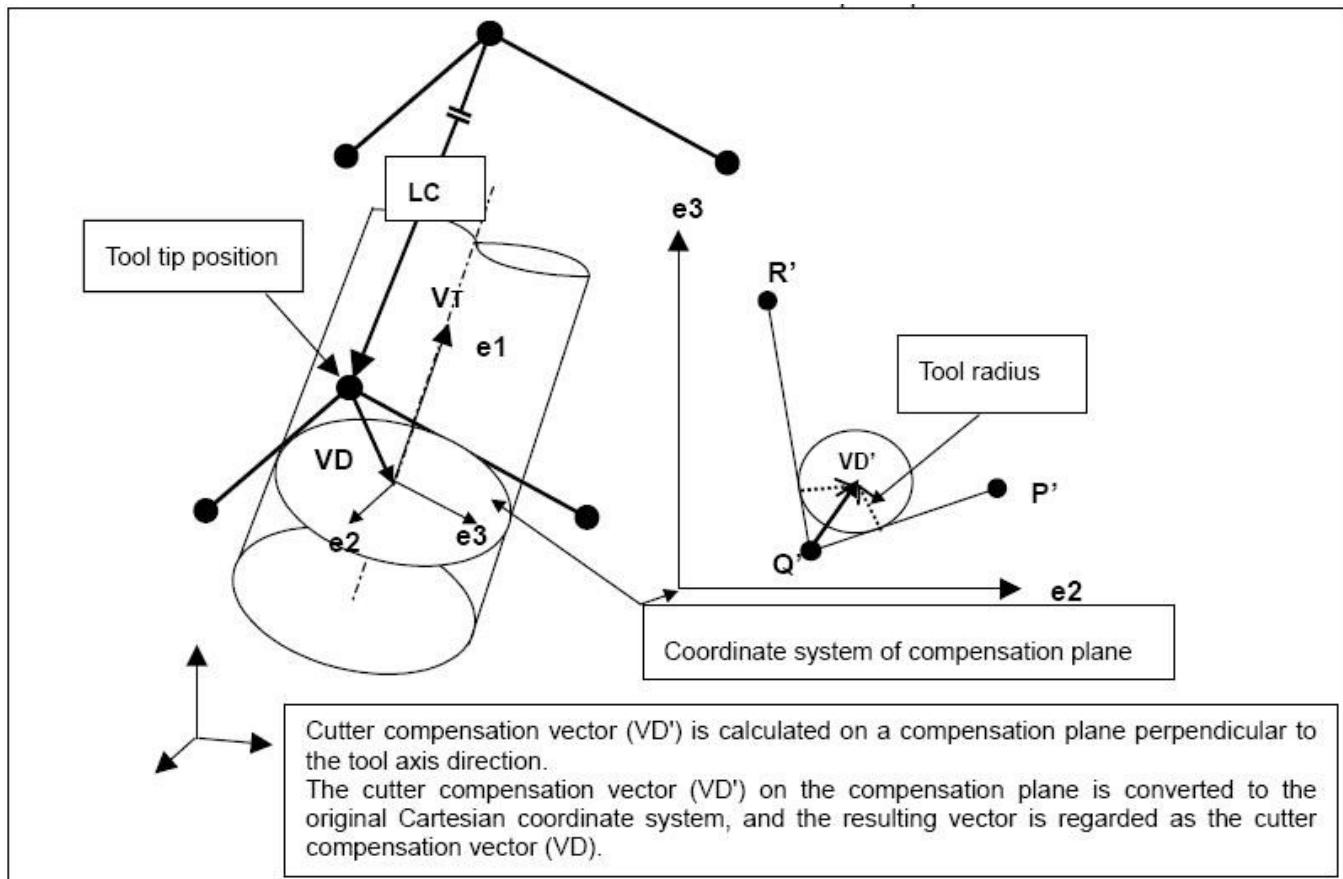


Figure 3.34: Operation

CAUTION

1. This function is disabled for leading edge offset.
2. With a command for a rotary axis only, this function does not calculate a cutter compensation vector.
3. This function cannot be used in the three-dimensional coordinate conversion mode. 4 In addition to
4. the cautions given here, the cautions on the cutter compensation function for 5-axis machining apply to this function.

3.2 Cutter Compensation in Table Rotation Type Machine

Cutter compensation can be performed for a 5-axis machine having a rotary table as shown in Fig. 21.4.2 (a).

Shown below is a 5-axis machine that has table rotation axis A on the X-axis and table rotation axis B on the Y-axis.

This machine configuration is used as a sample configuration in the following explanation unless otherwise noted:

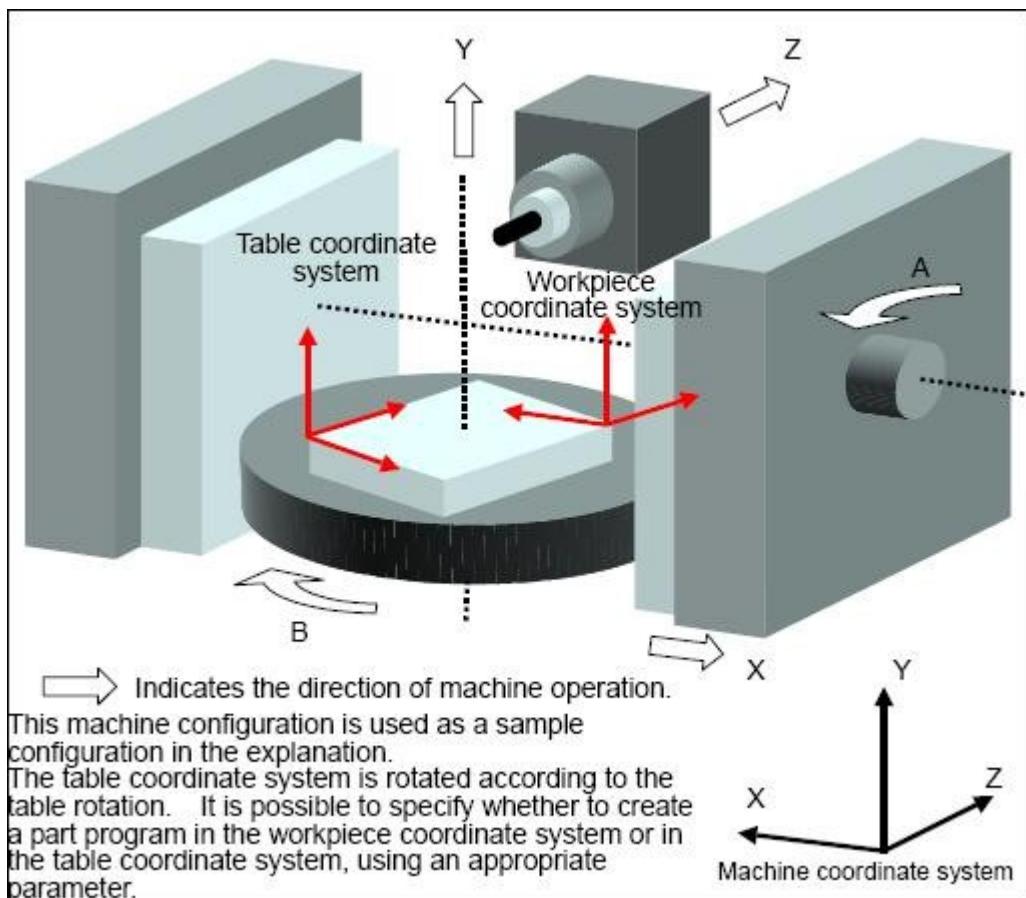


Figure 3.35: Machine having a rotary table

3.2.1 Programming

Startup of cutter compensation type 1

When bit 1 (SPG) of parameter No. **19607** is 0

G41.2 IP_D_	Starts Cutter compensation left (group 07)
G42.2 IP_D_	Cutter compensation right (group 07)

IP_ Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)

D_ Code specifying the cutter compensation amount (1 to 3 digits)

When bit 1 (SPG) of parameter No. **19607** is 1

G41.4 IP_D_	Starts Cutter compensation left (group 07)
G42.4 IP_D_	Cutter compensation right (group 07)

IP_ Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)

D_ Code specifying the cutter compensation amount (1 to 3 digits)

NOTE

1. In a table rotation type machine (parameter No. **19680** = 12), if an attempt is made to issue G41.4 or G42.4 with SPG, bit 1 of parameter No. **19607**, equal to 0, alarm PS0010 is generated.
2. In a table rotation type machine, if an attempt is made to issue G41.2 or G42.2 with SPG, bit 1 of parameter No. **19607**, equal to 1, alarm PS5460 is generated.
3. In machine not of the table rotation type machine, if an attempt is made to issue G41.4 or G42.4 with SPG, bit 1 of parameter No. **19607**, equal to 1, alarm PS5460 is generated.

Startup of cutter compensation type 2

G41.6 IP_ D_	Starts Cutter compensation left (group 07)
G42.6 IP_ D_	Cutter compensation right (group 07)

IP_ Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)

D_ Code specifying the cutter compensation amount (1 to 3 digits)

Q_ Tool's angle of gradient (in degrees)

I_J_K_ Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (workpiece coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

In a tool rotation type machine, I, J, and K can be specified in a G41.6/G42.6 command block; in a table rotation type machine, however, they cannot. If an attempt is made to specify them, alarm PS5460 is generated.

The following are the notes on type 2.

NOTE

1. If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2. In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3. If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4. If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. **1260** (mount of travel per rotation about the rotation axis).
5. They can be used only with the settings that select the table coordinate system as a programming coordinate system (WKP, bit 5 of parameter No. **19696**, = 0 and TBP, bit 4 of parameter No. **19746**, = 1). If an attempt is made to issue G41.6/G42.6 with the settings that select the workpiece coordinate system as a programming coordinate system, alarm PS5460 is generated.

Cancelling cutter compensation

G40 IP_ ;	Cancellation of cutter compensation (group 07)
------------------	--

IP_ Value specified for axis movement (including rotary axis)

Selecting an offset plane

When parameter PTC (No. **19746**) is 1, compensation is performed on the selected plane, on the assumption that the tool is pointing to the direction perpendicular to that plane.

Offset plane	Plane selection command	IP
XpYp	G17 ;	Xp Yp
ZpXp	G18 ;	Xp Zp
YpZp	G19 ;	Yp Zp

(Example: In the case of Fig. 21.4.2 (a), the XpYp plane is selected.)

The two axes of a selected plane must be included in the three basic axes (the axes for which parameter No. 1022 is set to 1 to 3).

When parameter PTC (No. **19746**) is 0, compensation is performed on the plane perpendicular to the tool direction specified with parameters Nos. **19697**, **19698**, and **19699**, regardless of which plane is selected.

NOTE

This function is enabled for tool side offset only. If leading edge offset is specified, alarm PS5460 is generated.

3.2.2 Explanation

Tool's angle of gradient in type 2

For type 2 of cutter compensation for 5-axis machining, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Figure 3.36.)

Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command. If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

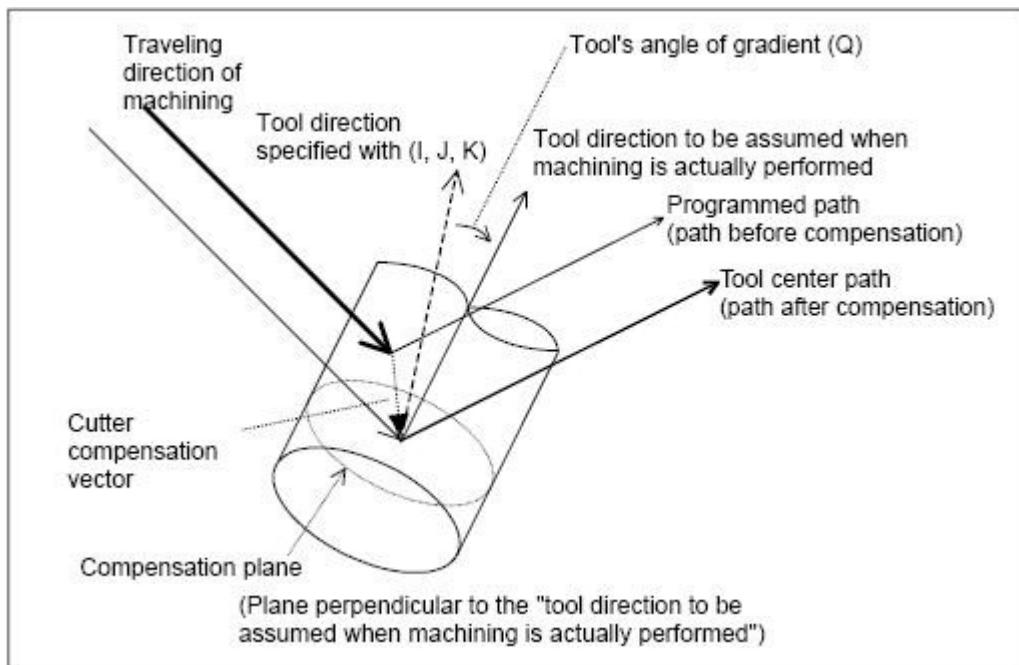


Figure 3.36: Tool's angle of gradient in type 2

Cutter compensation

The cutter compensation function in table rotation type machines basically performs operations in conformance with cutter compensation. The operations different from those of cutter compensation are mainly described below. For the specifications and cautions not mentioned here, see the description of cutter compensation.

Startup

When cutter compensation for the rotary table is specified (G41.2 or G42.2, G41.4 or G42.4, a dimension word other than 0 in the offset plane, or a D code other than D0) in the offset cancel mode, the CNC enters the offset mode. Startup is specified with positioning (G00) or linear interpolation (G01).

NOTE

If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified at startup, alarm PS0034 is issued.

Commands in the offset mode

In the offset mode, compensation is performed for positioning (G00) and linear interpolation (G01).

NOTE

If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified in the mode for cutter compensation for 5-axis machining in a table rotation type machine, alarm PS5460 is issued.

Offset mode cancellation

If a block satisfying either of the following conditions is executed in the offset mode, the CNC enters the offset cancel mode:

1. G40 is specified.
2. 0 is specified as a code specifying the cutter compensation amount (D code).

When offset cancellation is to be performed, neither circular (G02 or G03) nor involute (G02.2 or G03.2) command can be specified. If offset cancellation is specified in such a mode, alarm PS0034 is issued.

If selecting the table coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. **19746**, is 1 and WKP, bit 5 of parameter No. **19696**, is 0, specifying cutter compensation for 5-axis machining causes the table coordinate system to be selected as a programming coordinate system. The table coordinate system refers to the workpiece coordinate system fixed to the table when cutter compensation for 5-axis machining.

In the blocks subsequent to the one in which cutter compensation for 5-axis machining is specified, the table coordinate system rotates with the rotation of the table.

A linear axis (X, Y, Z) command is assumed to be issued on the table coordinate system.

Specifying linear interpolation causes cutter compensation to be performed on the linear interpolation command on the table coordinate system. The table coordinate system does not rotate with the rotation of the tool head.

The cancel (G40) block becomes a block that suppresses buffering. It is possible to switch between absolute and relative coordinate displays, using DET, bit 2 of parameter No. 19608, as follows:

- When DET is 0, the position in the table coordinate system is displayed.
- When DET is 1, the position in the workpiece coordinate system is displayed.

Note, however, the distance to go is always that in the programming coordinate system.

NOTE

(1) Either the AI contour control I or AI contour II control option is required. In addition, be sure to specify the following parameters:

- (1) LRP, bit 1 of parameter No. **1401**, = 1: Linear rapid traverse
- (2) FRP, bit 5 of parameter No. 19501, = 1: Acceleration/deceleration before interpolation is used for rapid traverse.
- (3) Parameter No. **1671**: Acceleration of acceleration/deceleration before interpolation for rapid traverse
- (4) Parameter No. **1672**: Change time for bell-shaped acceleration/deceleration before interpolation for rapid traverse
- (5) Parameter No. **1660**: Maximum permissible acceleration for acceleration/deceleration before interpolation

If they are not specified, alarm PS5483 is generated.

- (2) When table rotation axis movement is specified in the start block of cutter compensation for 5-axis machining, after the movement is completed, the workpiece coordinate system is fixed to the table and assumed to be a table coordinate system.
- (3) In the mode of cutter compensation for 5-axis machining, do not change the workpiece coordinate system or change the workpiece offset value. If an attempt is made to specify workpiece coordinate system selection (G54 to G59), alarm PS5460 is generated.

If selecting the workpiece coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. **19746**, is 0 or if TBP, bit of parameter No. **19746**, is 1 and WKP, bit 5 of parameter No. **19696**, is 1, the programming coordinate system does not rotate with the rotation of the table, being fixed to the workpiece coordinate system.

Angle of the rotation axis in type 2

For an explanation of how the rotation axis end point is determined when the tool direction is specified with an IJKQ command in type 2, see the explanations of cutter compensation in a tool rotation type machine, "Angle of the rotation axis in type 2 (if the operating range is not specified)" and "Angle of the rotation axis in type 2 (if the operating range is specified)".

3.3 Cutter Compensation in Mixed-Type Machine

This function can perform three-dimensional cutter compensation in a 5-axis machine having a rotary table and a tool axis as shown in Figure 3.37.

Shown below is a 5-axis machine that has tool axis A on the X-axis (the tool axis direction is along the Z-axis) and table rotation axis B on the Y-axis.

This machine configuration is used as a sample configuration in the following explanation unless otherwise noted:

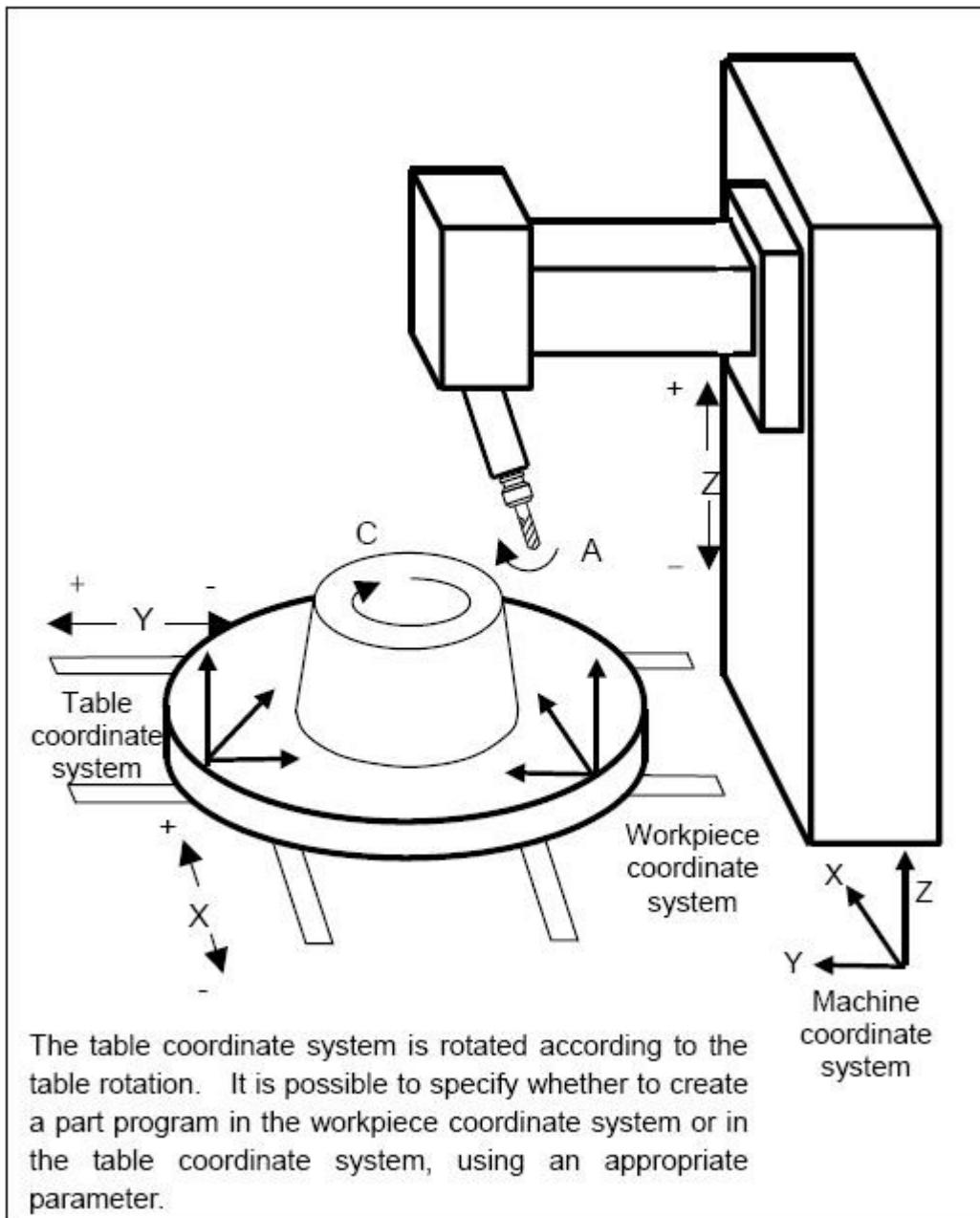


Figure 3.37: Machine having a tool rotation axis and table rotation axis

3.3.1 Programming

Startup of cutter compensation type 1

When bit 1 (SPG) of parameter No. **19607** is 0

G41.2 IP_D_	Starts Cutter compensation left (group 07)
G42.2 IP_D_	Cutter compensation right (group 07)

IP_ Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)

D_ Code specifying the cutter compensation amount (1 to 3 digits)

When bit 1 (SPG) of parameter No. **19607** is 1

G41.4 IP_D_	Starts Cutter compensation left (group 07)
--------------------	--

G42.4 IP_ D_

Cutter compensation right (group 07)

IP_	Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_	Code specifying the cutter compensation amount (1 to 3 digits)

NOTE

1. In a mixed-type machine (parameter No. **19680** = 21), if an attempt is made to issue G41.5 or G42.5 with SPG, bit 1 of parameter No. **19607**, equal to 0, alarm PS0010 is generated.
2. In a mixed-type machine, if an attempt is made to issue G41.2 or G42.2 with SPG, bit 1 of parameter No. **19607**, equal to 1, alarm PS5460 is generated.
3. In machine not of the mixed-type machine, if an attempt is made to issue G41.5 or G42.5 with SPG, bit 1 of parameter No. **19607**, equal to 1, alarm PS5460 is generated.

Startup of cutter compensation type 2

G41.6 IP_ D_	Starts Cutter compensation left (group 07)
G42.6 IP_ D_	Cutter compensation right (group 07)
IP_	Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_	Code specifying the cutter compensation amount (1 to 3 digits)
Q_	Tool's angle of gradient (in degrees)
I_J_K_	Tool axis direction at the block end point as viewed from the programming coordinate system

In a mixed-type machine, I, J, and K can be specified in a G41.6/G42.6 command block; in a table rotation type machine, however, they cannot. If an attempt is made to specify them, alarm PS5460 is generated.

The following are the notes on type 2.

NOTE

1. If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2. In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3. If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4. If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. **1260** (mount of travel per rotation about the rotation axis). They can be used only with the settings that select the table coordinate system as a programming coordinate system (WKP, bit 5 of parameter No. **19696**, = 0 and TBP, bit 4 of parameter No. **19746**, = 1). If an attempt is made to issue G41.6/G42.6 with the settings that select the workpiece coordinate system as a programming coordinate system, alarm PS5460 is generated.

Canceling cutter compensation

G40 IP_ ;	Cancellation of cutter compensation (group 07)
IP_	Value specified for axis movement (including rotary axis)

NOTE

This function is enabled for tool side offset only. If leading edge offset is specified, alarm PS5460 is generated.

3.3.2 Explanation

Tool's angle of gradient in type 2

For type 2 of cutter compensation for 5-axis machining, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Figure 3.38.)

Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command. If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

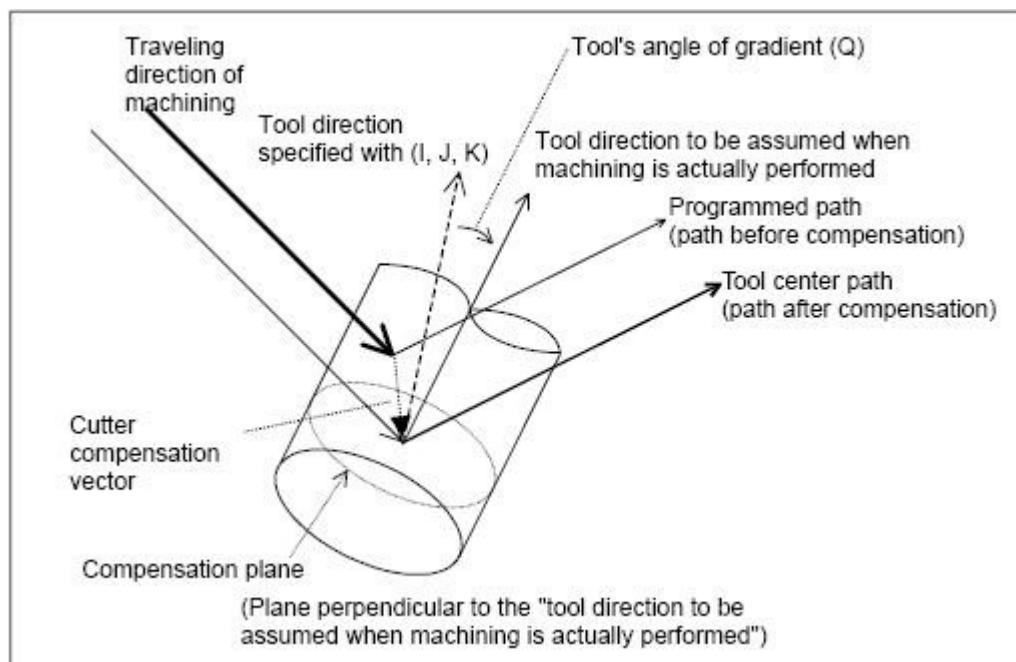


Figure 3.38: Tool's angle of gradient in type 2

Cutter compensation

The cutter compensation function in mixed type machines basically performs operations in conformance with cutter compensation. The operations different from those of cutter compensation are mainly described below. For the specifications and cautions not mentioned here, see the description of cutter compensation.

Startup

When cutter compensation for the rotary table is specified (G41.2 or G42.2, G41.4 or G42.4, a dimension word other than 0 in the offset plane, or a D code other than D0) in the offset cancel mode, the CNC enters the offset mode. Startup is specified with positioning (G00) or linear interpolation (G01).

NOTE

If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified at startup, alarm PS0034 is issued.

Commands in the offset mode

In the offset mode, compensation is performed for positioning (G00) and linear interpolation (G01).

NOTE

If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified in the mode for cutter compensation for 5-axis machining in a table rotation type machine, alarm PS5460 is issued.

Offset mode cancellation

If a block satisfying either of the following conditions is executed in the offset mode, the CNC enters the offset cancel mode:

3. G40 is specified.
4. 0 is specified as a code specifying the cutter compensation amount (D code).

When offset cancellation is to be performed, neither circular (G02 or G03) nor involute (G02.2 or G03.2) command can be specified. If offset cancellation is specified in such a mode, alarm PS0034 is issued.

NOTE

1. This function cannot be used in the three-dimensional coordinate conversion mode.
2. In addition to the cautions given here, the cautions on the cutter compensation function for 5-axis machining apply to this function.

If selecting the table coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. **19746**, is 1 and WKP, bit 5 of parameter No. **19696**, is 0, specifying cutter compensation for 5-axis machining causes the table coordinate system to be selected as a programming coordinate system. The table coordinate system refers to the workpiece coordinate system fixed to the table when cutter compensation for 5-axis machining.

In the blocks subsequent to the one in which cutter compensation for 5-axis machining is specified, the table coordinate system rotates with the rotation of the table.

A linear axis (X, Y, Z) command is assumed to be issued on the table coordinate system.

Specifying linear interpolation causes cutter compensation to be performed on the linear interpolation command on the table coordinate system. The table coordinate system does not rotate with the rotation of the tool head.

The cancel (G40) block becomes a block that suppresses buffering. It is possible to switch between absolute and relative coordinate displays, using DET, bit 2 of parameter No. 19608, as follows:

- When DET is 0, the position in the table coordinate system is displayed.
- When DET is 1, the position in the workpiece coordinate system is displayed.

Note, however, the distance to go is always that in the programming coordinate system.

NOTE

- (1) Either the AI contour control I or AI contour II control option is required. In addition, be sure to specify the following parameters:
 1. LRP, bit 1 of parameter No. **1401**, = 1: Linear rapid traverse
 2. FRP, bit 5 of parameter No. 19501, = 1: Acceleration/deceleration before interpolation is used for rapid traverse.
 3. Parameter No. **1671**: Acceleration of acceleration/deceleration before interpolation for rapid traverse
 4. Parameter No. **1672**: Change time for bell-shaped acceleration/deceleration before interpolation for rapid traverse
 5. Parameter No. **1660**: Maximum permissible acceleration for acceleration/deceleration before interpolation

If they are not specified, alarm PS5483 is generated.

- (2) When table rotation axis movement is specified in the start block of cutter compensation for 5-axis machining, after the movement is completed, the workpiece coordinate system is fixed to the table and assumed to be a table coordinate system.
- (3) In the mode of cutter compensation for 5-axis machining, do not change the workpiece coordinate system or change the workpiece offset value. If an attempt is made to specify workpiece coordinate system selection (G54 to G59), alarm PS5460 is generated.

If selecting the workpiece coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. **19746**, is 0 or if TBP, bit of parameter No. **19746**, is 1 and WKP, bit 5 of parameter No. **19696**, is 1, the programming coordinate system does not rotate with the rotation of the table, being fixed to the workpiece coordinate system.

Angle of the rotation axis in type 2

For an explanation of how the rotation axis end point is determined when the tool direction is specified with an IJKQ command in type 2, see the explanations of cutter compensation in a tool rotation type machine, "Angle of the rotation axis in type 2 (if the operating range is not specified)" and "Angle of the rotation axis in type 2 (if the operating range is specified)".

3.4 Interference check and interference avoidance

By setting NI5, bit 1 of parameter No. **19608**, to 1, this function performs an interference check on the plane (compensation plane) perpendicular to the tool axis direction regardless of the machine configuration.

If CAV, bit 5 of parameter No. **19607**, is set to 1, a vector is generated to avoid interference on the same plane.

3.4.1 Explanation

For a tool rotation type machine

An interference check is performed, as well as interference avoidance, with the tool path as projected from the workpiece coordinate system (X-Y-Z) onto the compensation plane (X'-Y'-Z') and a compensation vector.

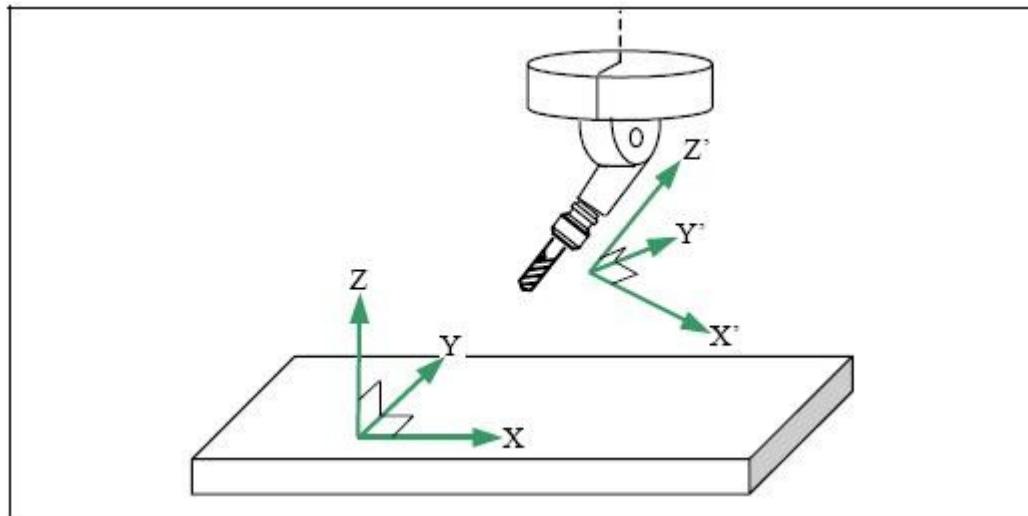


Figure 3.39: Tool rotation type

For a table rotation type machine

An interference check is performed, as well as interference avoidance, with the tool path as converted from the workpiece coordinate system (X-Y-Z) into the table coordinate system (X'-Y'-Z') and a compensation vector.

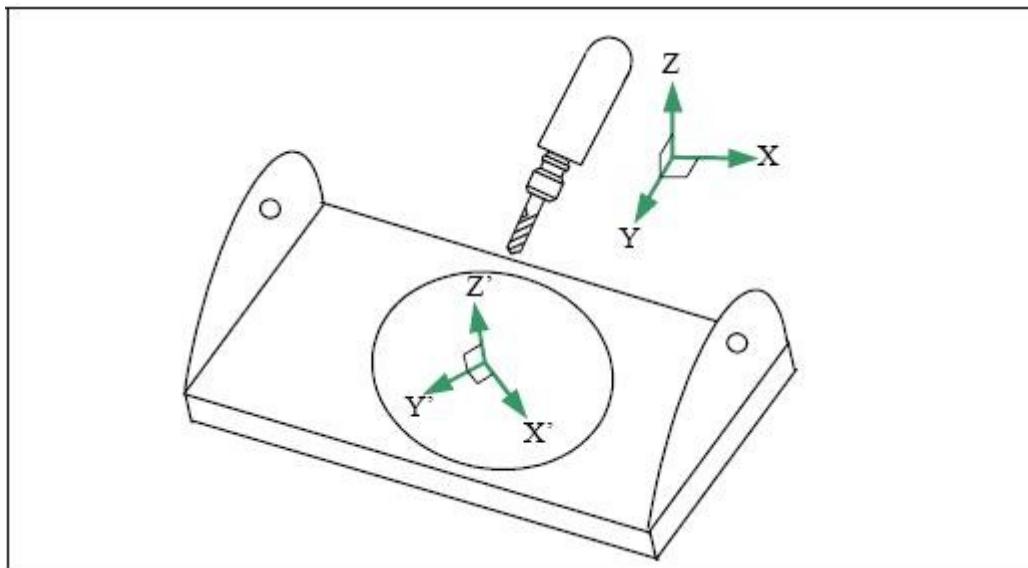


Figure 3.40: Table rotation type

For a mixed-type machine

An interference check is performed, as well as interference avoidance, with the tool path as projected from the workpiece coordinate system (X-Y-Z) onto the table coordinate system (X'-Y'-Z') and then onto the compensation plane perpendicular to the tool axis direction (X''-Y''-Z'') and a compensation vector.

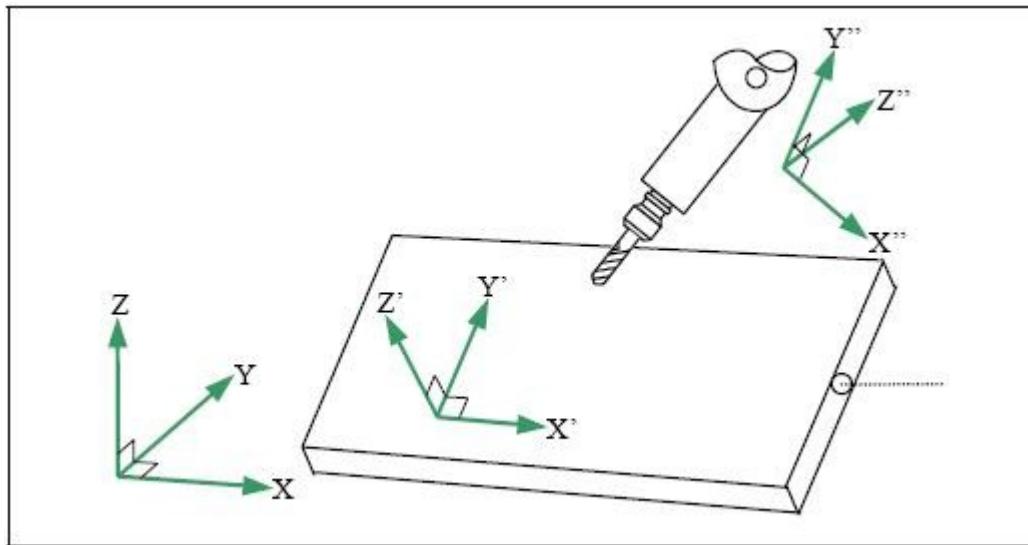


Figure 3.41: Mixed-type

Interference avoidance

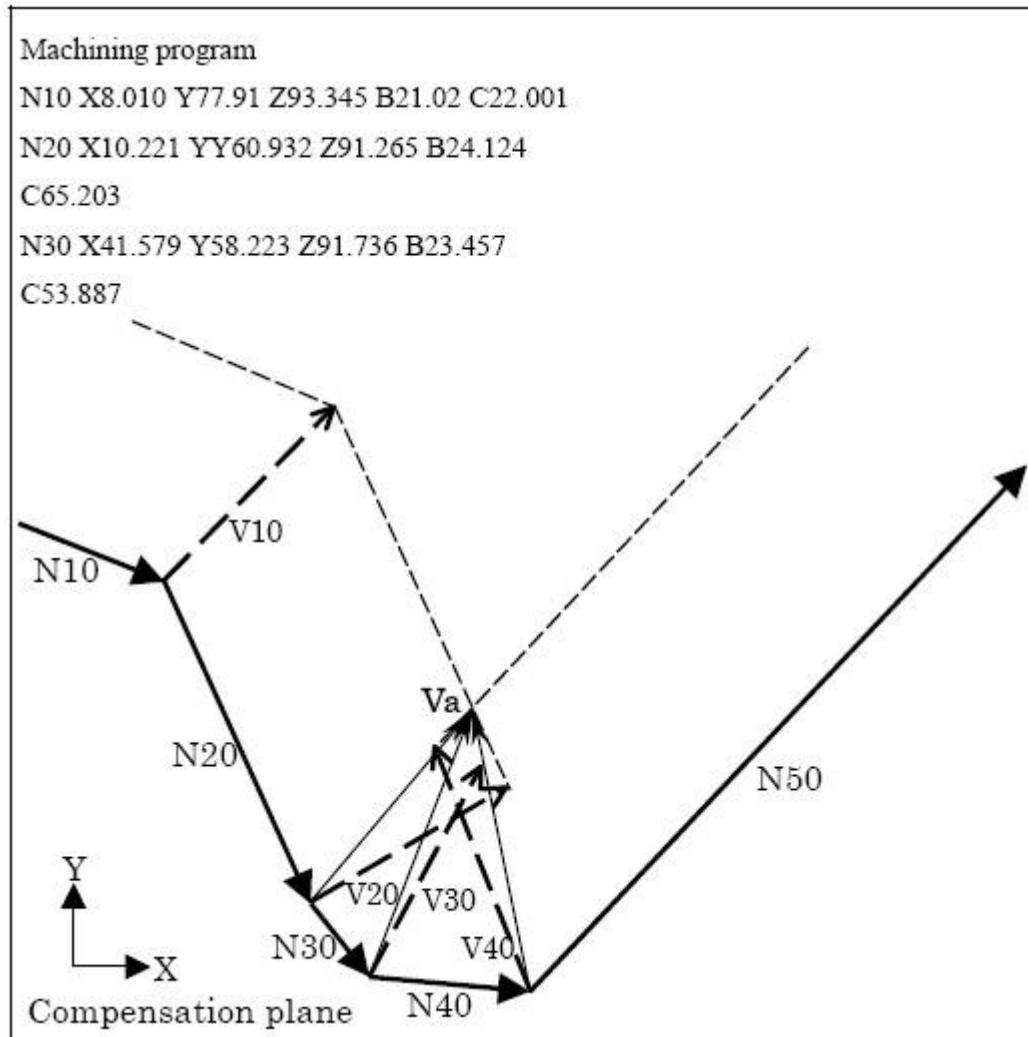


Figure 3.42: Example of interference avoidance

Figure 3.42 shows a tool path in the workpiece coordinate system as projected onto the compensation plane. For interference avoidance, calculation is performed with the tool path resulting from looking at up to four blocks

ahead. At the start of the execution of the N10 block, the system looks at N20 to N50 ahead and generates V20 to V40.

Then, because the movement direction of N30 greatly differs from the direction from V20 to V30, V20 and V30 are considered interferences and eliminated. Similarly, because the movement direction from N30 to N40 greatly differs from the movement direction from V20 to V40, V40 is also considered an interference and eliminated.

Then, the interference avoidance vector V_a is generated between N20 and N50 and used instead of V20, V30, and V40. At this time, because N20 and N50 generally do not intersect, the plane perpendicular to the tool axis direction at the N20 end point is regarded as a compensation plane, N20 and N50 are projected onto this plane, and V_a is determined by calculating the intersection point.

NOTE

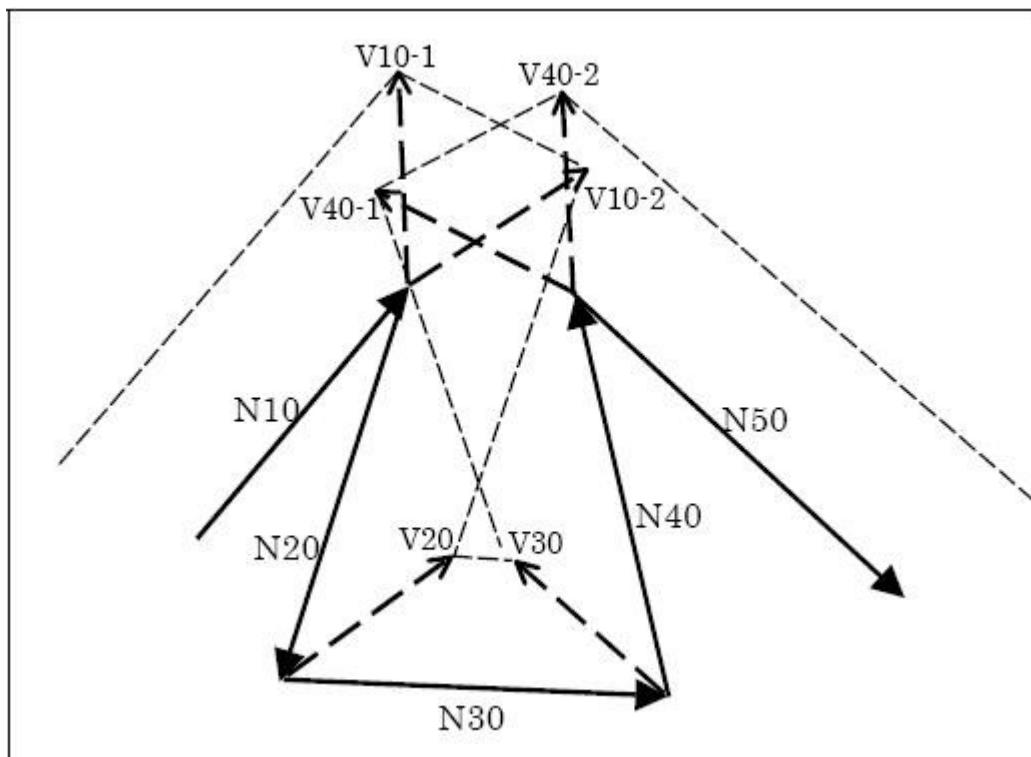
Strictly speaking, if the tool axis direction at the N20 end point differs from the tool axis direction at the N50 start point, correct intersection point calculation is not possible. For this reason, the maximum permissible angle by which the tool axis directions in the two blocks used to determine the interference avoidance vector may differ can be set in parameter No. 19636, and if the change in tool axis direction is within the range, an approximate interference avoidance vector is calculated.

If the maximum angle is exceeded, the compensation vector is determined but interference avoidance is not performed.

If interference avoidance is not possible

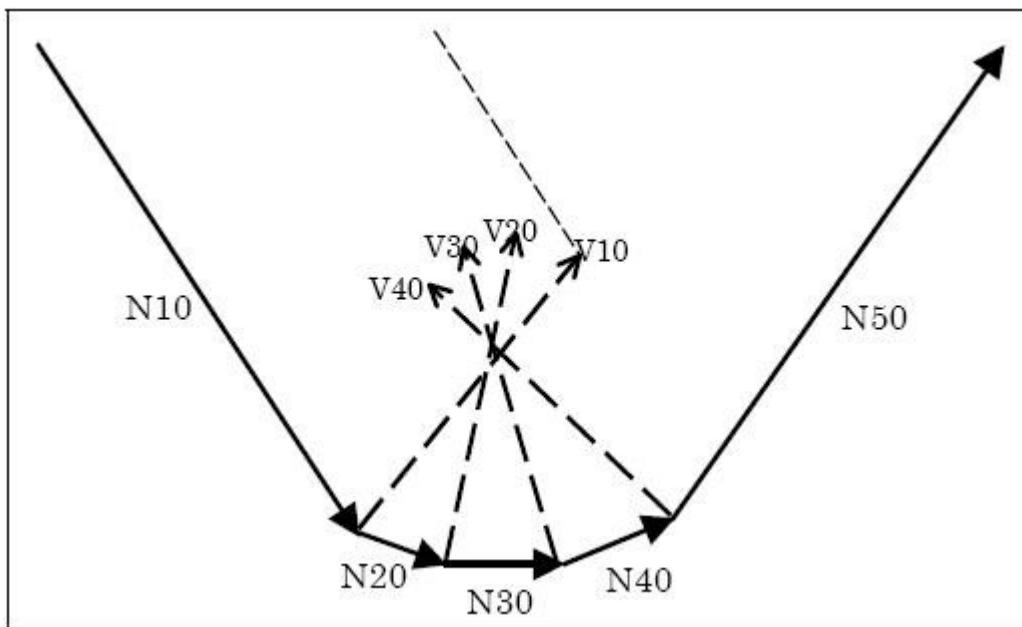
If there are three consecutive interfering blocks, no interference vector can be generated.

Example 1



N20 to N40 interfere, so that no interference avoidance vector can be generated. Too much cutting results.

Example 2



N10 to N40 interfere, so that no interference avoidance vector can be generated. V10 causes an interference alarm.

3.5 Restrictions

3.5.1 Common Restrictions

Interference check

In the mode for cutter compensation for 5-axis machining, interference checks are made using a specified position in the workpiece coordinate system and a compensation vector. The interference check avoidance function cannot be used.

Corner rounding (G39)

In the mode for cutter compensation for 5-axis machining, G39 cannot be specified. Specifying G39 causes an alarm.

Reset

Whenever a reset is made in the mode for cutter compensation for 5-axis machining (G41.2, G42.2, G41.4, G42.4, G41.5, or G42.5), the cancel mode (G40) is entered.

AI contour control I and II

To set the AI contour control I or II mode, the corresponding G code must be specified. Specifying cutter compensation for 5-axis machining does not automatically place the CNC in the AI contour control I or II mode.

Restricted commands

In the mode for cutter compensation for 5-axis machining, the following functions are available, but their statuses cannot be changed:

- Inch/metric input (If an attempt is made to change the status by using G20 or G21, alarm PS5000 is issued.)
- Mirror image (The signal status cannot be changed.)
- One-digit F code feed (The feedrate cannot be changed by using the manual handle.)

Unavailable commands

In the mode for cutter compensation for 5-axis machining, the functions listed below cannot be specified. Specifying any of these functions results in an alarm.

- Hypothetical axis interpolation.....G07
- Circular interpolation.....G07.1
- Polar coordinate interpolation G12.1, G13.1
- Polar coordinate commandG15, G16
- Reference position return checkG27
- Reference position return G28, G29, G30
- Skip.....G31
- Threading.....G33
- Automatic tool length measurementG37
- Normal direction control..... G40.1, G41.1, G42.1
- Cutter or tool nose radius compensation G41, G42, G39
- Three-dimensional cutter compensation.....G41
- Wheel wear compensation.....G41
- Tool offset..... G45, G46, G47, G48
- Programmable mirror image.....G50.1, G51.1
- Local coordinate systemG52
- Machine coordinate systemG53
- Workpiece coordinate system settingG54-G59, G54.1
- Rotary table dynamic fixture offsetG54.2
- Single direction positioning.....G60
- Automatic corner override.....G62
- Tapping mode.....G63
- Three-dimensional coordinate conversion.....G68, G69
- Tilted working plane commandG68.2, G69
- Figure copy G72.1, G72.2
- Canned cycle.....G73-G79, G80, G81-G89, G98, G99
- Electric gear box.....G80, G81
- Hobbling machine functionG80, G81
- External operation functionG81
- ChoppingG81.1
- Small hole peck drilling cycleG83
- Workpiece coordinate system settingG92
- Workpiece coordinate system presettingG92.1
- Feed per revolution.....G95
- Constant surface speed controlG96, G97

Unavailable functions

If the following function is specified in the cutter compensation mode for 5-axis machining, a warning message is issued:

- MDI interruption

If one of the following functions is specified in the cutter compensation mode for 5-axis machining, a PS alarm is

issued:

- Manual interruption operation
- Tool retract and recover

In the cutter compensation mode for 5-axis machining, the following functions cannot be used:

- • Twin table control
- • Flexible synchronous control
- • Sequence number comparison and stop (It is not possible to cause a stop by sequence number in the cutter compensation mode for 5-axis machining.)
- Index table indexing
- Retrace function (Programs using the cutter compensation mode for 5-axis machining must not use the retrace function.)
- Rotary axis control
- Manual handle interruption
- External deceleration (External deceleration is not performed.)
- Angular axis control

Combinations of other NC commands may be restricted. Refer to the manual on each function.

3.5.2 Restrictions on tool rotation type

Unavailable commands (leading edge offset)

In the G41.3 mode, the following commands cannot be specified:

- G functions of group 01 other than G00 and G01

Use with tool center point control

If cutter compensation for 5-axis machining is specified before tool center point control when cutter compensation for 5-axis machining and tool center point control are specified together, tool center point control must be canceled before cutter compensation for 5-axis machining is canceled (Example 1). Contrariwise, if tool center point control is specified before cutter compensation for 5-axis machining, cutter compensation for 5-axis machining must be canceled before tool center point control is canceled (Example 2).

Example 1	Example 2
G41.2 D1	G43.4 H1
:	:
G43.4 H1	G41.2 D1
:	:
G49	G40
:	:
G40	G49

If the specification sequence of G40 and G49 is reversed, alarm PS5460 is issued.

When cutter compensation for 5-axis machining is specified before tool center point control, the block for canceling tool center point control suppresses buffering. Note that, as a result, the block before the G49 block generates a compensation vector for cutter compensation for 5-axis machining, which is perpendicular to the movement.

For each of cutter compensation for 5-axis machining and tool center point control for 5-axis machining, two commands, type 1 and type 2, are provided. Be sure to specify commands of the same type. If commands of different types are specified, alarm PS5460 is issued.

When the tool is tilted by address Q in type 2 command specification, if Q is specified in both of the block for starting tool center point control for 5-axis machining and the block for starting cutter compensation for 5-axis machining, the Q command specified earlier becomes valid.

3.5.3 Restriction on machine configurations having table rotation axes (table rotation type and mixed-type)

Unavailable commands

For machines having table rotation axes, the following commands cannot be specified during cutter compensation for 5-axis machining:

- G functions of group 01 other than G00 and G01

Use with tool center point control

When the workpiece coordinate system is used as the programming coordinate system, the same restrictions as for the tool rotation type apply. See "Use with tool center point control" in the restrictions on the tool rotation type described previously. When the table coordinate system is used as the programming coordinate system, the restrictions to apply will be explained below.

Restrictions when the table coordinate system is used as the programming coordinate system

Use with tool center point control

When cutter compensation for 5-axis machining and tool center point control for 5-axis are specified together, tool center point control must be specified before cutter compensation. In addition, tool center point control must be canceled after cutter compensation for 5-axis machining is canceled. (Example 1)

If tool center point control is specified in the cutter compensation mode for 5-axis machining (Example 2), or if tool center point control is canceled without canceling cutter compensation for 5-axis machining (Example 3), alarm PS5460 is issued.

Example 1 (Correct specification)	Example 2 (Specification resulting in alarm)	Example 3 (Specification resulting in alarm)
G43.4 H1 : G41.2 D1 : G40 : G49	G41.2 D1 : G43.4 H1 : G49	G43.4 H1 : G41.2 D1 : G49

If the setting of the programming coordinate system differs between cutter compensation for 5-axis machining and tool center point control for 5-axis machining, specifying both functions together results in alarm PS5460. (See the following table:)

	TBP = 0		TBP = 1	
	WKP = 0	WKP = 1	WKP = 0	WKP = 1
Tool center point control for 5-axis machining	Table coordinate system	Workpiece coordinate system	Table coordinate system	Workpiece coordinate system
Cutter compensation for 5-axis machining	Workpiece coordinate system			
Both specified together	Alarm PS5460			

Programming coordinate system determined by bit 4 (TBP) of parameter No. **19746** and bit 5 (WKP) of parameter No. **19696**. The table rotation axis position at the start of tool center point control for 5-axis machining and the table rotation axis position at the start of cutter compensation for 5-axis machining must always match.

Each programming coordinate system matches the workpiece coordinate system used when each function is started.

If these functions are started when their table rotation axis positions differ, the programming coordinate system for tool center point control for 5-axis machining and the programming coordinate system for cutter compensation for 5-axis machining do not match, which results in alarm PS5460.

Example: When the A-axis is the table rotation axis:

(Correct example)

G90 G00 A0.0

G43.4 H1

G01 Z100.0 F1000.

G41.2 D1 ← After G43.4 is specified, G41.2 is specified without A-axis movement.

:

(Wrong example)

G90 G00 A0.0

G43.4 H1

G01 Z100.0 A30.0 F1000.

G41.2 D1 ← After G43.4 is specified, A-axis movement is made, then G41.2 is specified.

:

* Furthermore, no A-axis command can be included in the block specifying G41.2.

For each of cutter compensation for 5-axis machining and tool center point control for 5-axis machining, two commands, type 1 and type 2, are provided. Be sure to specify commands of the same type. If commands of different types are specified, alarm PS5460 is issued. When the tool is tilted by address Q in type 2 command specification, if Q is specified in both of the block for starting tool center point control for 5-axis machining and the block for starting cutter compensation for 5-axis machining, the Q command specified earlier becomes valid.

Deceleration at a corner

Under cutter compensation for 5-axis machining, the controlled point may move along a curve even if a straight-line command is issued. Some commands may cause a corner movement.

For this reason, the tool may decelerate if small values are set as the permissible speed difference in a corner (parameter No. 1783) and permissible acceleration (parameter Nos. **1660** and **1737**).

Specifiable G codes

When the table coordinate system is used as the programming coordinate system, the G codes that can be specified in the cutter compensation mode for 5-axis machining are listed below.

Specifying a G code other than these codes results in alarm PS5460:

- Positioning (G00)
- Linear interpolation (G01)
- Dwell (G04)
- Exact stop (G09)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Stored stroke check function (G22/G23)
- Cutter or tool nose radius compensation : preserve vector (G38)
- Cutter or tool nose radius compensation : corner circular interpolation (G39)
- Cutter compensation : cancel (G40)
- Tool length compensation cancel (G49)
- Scaling (G50/G51)
- Exact stop mode (G61)
- Automatic corner override mode (G62)
- Cutting mode(G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Absolute programming (G90)
- Incremental programming (G91)

Modal G codes that allow specification of cutter compensation for 5-axis machining

When the table coordinate system is used as the programming coordinate system, cutter compensation for 5-axis machining can be specified in the modal G code states listed below.

In a modal state other than the following modal G codes, specifying tool center point control results in alarm PS5421.

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Canned cycle cancel (G80)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

Milling

- Coordinate system rotation start or 3-dimensional coordinate conversion mode on (G69)

- Feed per minute (G94)
- Polar coordinate interpolation mode cancel (G113)

Turning

- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))

Specification of axes not relating to cutter compensation for 5-axis machining

Axes not relating to cutter compensation for 5-axis machining cannot be specified. If such an axis is specified, alarm PS5460 is issued.

3.6 Examples

This is an example in which each side of a square is cut at an angle of 30 degrees on the B-axis in a mixed-type machine.

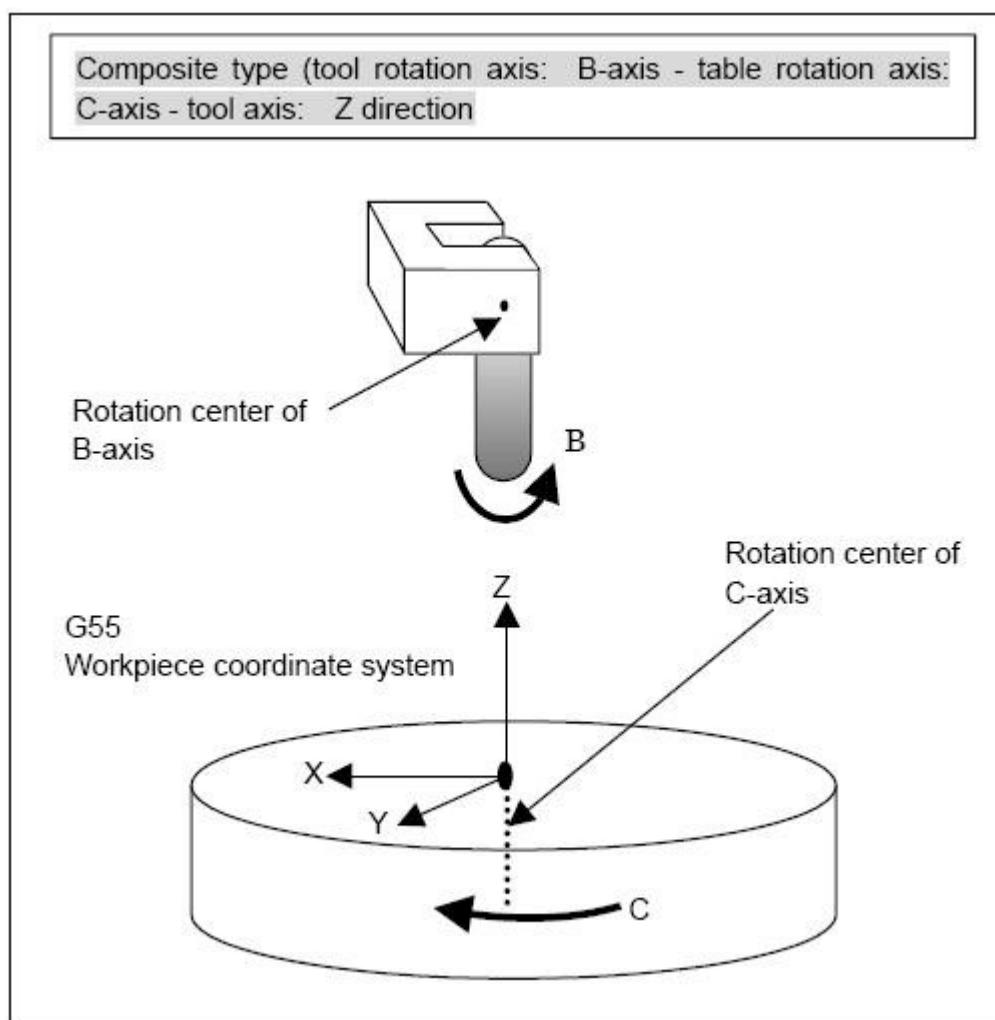


Figure 3.43: Machine configuration for the examples

Programs 1 to 3 all perform the same machining.

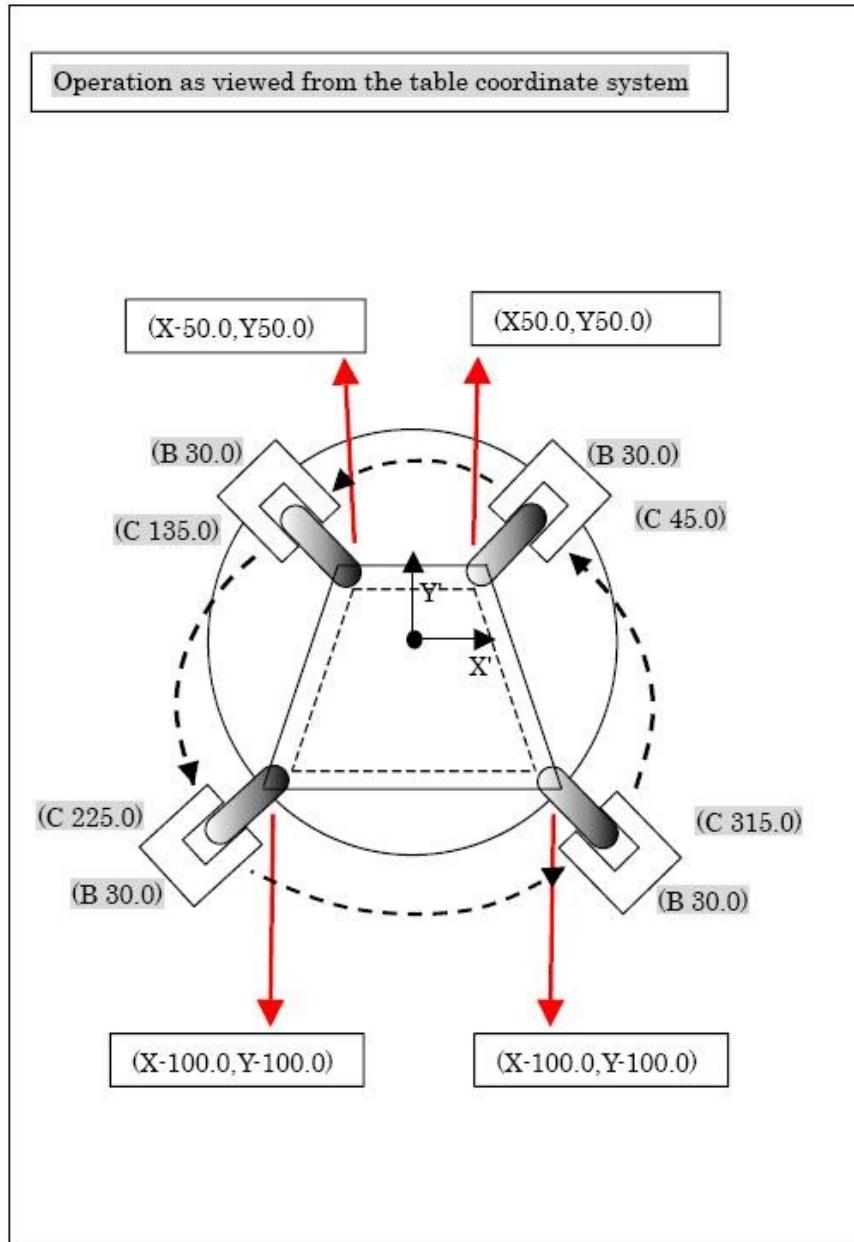


Figure 3.44: Illustration of the examples

Figure 3.44 shows the attitudes of the workpiece (object to be machined) and the tool head (relative to the workpiece (object to be machined)) as viewed in the positive Z direction of the programming coordinate system fixed to the table (table coordinate system).

Program 1: Type 1 and the table coordinate system is selected as a programming coordinate system

O101 (Sample Program1) ;

N10 G55 ;

Preparations for the programming coordinate system

N20 G90 X0 Y0 Z300.0 B0 C0 ;

Movement to the initial position

N30 G01 G43.4 H01 Z40.0 F500. ;

Start of tool center point control

H01 is a tool length compensation number

N40 G41.2 D01	Start of cutter compensation D01 is a tool cutter compensation number
N50 X50.0 Y50.0 Z20.0 B30.0 C45.0 ;	The Z-axis height on the machining plane is 20.0.
N60 X-50.0 C135.0 ;	
N70 X-100.0 Y-100.0 C225.0 ;	
N80 X100.0 C315.0 ;	
N90 X50.0 Y50.0 C405.0 ;	
N100 X0 Y0 Z40.0 B0 C360.0 ;	
N110 G40	Cutter compensation cancellation N130 M30;
N120 G49 Z300.0 ;	Tool center point control cancellation Movement to the initial position on the Z-axis
N130 M30;	

Program 2: Type 1 and the workpiece coordinate system is selected as a programming coordinate system
(Note that the values specified in N50 to N90 differ from those in program 1.)

O102 (Sample Program 2) ;	
N10 G55 ;	Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;	Movement to the initial position
N30 G01 G43.4 H01 Z40.0 F500. ;	Start of tool center point control H01 is a tool length compensation number
N40 G41.2 D01	Start of cutter compensation D01 is a tool cutter compensation number
N50 X70.711 Y0 Z20.0 B30.0 C45.0 ;	The Z-axis height on the machining plane is 20.0.
N60 C135.0 ;	
N70 X141.421 C225.0 ;	
N80 C315.0 ;	
N90 X70.711 C405.0 ;	
N100 X0 Y0 Z40.0 B0 C360.0 ;	
N110 G40	Cutter compensation cancellation N130 M30;
N120 G49 Z300.0 ;	Tool center point control cancellation Movement to the initial position on the Z-axis
N130 M30;	

Program 3: When the type 2 is used: (The table coordinate system is selected as a programming coordinate system)

O103 (Sample Program 3) ;	
N10 G55 ;	Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;	Movement to the initial position
N30 G01 G43.4 H01 Z40.0 F500. ;	Start of tool center point control H01 is a tool length compensation number
N40 G41.6 D01	Start of cutter compensation D01 is a tool cutter compensation number
N50 X50.0 Y50.0 Z20.0 I35.355 J35.355 K86.603 ;	The Z-axis height on the machining plane is 20.0.
N60 X-50.0 I-35.355 J35.355 K86.603 ;	
N70 X-100.0 Y-100.0 I-35.355 J-35.355 K86.603 ;	
N80 X100.0 I35.355 J-35.355 K86.603 ;	
N90 X50.0 Y50.0 I35.355 J35.355 K86.603 ;	
N100 X0 Y0 Z40.0 K1.0 ;	
N110 G40	Cutter compensation cancellation N130 M30;
N120 G49 Z300.0 ;	Tool center point control cancellation Movement to the initial position on the Z-axis
N130 M30;	

By using type 2 as in program 3, the same program can be used with machines with different configurations, regardless of whether the machine configuration is the tool rotation type, table rotation type, or mixed-type.

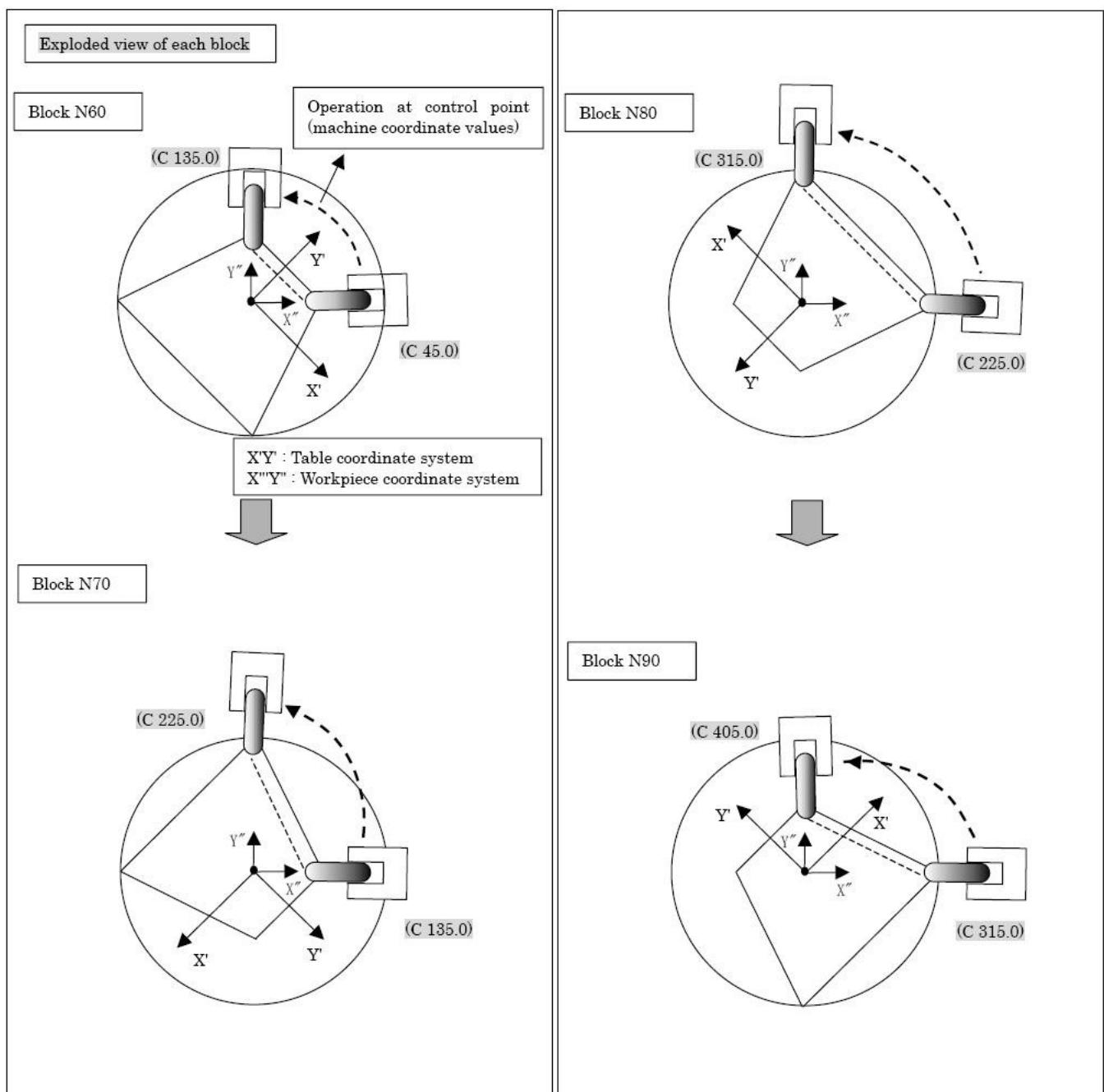


Figure 3.45: Exploded view

4 NURBS for 5-axis machining

Many computer-aided design (CAD) systems used to design metal dies for automobiles and airplanes utilize non-uniform rational B-spline (NURBS) to express a sculptured surface or curve for the metal dies. NURBS interpolation function enables NURBS curve expression to be directly specified to the CNC. This eliminates the need for approximating the NURBS curve with minute line segments and offers many advantages such as short part program. In the conventional NURBS interpolation, up to three axes can be specified as the axis of NURBS interpolation. On the other hand, in NURBS interpolation for 5-axis machining, up to five axes including two rotary axes can be specified. According to 5-axis machining with rotary axes, much smoother surface is realized. Moreover NURBS interpolation for 5-axis machining can be used during Tool center point control for 5-axis machining (type I: G43.4). And according to this, the same program can be used even if tool length is changed by tool change.

This feature is included in the option feature NURBS interpolation.

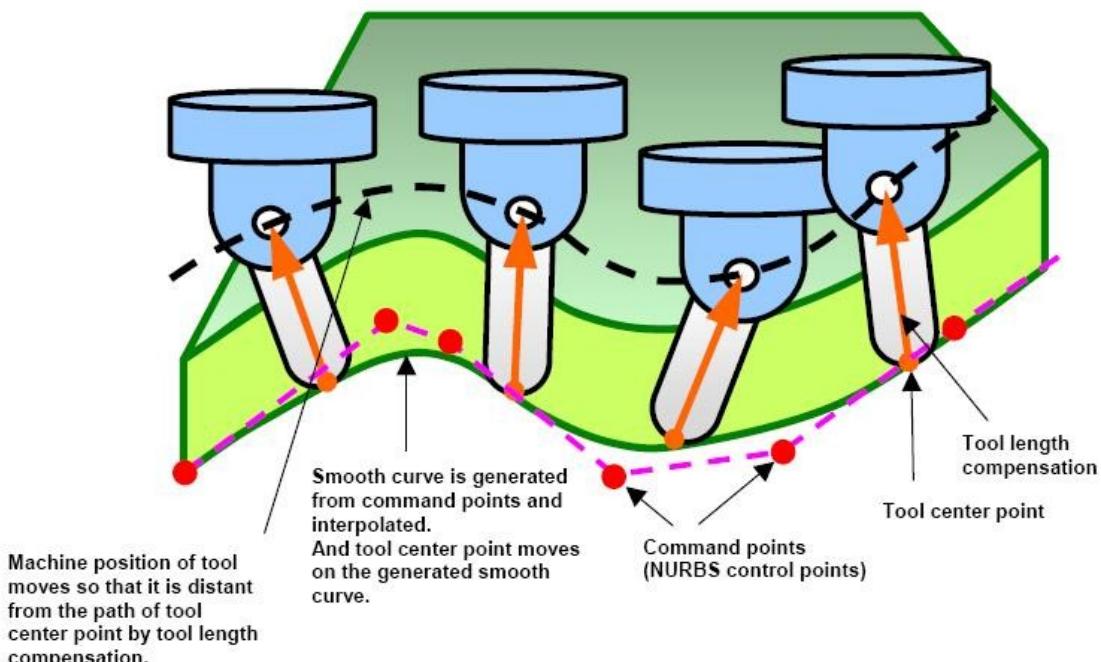


Figure 4.1: NURBS for 5-axis machining with TCP

4.1 Programming

G06.2 [P_] K_X_Y_Z_α_β_[R_] [F_];	Starts NURBS interpolation mode
K_X_Y_Z_α_β_[R_];	
K_X_Y_Z_α_β_[R_];	
...	
K_X_Y_Z_α_β_[R_];	
K_;	
...	
K_;	
G01 ... ;	

P_ Rank of NURBS curve

X_Y_Z_ Control point

α_β_ Control point (rotary axes)

R_	Weight
K_	Knot
F_	Feedrate

Format for NURBS interpolation for 5-axis machining is same as that for the conventional NURBS interpolation, but up to five axes that include two rotary axes can be specified as the axis of NURBS interpolation.

When NURBS interpolation for 5-axis machining and Tool center point control (type I: G43.4) are used together, Tool center point control (G43.4) should be commanded first, and after that NURBS interpolation for 5-axis machining (G06.2) should be commanded. When they are canceled, NURBS interpolation for 5-axis machining should be canceled (G-code command in group 1 except for G06.2) first, and after that Tool center point control should be canceled (G49).

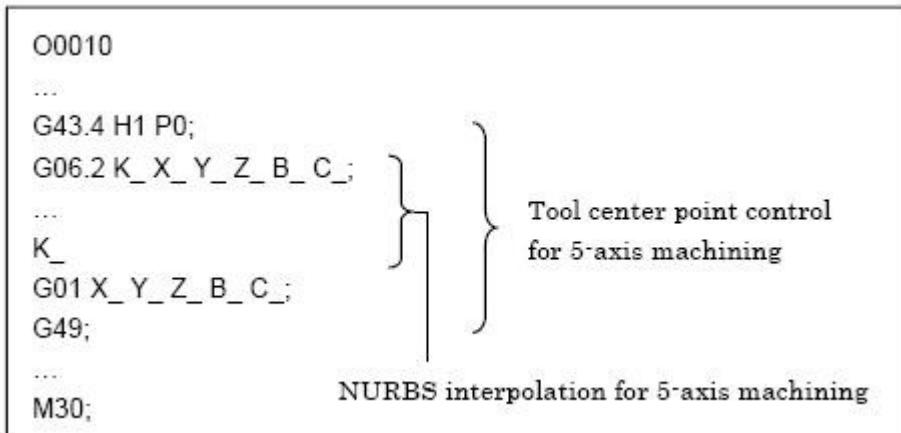


Figure 4.2: NURBS with TCP

4.2 Explanations

Explanation is same as the conventional NURBS interpolation.

4.3 Limitations

Tool center point control for 5-axis machining	NURBS interpolation for 5-axis machining can be used together with Tool center point control for 5-axis machining (type I: G43.4), but cannot be used together with Tool center point control for 5-axis machining (type II: G43.5) and Tool posture control of Tool center point control for 5-axis machining.
Tool radius compensation for 5-axis machining	Tool radius compensation for 5-axis machining cannot be simultaneously executed. NURBS interpolation for 5-axis machining can only be specified after Tool radius compensation for 5-axis machining has been canceled.

Other restrictions except for the number of controlled axes are same as the conventional NURBS interpolation.

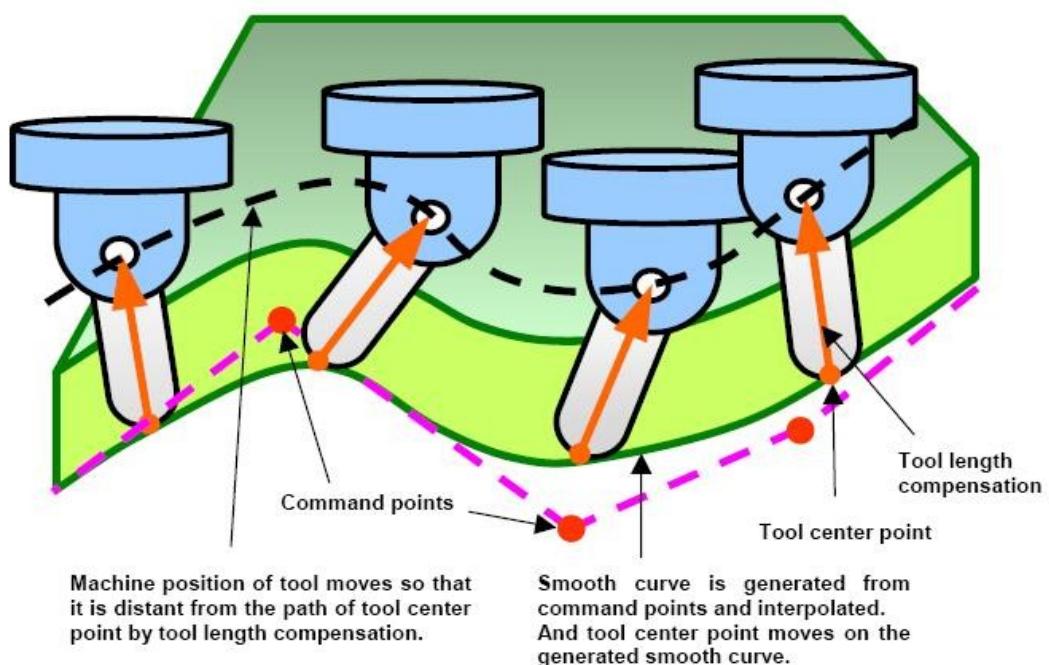
5 Nano smoothing for 5-axis machining

When a desired sculptured surface is approximated by minute segments, Nano smoothing function generates a smooth curve inferred from the programmed segments and performs necessary interpolation.

In the conventional Nano smoothing, only the three basic axes (X, Y, and Z) or their parallel axes can be specified as the axis of Nano smoothing. On the other hand, in Nano smoothing for 5-axis machining, two rotary axes, besides the three basic axes (and their parallel axes), can also be specified. Much smoother surface is realized by 5-axis machining with rotary axes.

Moreover Nano smoothing for 5-axis machining can be used during Tool center point control for 5-axis machining (type I: G43.4). And according to this, the same program can be used even if tool length is changed by tool change.

This feature is an option, and the other option Nano smoothing is also needed.



5.1 Programming

G5.1 Q3 Xp0 Yp0 Zp0 α0 β0 ;	Nano smoothing for 5-axis machining
G5.1 Q0 ;	Nano smoothing for 5-axis machining cancel
Xp	X-axis or an axis parallel to the X-axis
Yp	Y-axis or an axis parallel to the Y-axis
Zp	Z-axis or an axis parallel to the Z-axis
α, β	Rotary axes

Format for Nano smoothing for 5-axis machining is same as that for the conventional Nano smoothing, but two rotary axes, adding to three basic axes (X, Y, and Z) and their parallel axes, can be specified as the axis of Nano smoothing.

Please see the explanation for Nano smoothing about conditions to enable Nano smoothing and note.

When Nano smoothing for 5-axis machining and Tool center point control for 5-axis machining (type I: G43.4) are used together, Tool center point control for 5-axis machining (G43.4) should be commanded first, and after that Nano smoothing for 5-axis machining (G05.1 Q3) should be commanded. When they are canceled, Nano smoothing for 5-axis machining should be canceled (G05.1 Q0) first, and after that Tool center point control for 5-axis machining should be canceled (G49).

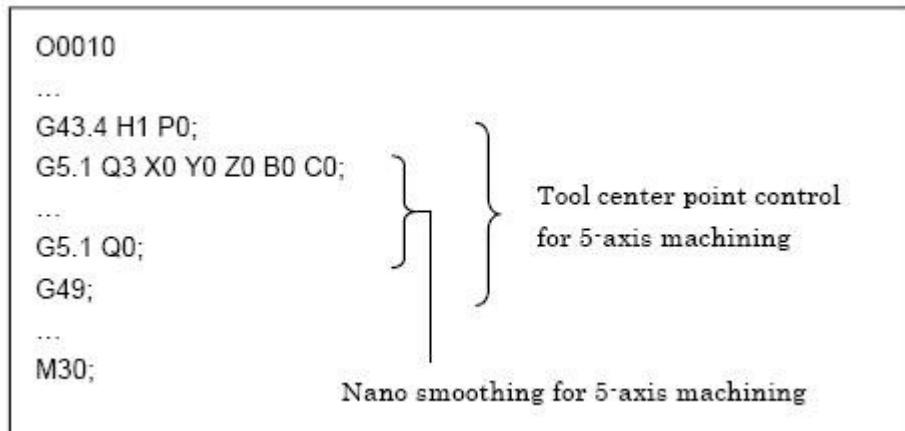


Figure 5.1: Nano-smoothing

5.2 Explanations

Three basic axes (and their parallel axes) and two rotary axes are interpolated independently in Nano smoothing for 5-axis machining.

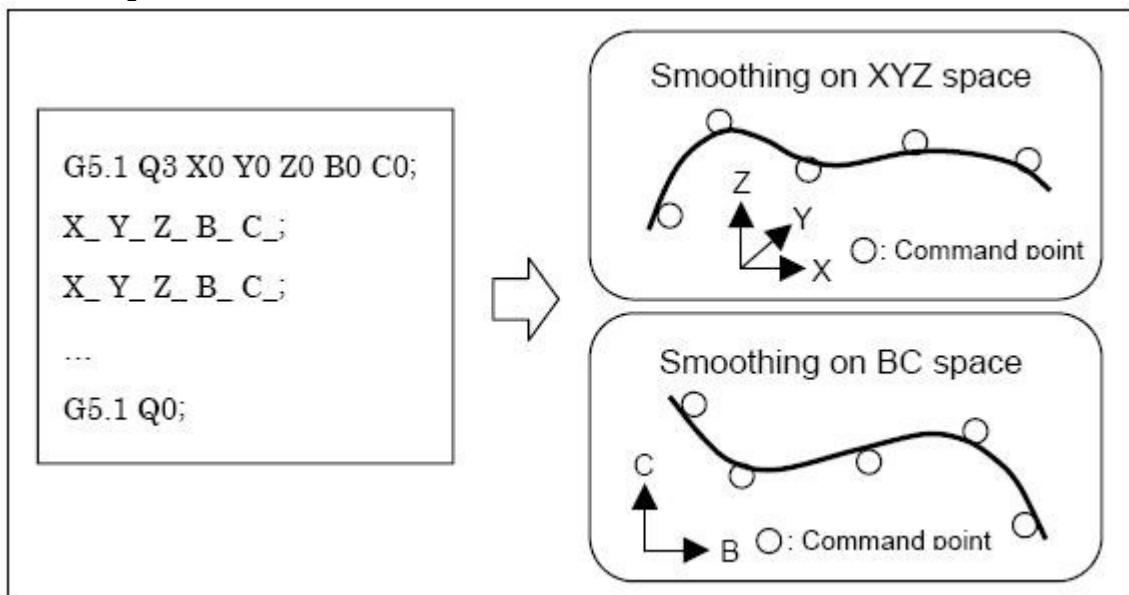


Figure 5.2: Method of Nano-smoothing for 5-axis machining

Correct insertion point

At the interpolation for Nano smoothing for 5-axis machining, insertion points are generated firstly, then the insertion points are corrected, and smooth curve is generated with the corrected insertion points. This method is same as that of the conventional Nano smoothing. In three basic axes (or their parallel axes) interpolation, an insertion points are corrected so that the difference (compound difference of three axes) between the insertion points and the corrected insertion points are less than the tolerance which is set in parameter.

In two rotary axes interpolation, an insertion point is corrected so that the both differences (elements for each axis) between the insertion point and the corrected insertion point are less than each tolerance which is set in parameter for each axis.

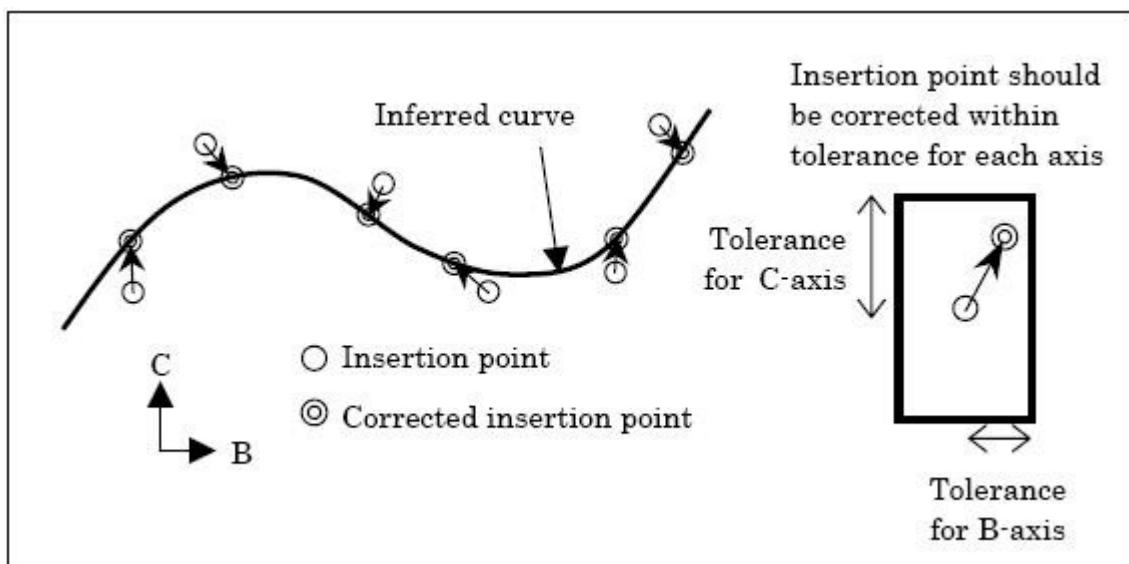


Figure 5.3: Correct insertion points for rotary axes

Making a decision on the basis of the spacing between adjacent programmed points, Making a decision at a corner

Only three basic axes (or their parallel axes) are related to “the programmed block length” in “Conditions to enable Nano smoothing” of explanation for Nano smoothing, and two rotary axes have no relation to it. In the same way, only three basic axes (or their parallel axes) are related to “Making a decision on the basis of the spacing between adjacent programmed points” and “Making a decision at a corner”, and two rotary axes have no relation to them. However, in the block that Nano smoothing mode is canceled according to these conditions, Nano smoothing for rotary axes is also canceled.

Other explanation is same as the conventional Nano smoothing.

5.3 Limitations

Nano smoothing for 5-axis machining can be used together with Tool center point control for 5-axis machining (type I: G43.4). However PS5421 alarm (ILLEGAL COMMAND IN G43.4/G43.5) occurs if Nano smoothing for 5-axis machining is used with Tool center point control for 5-axis machining (type II: G43.5) or Tool posture control of Tool center point control for 5-axis machining.

Other restrictions are same as the conventional Nano smoothing.

III TILTED WORKING PLANE

1 Tilted Working Plane Command

1.1 Overview

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. The feature coordinate

system is defined in the workpiece coordinate system.

For explanations about the relationship between the feature coordinate system and workpiece coordinate system, see Figure 1.1.

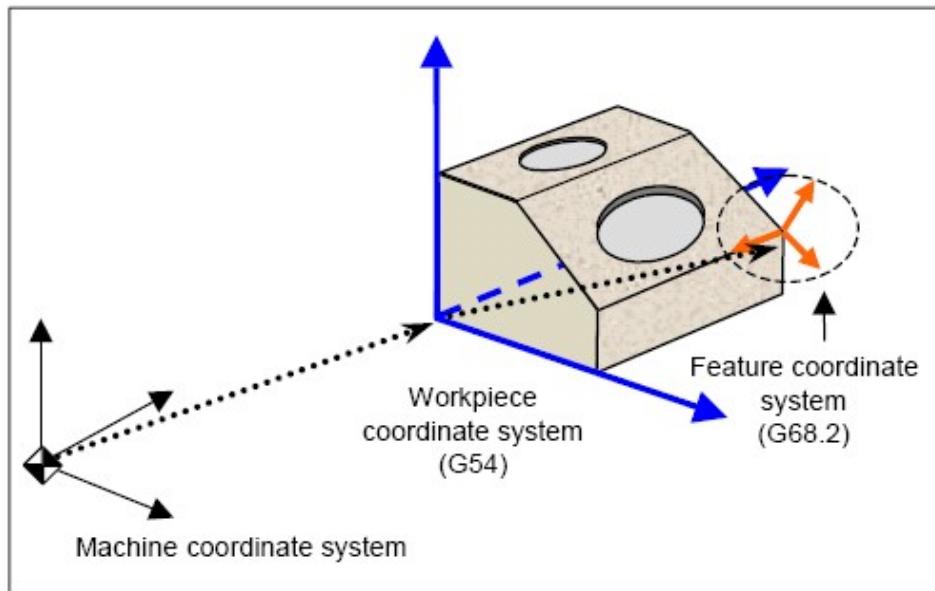


Figure 1.1: Feature coordinate system

The G68.2 command causes the programming coordinate system to switch to the feature coordinate system. The commands in all subsequent blocks are assumed to be specified in the feature coordinate system until G69 appears.

If G68.2 specifies the relationship between the feature coordinate system and the workpiece coordinate system, G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction even if no angle is specified for the rotary axis. (See Figure 1.3)

For explanations about the tool axis direction, see Figure 1.2.

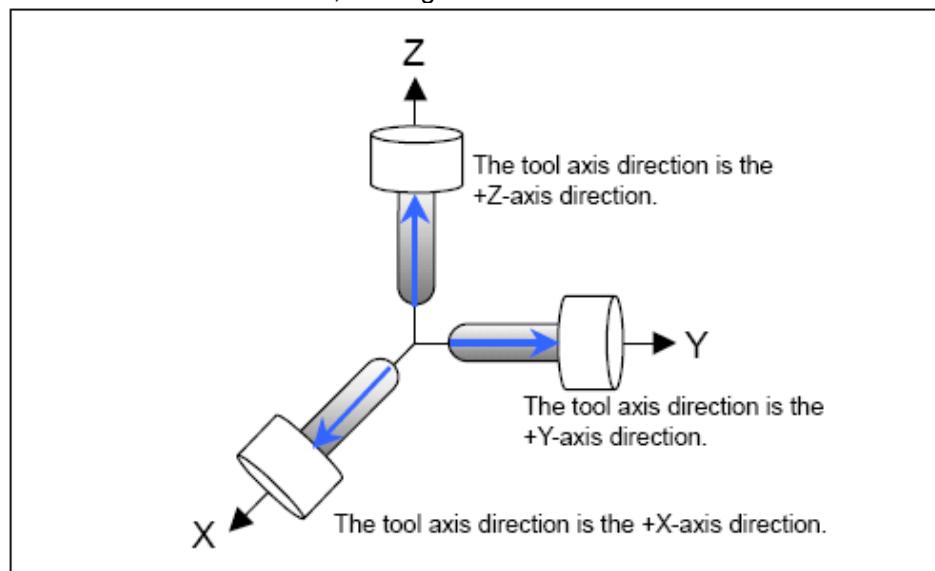


Figure 1.2: Tool axis direction

This function regards the direction normal to the machining plane as the +Z-axis direction of the feature

coordinate system. After the G53.1 command, the tool is controlled so that it remains perpendicular to the machining plane.

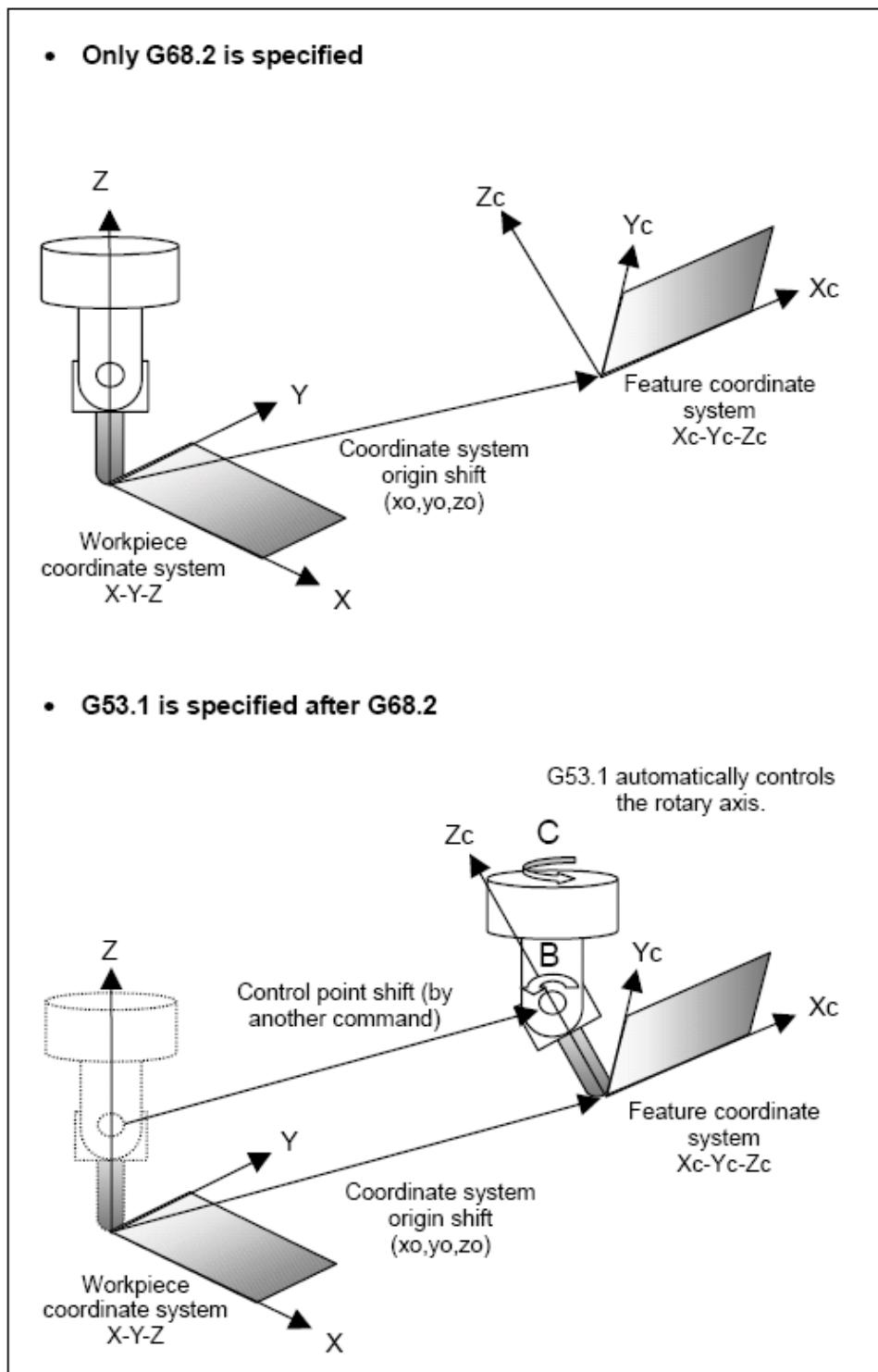


Figure 1.3: G68.2 and G53.1 command

This function is applicable to the following machine configurations.

- Tool rotation type machine controlled with two tool rotation axes
- Table rotation type machine controlled with two table rotation axes
- Mixed-type machine controlled with one tool rotation axis and one rotary axis

The function can also be used for a machine configuration in which the rotary axis for controlling the tool does not intersect the rotary axis for controlling the table.

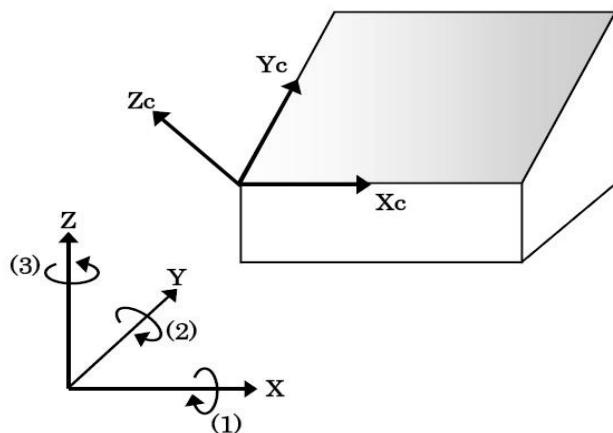
1.2 Programming

1.2.1 Feature coordinate system setting by Euler's angles

G68.2 X_ Y_ Z_ Iα Jβ Kγ ;	Set's feature coordinate system
G53.1 ;	Controls tool axis direction

X_ Y_ Z_ Origin of feature coordinate system
I α J β K γ Euler's angle for determining the orientation of the feature coordinate system.

1.2.2 Feature coordinate system setting by Roll-Pitch-Yaw



G68.2 P1 Q_ X_ Y_ Z_ Iα Jβ Kγ ;	Set's feature coordinate system
G53.1 ;	Controls tool axis direction

Q

Order to rotate the coordinate system:

	1 st rotation	2 nd rotation	3 rd rotation
Q123	Around X	Around Y	Around Z
Q132	Around X	Around Z	Around Y
Q213	Around Y	Around X	Around Z
Q321	Around Z	Around Y	Around X
Q231	Around Y	Around Z	Around X
Q312	Around Z	Around X	Around Y

CAUTION

- 1) When address Q is omitted, a coordinate system is converted in order of rotation around X axis, Y axis and Z axis of the work coordinate system (equivalent to Q123).
- 2) The alarm PS5457 occurs, when a value other than the above is set to address Q.

X, Y, Z

Origin of feature coordinate system

I

Rotation angle around X axis (**Roll** angle (1))

J

Rotation angle around Y axis (**Pitch** angle(2))

K

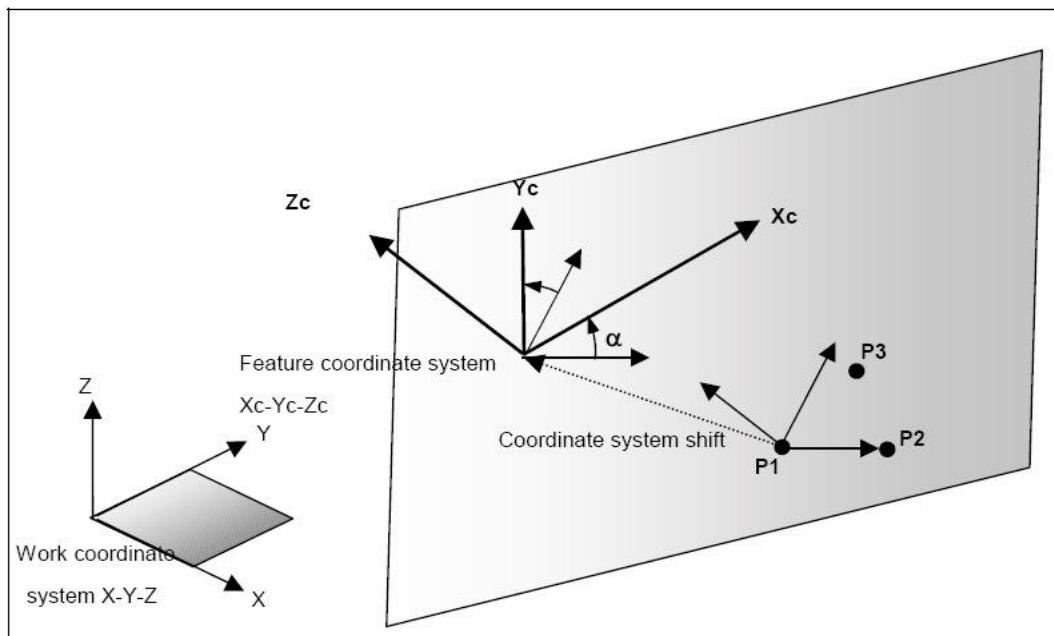
Rotation angle around Z axis (**Yaw** angle (3))**1.2.3 Feature coordinate system setting by three points**

G68.2 P2 Q0 X_{x0} Y_{y0} Z_{z0} Rα ;	Set's feature coordinate system (including shift and rotation)
G68.2 P2 Q1 X_{x1} Y_{y1} Z_{z1}	Define 1 st point
G68.2 P2 Q2 X_{x2} Y_{y2} Z_{z2}	Define 2 nd point
G68.2 P2 Q3 X_{x3} Y_{y3} Z_{z3}	Define 3 rd point
G53.1 ;	Controls tool axis direction

Q0 X_{x0} Y_{y0} Z_{z0}Shift value from the 1st point to the feature coordinate system origin (regarded as 0, when not commanded)Q1 X_{x1} Y_{y1} Z_{z1}The 1st point (feature coordinate system origin)Q2 X_{x2} Y_{y2} Z_{z2}The 2nd pointQ3 X_{x3} Y_{y3} Z_{z3}The 3rd point

R

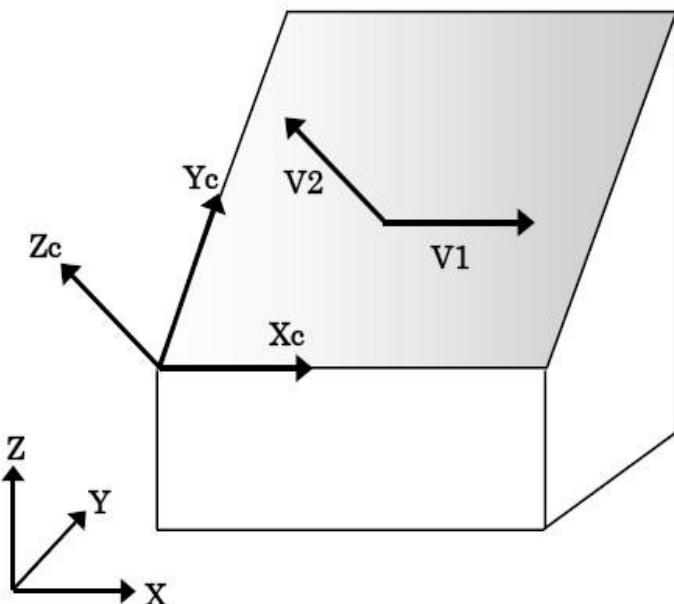
Rotation angle around Z axis on feature coordinate system (Regarded as 0, when not commanded). This can be commanded in any block of G68.2 P2.


CAUTION

- 1) Tilted working plane is defined by three G68.2P2 commands (Q1, Q2, Q3). When G68.2P2 commands are not given consecutively, the alarm PS5457 occurs.
- 2) In the following cases and when a value other than the above is set to address Q, the alarm PS5457 occurs.
 - Same point in three points (A plane is not defined.)
 - Three points in alignment (A plane is not defined.)
 - The distance between the line including two points in three points and the other point is shorter than parameter No.11220. (A plane is unstable.)

1.2.4 Feature coordinate system setting by two vectors

The tilted working plane can be commanded by X-axis and Z-axis on the feature coordinate system.



G68.2 P3 Q1 X_ Y_ Z_ Iα_1 Jβ_1 Kγ_1 ;	Set's feature coordinate system
G68.2 P3 Q2 Iα_2 Jβ_2 Kγ_2 ;	
G53.1 ;	Controls tool axis direction

X_ Y_ Z_	Origin of feature coordinate system (commanded in Q1 block)
Q1 I α_1 J β_1 K γ_1	The direction of X axis of feature coordinate system direction on Work coordinate system (1 st vector) The direction of Z axis of feature coordinate system direction on Work coordinate system (2 nd vector)
Q2 I α_2 J β_2 K γ_2	The 2 nd point (feature coordinate system origin) The direction of Z axis of feature coordinate system direction on Work coordinate system (2 nd vector)

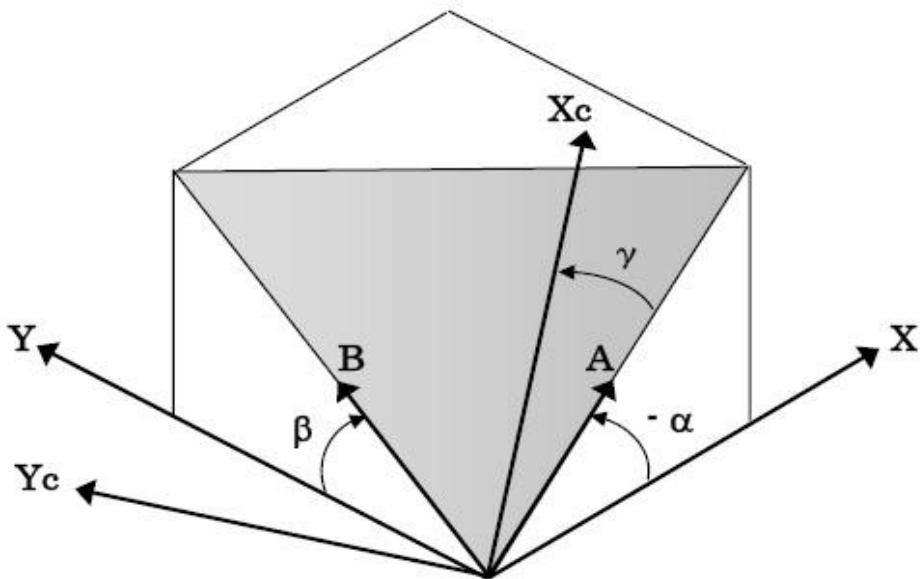
CAUTION

- 1) Tilted working plane is defined by two G68.2P3 commands (Q1,Q2). When G68.2P2 commands are not given consecutively, the alarm PS5457 occurs.
- 2) The angle between 1st vector and 2nd vector is less than 85 deg or more than 95 deg, the alarm PS5457 occurs.
- 3) When zero vector is set to 1st vector or 2nd vector, the alarm PS5457 occurs.

1.2.5 Feature coordinate system setting by projection angles

The tilted working plane can be commanded by commanding Projection angles.

The tilted working plane is defined by the two vector A and B rotated around X axis and Y axis of Work coordinate system.

**G68.2 P4 X_ Y_ Z_ I α J β K γ ;**

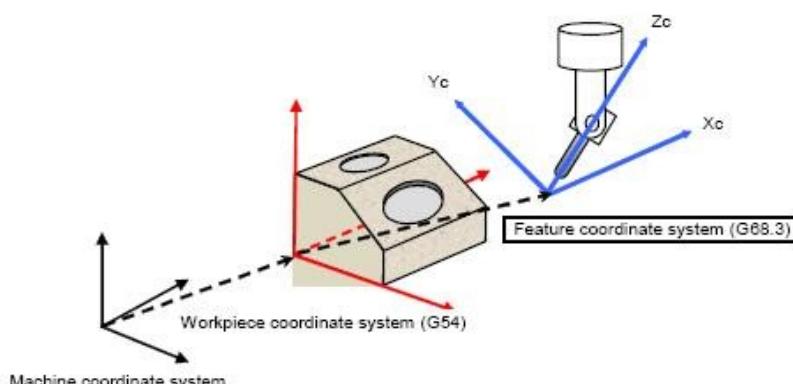
Set's feature coordinate system

G53.1

X_ Y_ Z_	Origin of feature coordinate system
α	Rotation angle for X axis around Y axis of work coordinate system
β	Rotation angle for Y axis around X axis of work coordinate system
γ	Rotation angle around Z axis of feature coordinate system

CAUTION

When two vectors are regarded as parallel (the angle between vector A and B is less than 1 degree), the alarm PS5457 occurs.

1.2.6 Feature coordinate system setting by tool axis direction**G68.3 X_{x0} Y_{y0} Z_{z0} R α**

Set's feature coordinate system

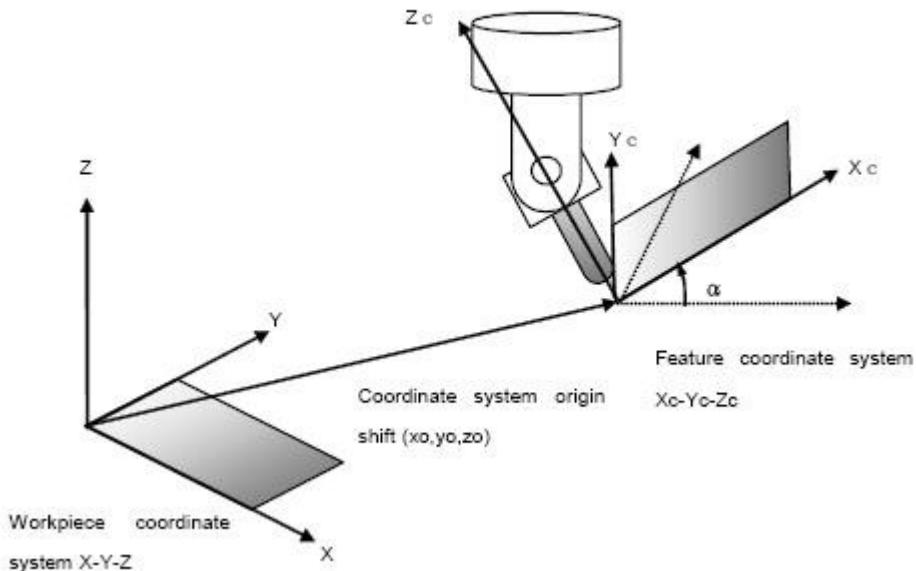
X_Y_Z_

Origin of feature coordinate system (absolute)

When 1 address or 2 addresses are omitted in X, Y, Z, alarm P/S5457 is issued.
 When 3 addresses are omitted, the current position becomes the origin of the feature coordinate system.

R

Angular displacement about the Z-axis in the feature coordinate system.
 The default is 0°.

**Figure 1.4: G68.3 command**

1.2.7 Feature coordinate system canceling

G69 ;	Canceling of feature coordinate system on Milling machines
G69.1	Canceling of feature coordinate system on Turning machines

1.2.8 Multiple tilted working plane command

Absolute tilted working plane command

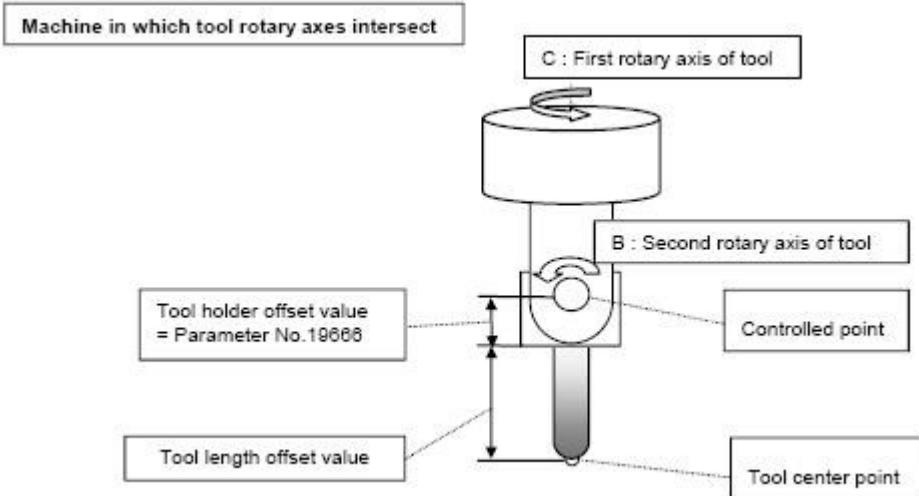
In tilted working plane mode, a new feature coordinate system based on the currently active work coordinate system is defined by additional G68.2 command.

G69/G69.1 command cancels tilted working plane mode. This function is enabled by setting parameter MTW (No.1221#0) to 1.

Tilted working plane command formats apply to absolute tilted working plane command format. Origin of the feature coordinate system is specified by the work coordinate system.

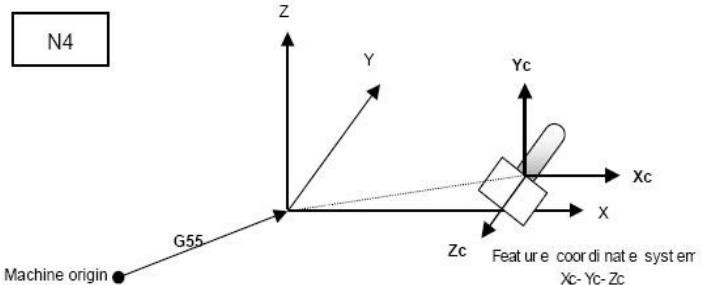
The following explains operation example by tool rotation type machine.

Rotation axis C is a tool rotation axis (master) on the Z-axis. Rotation axis B is a tool rotation axis (slave) on the Y-axis.

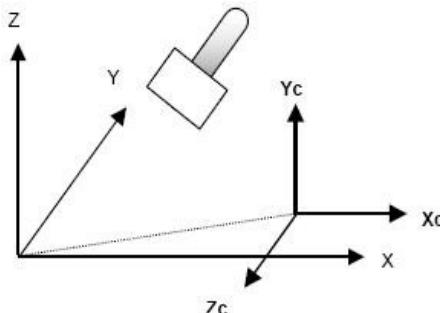


O0100; (Sample Program)

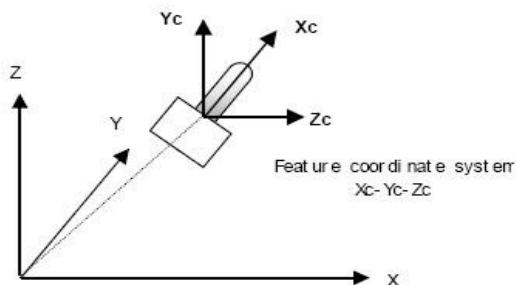
```
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
```



The tool is controlled so that it is perpendicular to the tilted working plane.

N5 X-15.0 Y0 Z-15.0 ;

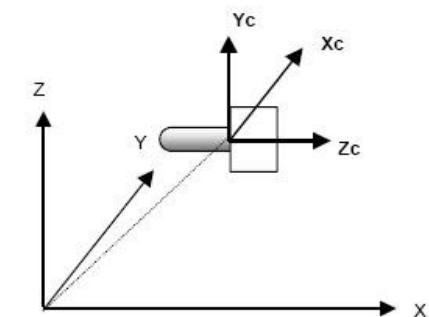
The coordinate value on the feature coordinate system is specified.

N6 G68.2 X5.0 Y20.0 Z0 I90.0 J90.0 K0 ;

The new feature coordinate system is defined.

N7 G53.1;

The tool is controlled so that it is perpendicular to the new tilted



working plane.

N8 G69 ;

Incremental tilted working plane commanding

In tilted working plane mode, a new feature coordinate system based on the currently active feature coordinate system is defined by additional G68.4 command.

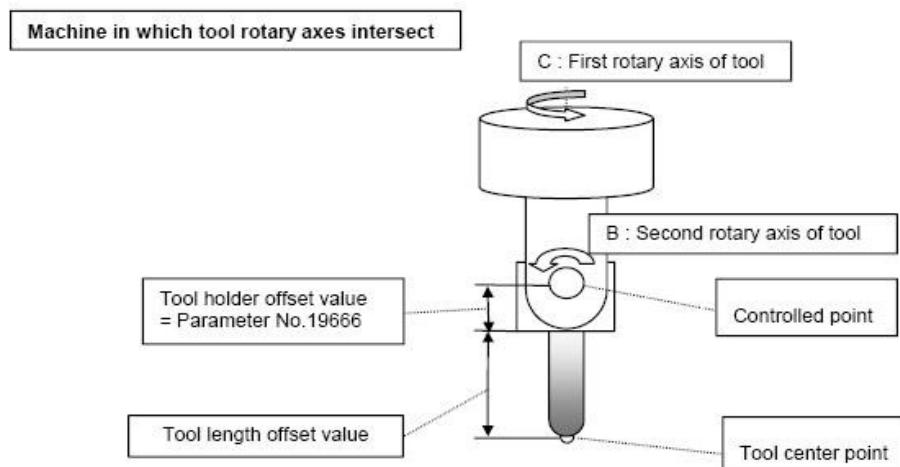
G69/G69.1 command cancels tilted working plane mode. This function is enabled by setting parameter MTW (No. 11221#0) to 1.

Tilted working plane command formats apply to incremental tilted working plane command format. Origin of the feature coordinate system is specified by the feature coordinate system.

Command type	Incremental tilted working plane command
Euler's angle	G68.4
Roll-Pitch-Yaw	G68.4 P1
3 points	G68.4 P2
2 vectors	G68.4 P3
Projection angle	G68.4 P4

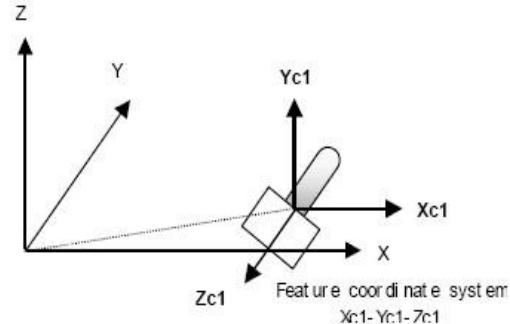
The following explains operation example by tool rotation type machine.

Rotation axis C is a tool rotation axis (master) on the Z-axis. Rotation axis B is a tool rotation axis (slave) on the Y-axis.



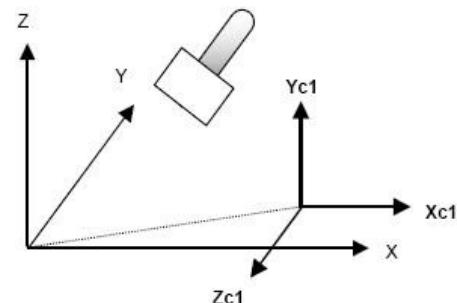
```
O0200 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
```

Feature coordinate system setting



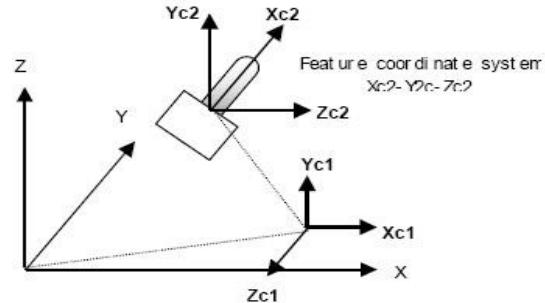
The tool is controlled so that it is perpendicular to the tilted working plane.

```
N5 X-15.0 Y0 Z-15.0 ;
```



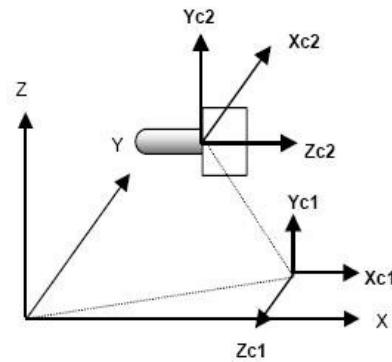
The coordinate value on the feature coordinate system is specified.

```
N6 G68.4 X-15.0 Y0 Z-15.0 I90.0 J90.0 K90.0 ;
```



The new feature coordinate system is defined.

```
N7 G53.1;
```



The tool is controlled so that it is perpendicular to the new tilted working plane.

```
N8 G69 ;
```

Feature coordinate system cancellation

```
M30 ;
```

1.3 Explanations

1.3.1 Modal commands that allows specification of tilted working plane

A tilted working plane command can be specified in the modal G code states listed below.

In a modal state other than the following modal G codes, specifying the tilted working plane command results in alarm PS5462:

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Stored stroke check function (G22/G23)
- Scaling cancel (G50)
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Exact stop mode (G61)
- Automatic corner override (G62)
- Tapping mode (G63)
- Cutting mode (G64)
- Inverse time feed (G93)
- Constant surface speed control cancel (G97)

Milling

- Polar coordinate interpolation mode cancel (G113)

Turning

- Programmable mirror image cancel (G50.1)
- Mirror image for double turret off/balanced cutting mode cancel (G69)

1.3.2 Commands that can be specified during tilted working plane command

The G codes that can be specified in the tilted working plane command mode are listed below.

Specifying a G code other than these codes results in alarm PS5462.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Automatic return to reference position (G28)
- Movement from reference position (G29)
- 2nd, 3rd and 4th reference position return (G30)

- Cutter compensation : cancel (G40)
- Cutter or tool nose radius compensation / Three-dimensional cutter compensation (G41/G42)
- Tool length compensation + (G43)
- Tool length compensation - (G44)
- Tool length compensation cancel (G49)
- Machine coordinate system setting (G53)
- Tool axis direction control (G53.1)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Absolute programming (G90)
- Incremental programming (G91)
- Canned cycle for drilling (G73, G74, G76, G80 to G89)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

Milling

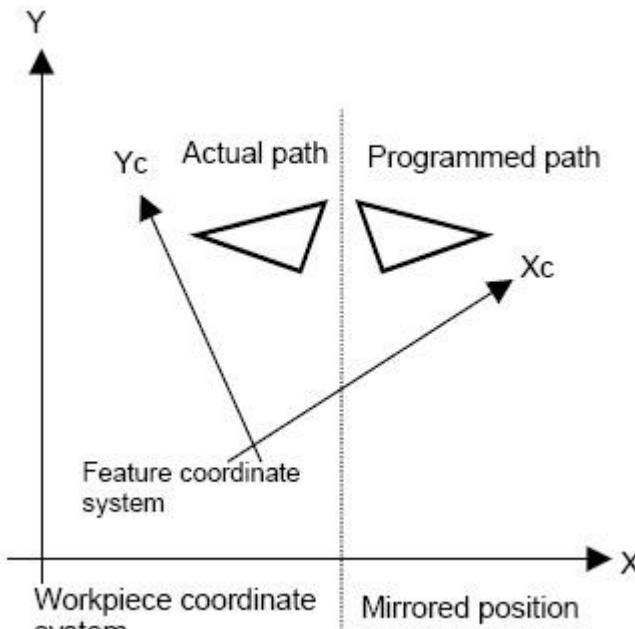
- Tool offset increase (G45)
- Tool offset decrease (G46)
- Tool offset double increase (G47)
- Tool offset double decrease (G48)
- Programmable mirror image (G50.1/G51.1)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69)
- Feed per minute (G94)
- Feed per revolution (G94)

Turning

- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G94))

1.4 Restrictions

Basic restrictions	The restrictions for this function are similar to those for the three-dimensional coordinate conversion function.
Increment system	The same increment system must be used for the basic three axes used by this function.
Rapid traverse command	The rapid traverse command must specify linear rapid traverse (parameter LRP (parameter No. 1401#1) = 1).
Feature coordinate system and three-dimensional coordinate conversion	An alarm occurs if an attempt is made to set a feature coordinate system in another feature coordinate system. An alarm also occurs if an attempt is made to set a new coordinate system by performing three-dimensional coordinate conversion in a feature coordinate system.

Positioning in the machine coordinate system	Positioning commands in the machine coordinate system, such as G28, G30, and G53, operate in the machine coordinate system rather than in the feature coordinate system.
External mirror image	If an attempt is made to use this function and the external mirror image function simultaneously, this function takes effect before the external mirror image function.
	 <p>The diagram shows a 2D coordinate system with a vertical Y-axis pointing up and a horizontal X-axis pointing right. A dotted line represents the mirror plane. The 'Feature coordinate system' is shown in the lower-left quadrant, with its origin labeled 'Yc' (vertical) and 'Xc' (horizontal). The 'Workpiece coordinate system' is shown in the upper-right quadrant, also with its origin labeled 'Yc' (vertical) and 'Xc' (horizontal). Two triangular shapes represent the 'Actual path' and 'Programmed path'. The 'Actual path' is a triangle in the Feature coordinate system, while the 'Programmed path' is a triangle in the Workpiece coordinate system. A vertical dashed line labeled 'Mirrored position' indicates the reflection of the programmed path across the mirror plane.</p>
Relationships with other modal commands	G41, G42, and G40 (cutter compensation), G43 and G49 (tool length compensation), G51.1 and G50.1 (programmable mirror image), and canned cycle commands must have nesting relationships with G68.2. In other words, first issue G68.2 when the modes mentioned above are off, then turn the modes on and off, and then issue G69.
Parallel axis control	When a parking signal is applied to an axis during parallel axis control, conversion to a feature coordinate system occurs for another axis if a move command is issued for that another axis. For this reason, an axis may move even if a parking signal has been applied to it.

2 Workpiece Setting Error Compensation

When a workpiece is placed on the machine, the workpiece is not always placed at an ideal position.

With this function, a displaced workpiece can be machined according to the program without making modifications to the program.

This function can compensate for a workpiece setting error on a rotation axis used with a function involving rotation axis operation such as a function for tool center point control for 5-axis machining, function for cutter radius compensation for 5-axis machining, and a tilted working plane command. So, a workpiece can be machined according to the program without making modifications to the program even during tool center point control for 5-axis machining, cutter compensation for 5-axis machining, and tilted working plane command.

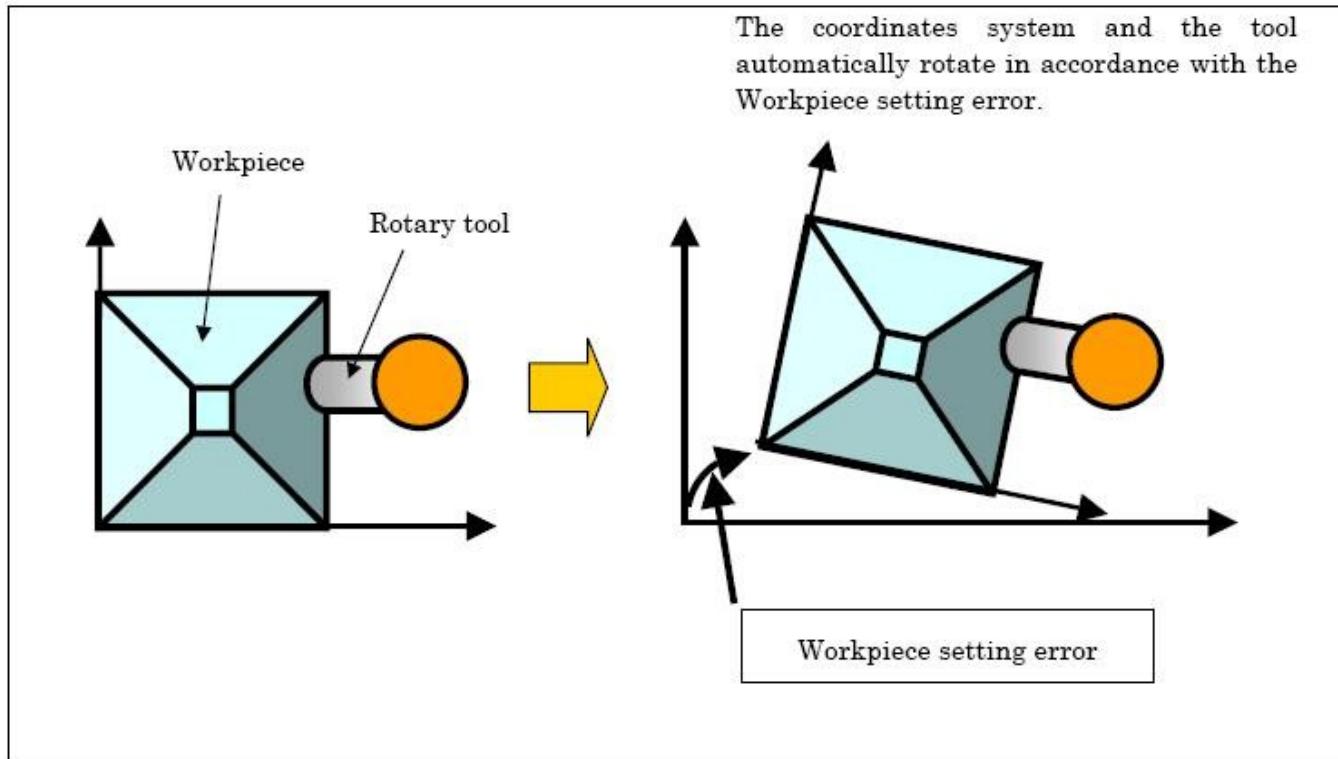


Figure 2.1: Outline of workpiece setting error compensation

2.1 Programming

G54.4 Pn	Start workpiece setting error compensation
...	
G54.4 P0	Cancel workpiece setting error compensation

Pn Workpiece setting error specification code
 n 1 to 7

A start block and cancellation block for workpiece setting error compensation suppress buffering.

2.2 Explanation

2.2.1 Workpiece setting error

A workpiece setting error is defined by the following eight variables:

- X direction error Δx
- Y direction error Δy
- Z direction error Δz
- Rotation direction error Δa (rotation error on the X-axis in degrees)
- Rotation direction error Δb (rotation error on the Y-axis in degrees)
- Rotation direction error Δc (rotation error on the Z-axis in degrees)
- Table rotation axis position 1
- Table rotation axis position 2

[About Δx , Δy , and Δz]

Δx , Δy , and Δz represent the coordinates of the origin of the workpiece coordinate system ($X'Y'Z'$ in the figure below, which is hereinafter referred to as the "workpiece setting coordinate system") based on a displaced workpiece, as viewed from the original workpiece coordinate system (XYZ in the figure below).

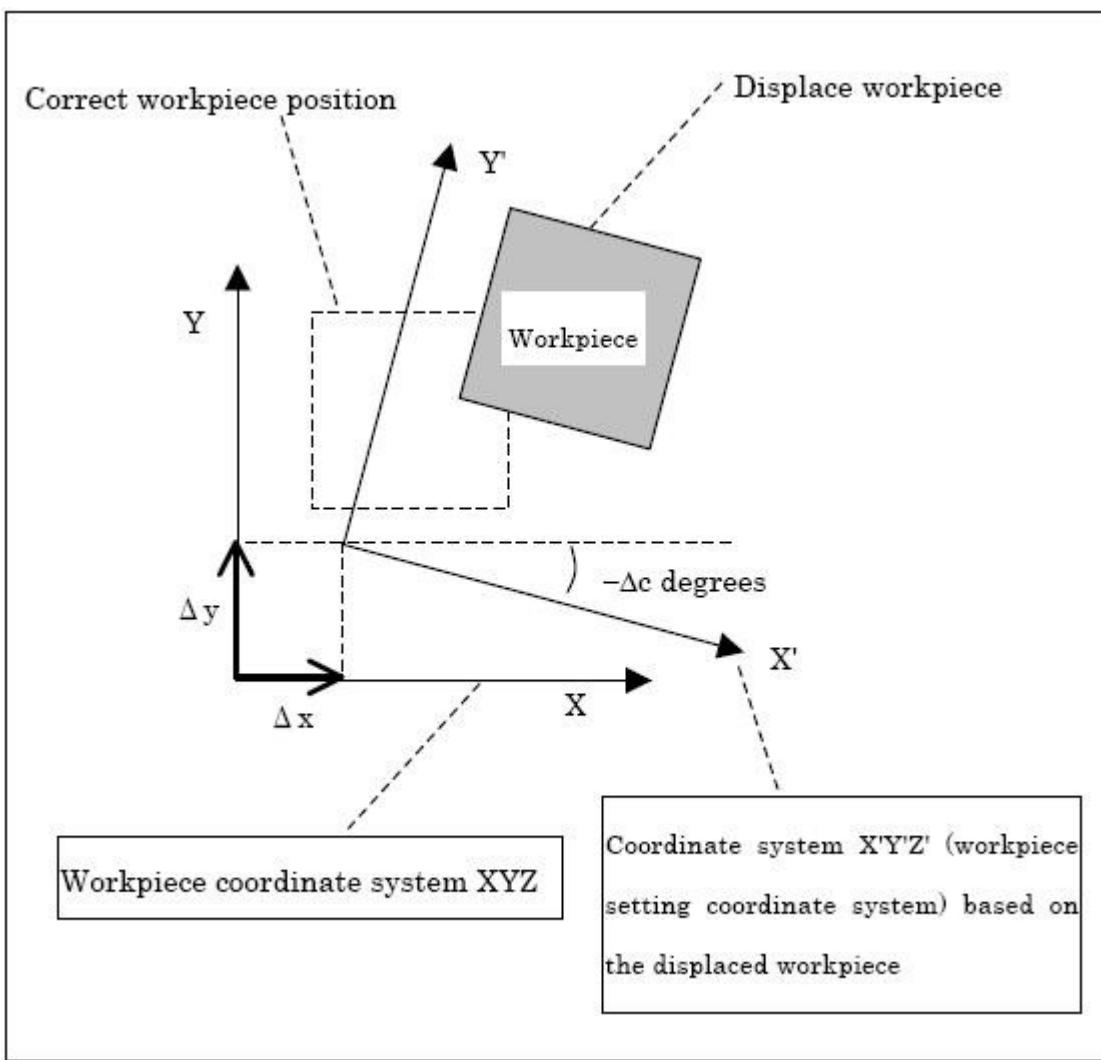


Figure 2.2: Error is X, Y, and Z directions

[About Δa , Δb , and Δc]

Δa , Δb , and Δc are defined as described below.

The workpiece coordinate system obtained by rotating a workpiece coordinate system about the X-axis by angle Δa , about the Y-axis by angle Δb , and about the Z-axis by angle Δc then shifting that workpiece coordinate system by $(\Delta x, \Delta y, \Delta z)$ from the origin of that workpiece coordinate system is supposed to match the "workpiece setting coordinate system".

(Note: In determination of positive/negative rotation, clockwise rotation is defined to be positive rotation.)

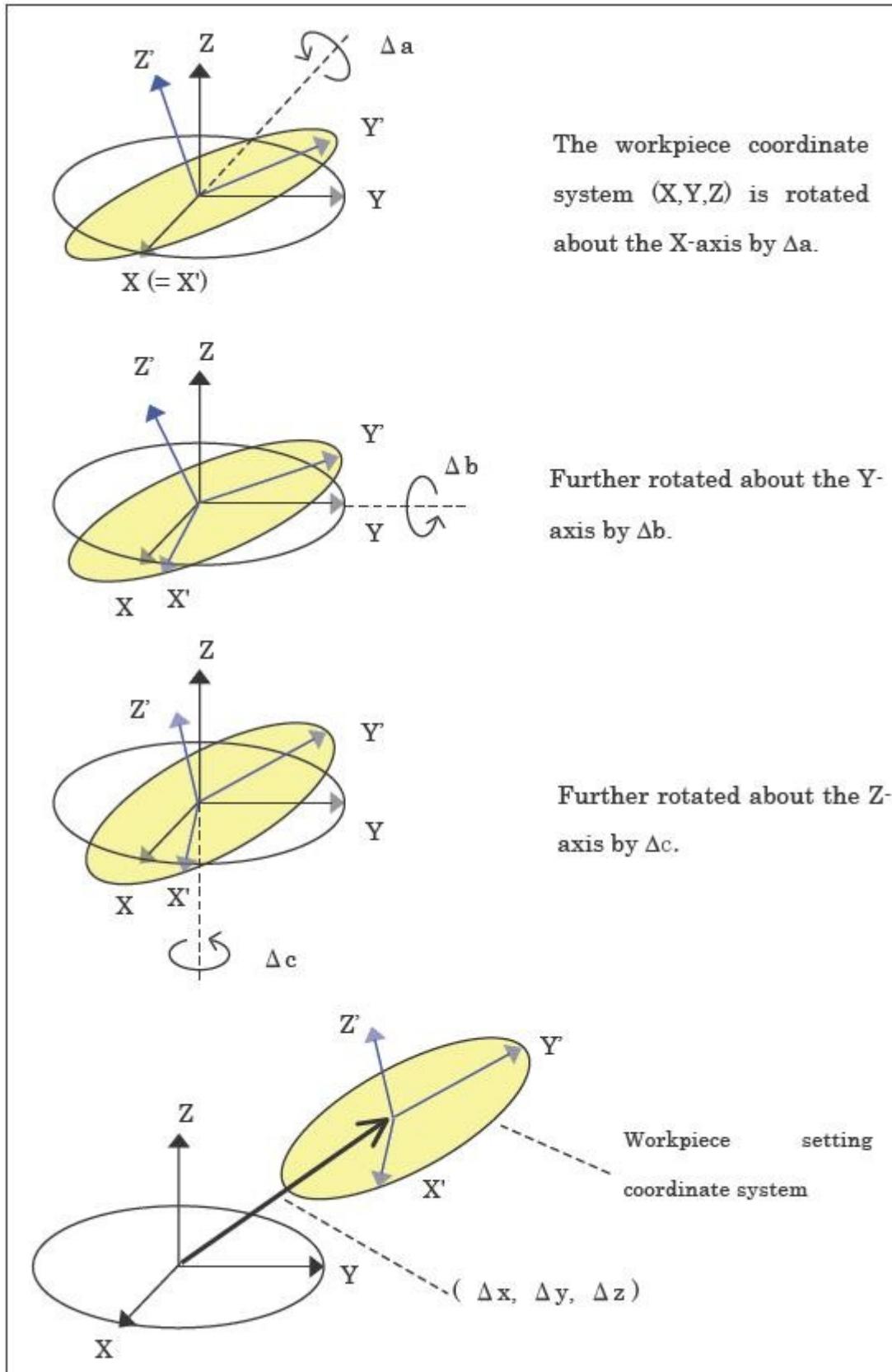


Figure 2.3: Error in rotation directions

The example of Fig. 2 assumes that $\Delta a = \Delta b = 0$ and Δc represents a nonzero value.

[About table rotation axis position 1 and table rotation axis position 2]

The table rotation axis position means the machine coordinate on the table rotation axis of a 5-axis machine with a table rotation axis (machine of table rotation type or composite type) when a workpiece setting error (displacement from the correct workpiece position to the actual workpiece position) is measured.

When two table rotation axes are used, set table rotation axis position 1 for the master rotation axis, and set table rotation axis position 2 for the slave rotation axis. (As for master rotation axis and slave rotation axis, refer to the Table 1 described later.)

When only one table rotation axis is used, set table rotation axis position 1 for the axis.

When no table rotation axis is used, or the machine used is not a 5-axis machine, table rotation axis position 1 and table rotation axis position 2 need not be set.

No setting can be made for a hypothetical axis.)

In the descriptions above, X, Y, and Z represent the three basic axes, X, Y, and Z, specified by parameter No. 1022.

If the specification of any of the three basic axes, X, Y, and Z, is missing, alarm P/S0436 is issued.

2.2.2 Specifiable G codes

When workpiece setting error compensation is enabled, the G codes listed below can be specified.

If a G code not listed below is specified, alarm PS0437 is issued.

- G00 Positioning
- G01 Linear interpolation
- G02 Circular interpolation (CW)
- G03 Circular interpolation (CCW)
- G04 Dwell
- G05.1 Q0/Q1 AI contour control mode OFF/ON
- G06.2 NURBS interpolation
- G10 Programmable data input
- G11 Programmable data input mode cancel
- G17 Plane selection (XY)
- G18 Plane selection (ZX)
- G19 Plane selection (YZ)
- G28 Return to reference position
- G29 Return from reference position
- G30 Return to 2nd, 3rd, or 4th reference position
- G40 Cutter compensation/tool-nose radius compensation/cutter compensation for
- 5-axis machining cancel
- G41 Cutter compensation/tool-nose radius compensation left
- G41.2/G41.4/G41.5 Cutter compensation left for 5-axis machining (type 1)
- G41.3 Cutter compensation for 5-axis machining (leading edge offset)
- G41.6 Cutter compensation left for 5-axis machining (type 2)
- G42 Cutter compensation/tool-nose radius compensation right
- G42.2/G42.4/G42.5 Cutter compensation right for 5-axis machining (type 1)
- G42.6 Cutter compensation right for 5-axis machining (type 2)

- G43 Tool length compensation +
- G43.1 Tool length compensation in tool axis direction
- G43.4 Tool center point control for 5-axis machining (type 1)
- G43.5 Tool center point control for 5-axis machining (type 2)
- G44 Tool length compensation -
- G49 (G49.1) Tool length compensation cancel
- G50 Scaling cancel
- G51 Scaling
- G53 Machine coordinate system selection
- G53.1 Tool axis direction control
- G65 Custom macro call
- G66 Custom macro modal call
- G67 Custom macro modal call cancel
- G68 Coordinate system rotation/three-dimensional coordinate conversion
- G68.2 Tilted working plane command
- G73, G74, G76, G80 to G89 Hole machining canned cycle

Milling

- G45 Tool offset expansion
- G46 Tool offset reduction
- G47 Tool offset expansion by factor of 2
- G48 Tool offset reduction by factor of 2
- G50.1 Programmable mirror image cancel
- G51.1 Programmable mirror image
- G69 Coordinate system rotation/three-dimensional coordinate conversion/tilted working plane command cancel
- G90 Absolute programming
- G91 Incremental programming
- G94 Feed per minute
- G95 Feed per revolution
- G98 Canned cycle initial level return
- G99 Canned cycle R point level return

Turning

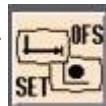
- G69.1 Coordinate system rotation/three-dimensional coordinate conversion/tilted working plane command cancel
- G90 Absolute programming (for G code system B and C)
- G91 Incremental programming (for G code system B and C)
- G94 Feed per minute (for G code system B and C)
- G95 Feed per revolution (for G code system B and C)
- G98 Canned cycle initial level return (for G code system B and C)
- G99 Canned cycle R point level return (for G code system B and C)

2.3 Setting Operations

A workpiece setting error is set on the workpiece setting error setting screen.

The workpiece setting error setting screen is displayed according to the procedure below.

1. Press the function key SETTING/OFFSET



2. Press the chapter selection soft key [WORKPIECE PLACEMENT ERROR]. The workpiece setting error setting screen is displayed.

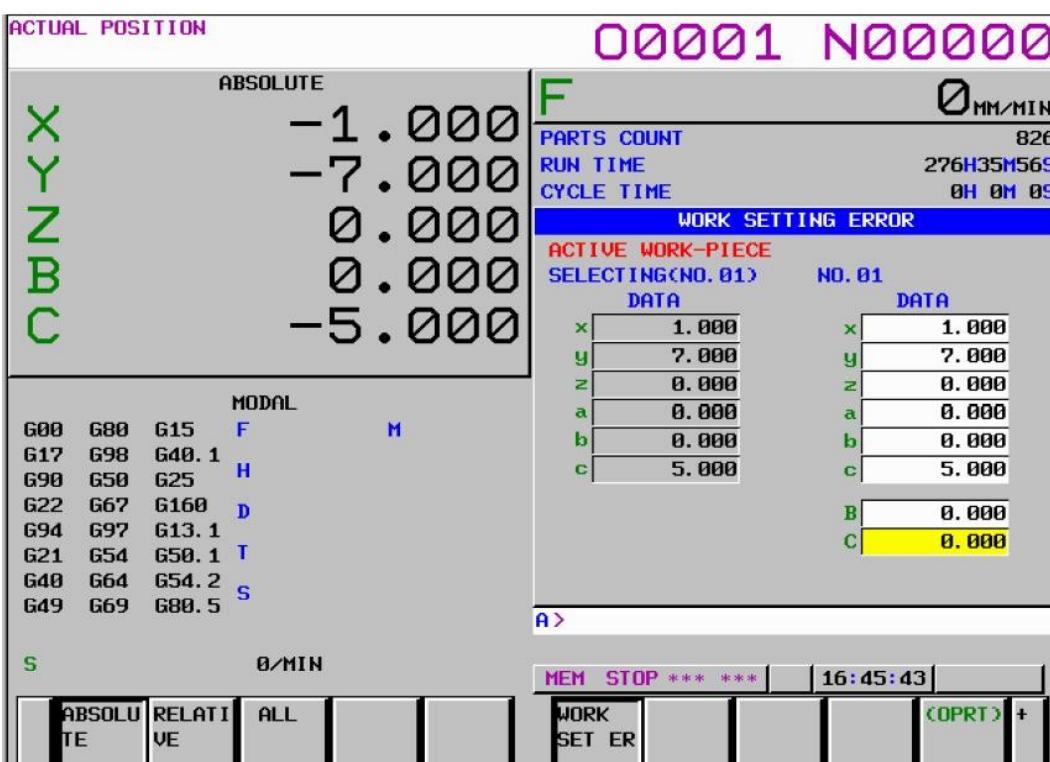


Figure 2.4: Workpiece setting error setting screen

Up to seven sets (No. 01 to No. 07) of different workpiece setting errors can be set. Moreover, a common error (No. 00 (COMMON)) to be added to the seven sets can be set.

In the fields of x, y, and z, set X direction error Δx , Y direction error Δy , and Z direction error Δz , respectively. In the fields of a, b, and c, set rotation direction error Δa , rotation direction error Δb , and rotation direction error Δc , respectively.

In the lower fields (B and C on the screen), set table rotation axis position 1 and table rotation axis position 2.

In the axis name fields, the rotation axis names set in parameter No. 19681 and No. 19686 are displayed.

Set table rotation axis position 1 and table rotation axis position 2 for table rotation axes. Table rotation axis position 1 and table rotation axis position 2 need not be set for axes (including hypothetical axes) other than the table rotation axes, so no setting item is displayed for those axes.

The necessary conditions to display the setting item of table rotation axis position are the following;

- To set the controlled-axis numbers for the rotation axes correctly In the case that the parameter No. 19680=12

(table rotation type), parameters No.19681 and No.19686 must be set correctly. In the case that the parameter No.19680=21 (composite type), parameter No.19686 must be set correctly.

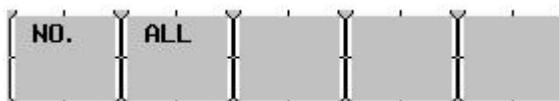
- To set the parameters No.1006#0=1 (rotation axis) as for the axes specified by the parameters No.19681 and No.19686.

In data setting, the following soft keys can be used.



When you press [NO. SRH] after entering a workpiece setting error number to be displayed, the screen for setting the desired workpiece setting error is displayed.

- When you enter a numeric value then press [+INPUT], the value is added.
- When you enter a numeric value then press [INPUT], the value is set.
- When you press [CLEAR], the following soft keys are displayed:



- When you press [NO.] after entering a workpiece setting error number to be deleted, the data of the workpiece setting error number is deleted.
- When you press [ALL], the data of all workpiece setting error numbers is deleted.

2.3.1 Workpiece setting error of No. 00 (COMMON)

Each of the values set in No. 00 (COMMON) is added to the corresponding value of each of No. 01 through No. 07.

Note that rotation direction errors for No.00 (COMMON) are not available.

Example:

Suppose that workpiece setting errors are set as follows:

No. 00 (COMMON)	No. 01	No. 02
X 10.000	x 0.500	x 0.800
Y 0.000	y 0.000	y 0.000
Z 0.000	z 0.000	z 0.000
	a 1.500	a 1.800
	b 0.000	b 0.000
	c 0.000	c 0.000

When the workpiece setting error of No. 01 is selected, workpiece setting error compensation is performed based on the following:

$$\Delta x = 10.000 + 0.500 = 10.500$$

$$\Delta a = 1.500$$

When the workpiece setting error of No. 02 is selected, workpiece setting error compensation is performed based on the following:

$$\Delta x = 10.000 + 0.800 = 10.800$$

$$\Delta a = 1.800$$

INITIAL WORKING PLANE

2.3.1 Workpiece setting error of No. 00 (COMMON)

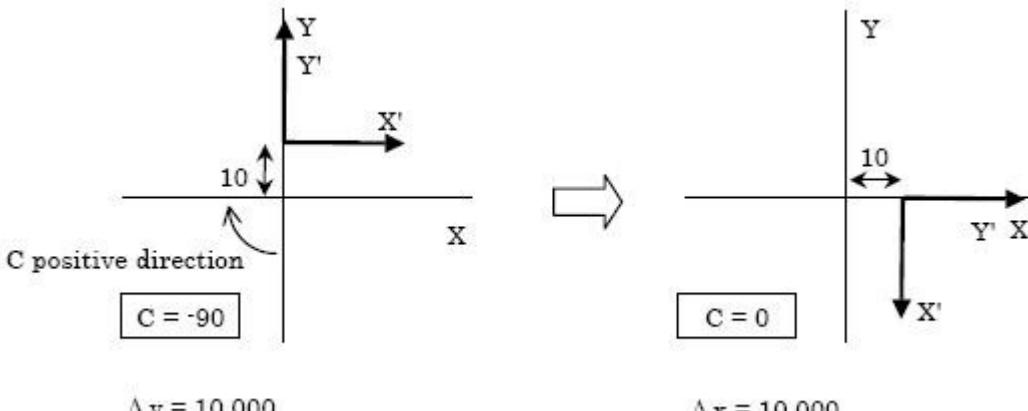
If the setting of table rotation axis position 1/table rotation axis position 2 differs between the workpiece setting error of No. 00 and each workpiece setting error when a 5-axis machine (machine of table rotation type or composite type) with table rotation axes is used, no simple additions are made. Instead, before additions are made, conversion is made to values based on the machine coordinate on a table rotation axis being 0.

Example:

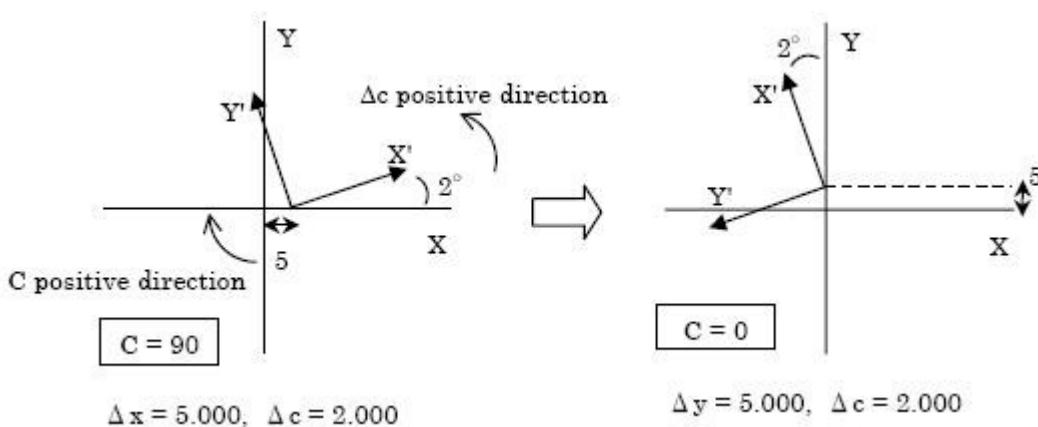
Suppose that workpiece setting errors are set as follows on a 5-axis machine with table rotation axis C about the Z-axis.

No. 00 (COMMON)	No. 01
X 0.000	x 5.000
y 10.000	y 0.000
z 0.000	z 0.000
	a 0.000
	b 0.000
	c 2.000
C -90.000	C 90.000

First, the error values of No. 00 are converted to those based on C = 0.000.



Next, the error values of No. 01 are converted to those based on C = 0.000.

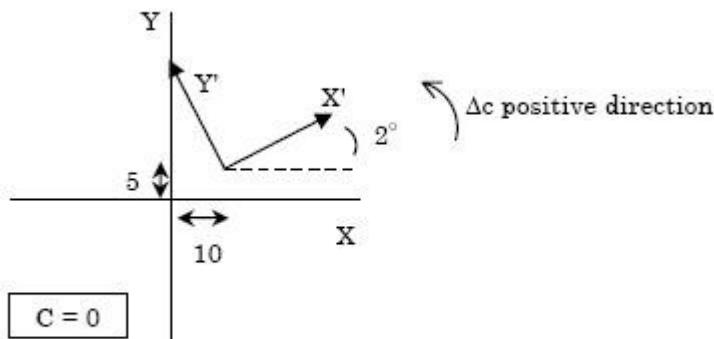


When the workpiece setting error of No. 01 is selected, workpiece setting error compensation is performed as follows based on the converted error values based on C = 0.000:

$$\Delta x = 10.000 + 0.000 = 10.000$$

$$\Delta y = 0.000 + 5.000 = 5.000$$

$$\Delta c = 2.000$$



$$\Delta x = 10.000, \Delta y = 5.000, \Delta c = 2.000$$

2.3.2 Least input increment of data and valid data range

[About errors Δx , Δy , and Δz]

The unit of data is the unit of input, and follows the least input increment for the reference axis specified by parameter No. 1031.

For metric input

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (mm)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

For inch input

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Maximum settable value (inch)	±99,999.99	±99,999.9999	±9,999.99999	±999.999999	±99.9999999

[About errors Δa , Δb , and Δc]

With parameter No. 11201, the number of decimal places of the least input increment can be specified.

Parameter No. 11201	1	2	3	4
Least input increment (deg)	0.1	0.01	0.001	0.0001
Maximum settable value (deg)	±99,999,999.9	±9999,999.99	±999,999.999	±99,999.9999

Parameter No. 11201	5	6	7	8
---------------------	---	---	---	---

Least input increment (deg)	0.00001	0.000001	0.0000001	0.00000001
Maximum settable value (deg)	±9,999.99999	±999.999999	±99.9999999	±9.99999999

Note, however, that a value from 1 to 8 can be specified in parameter No. 11201. If a value not within the specifiable range is specified in parameter No. 11201, the least input increment of the reference axis is followed.

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

[About table rotation axis position 1 and table rotation axis position 2] The unit of data follows the least input increment of each corresponding rotation axis.

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

2.3.3 "Active workpiece setting error"

In the workpiece setting error compensation mode, "Active workpiece setting error" on the workpiece setting error setting screen displays the currently selected workpiece setting error number and workpiece setting error (sum of the data of the currently selected workpiece setting error number and the data of No. 00).

If the workpiece setting error compensation mode is not set, "MODE OFF" is indicated, and 0 is displayed in each data field.

If a 5-axis machine (machine of table rotation type or composite type) with table rotation axes is used, and a nonzero value is set for table rotation axis position 1/table rotation axis position 2, error values obtained by conversion based on the machine coordinates on the table rotation axes being 0 are displayed.

2.3.4 Setting of workpiece setting errors with custom macro variables

By using custom macro variables #26000 to #26077, workpiece setting errors can be read and written. (The custom macro option is required.)

The numbers of macro variables correspond to Errors as follows :

	error No.00 (COMMON)	error No.01	error No.02	error No.03	error No.04	No.05 error	error No.06	error No.07
X direction error Δx	#26000	#26010	#26020	#26030	#26040	#26050	#26060	#26070

Y direction error Δy	#26001	#26011	#26021	#26031	#26041	#26051	#26061	#26071
Z direction error Δz	#26002	#26012	#26022	#26032	#26042	#26052	#26062	#26072
Rotation direction error Δa	-	#26013	#26023	#26033	#26043	#26053	#26063	#26073
Rotation direction error Δb	-	#26014	#26024	#26034	#26044	#26054	#26064	#26074
Rotation direction error Δc	-	#26015	#26025	#26035	#26045	#26055	#26065	#26075
Table rotation axis position 1	#26006	#26016	#26026	#26036	#26046	#26056	#26066	#26076
Table rotation axis position 2	#26007	#26017	#26027	#26037	#26047	#26057	#26067	#26077

2.3.5 Workpiece setting error compensation mode

By specifying G54.4 Pn (n: 1 to 7), the workpiece setting error compensation mode can be set.

With Pn, select a workpiece setting error from No. 01 to No. 07. In the workpiece setting error compensation mode, the program is executed in a "workpiece setting coordinate system" defined by shifting the workpiece coordinate system.

G54.4 is a modal G code that belongs to group 33 with the M series or group 26 with the T series.

By specifying G54.4 P0, the workpiece setting error compensation mode is canceled.

In a block for starting workpiece setting error compensation, the machine is not moved, but the absolute coordinates are changed by the error (Note 6). So, the next absolute command makes a movement to a specified position in the workpiece setting coordinate system. This means that an absolute command is needed after a block for starting workpiece setting error compensation.

NOTE

1. Be sure to specify Pn in a block specifying G54.4. If P is not specified, or a number (n) not within the specifiable range is specified after P, alarm PS0437 is issued.
2. Specify G54.4 singly. If another G code or axis command is specified together, alarm PS0437 is issued.
3. Workpiece setting error compensation is valid for movement in automatic operation.
4. Any workpiece setting error between starting blocks remains valid until the workpiece setting error compensation mode is canceled.
5. Workpiece setting error compensation may not be specified doubly. If G54.4 Pn (n ≠ 0) is specified in the workpiece setting error compensation mode, alarm PS0437 is issued.
6. When each of the following two angles is less than the parameter No.11204, the absolute coordinates are not moved in a block for starting workpiece setting error compensation.
 (1) The angle between the singular posture and the tool of the current absolute coordinates.
 (2) The angle between the singular posture and the tool of the absolute coordinates moved by the error.

2.3.6 Tool direction compensation on a 5-axis machine

With a 5-axis machine, tool direction compensation can be performed by setting bit 0 (RCM) of parameter No. 11200 to 1. This means that rotation axis position compensation is performed to direct the tool as programmed relative to the workpiece.

For tool direction compensation, the following parameters must be set:

Parameter No.	Description
19680	Type of mechanical section
19681	Controlled axis number of the first rotation axis
19682	Axis direction of the first rotation axis
19683	Inclination angle when the first rotation axis is inclined
19684	Rotation direction of the first rotation axis
19685	Rotation angle when the first rotation axis is a hypothetical axis
19686	Controlled axis number of the second rotation axis
19687	Axis direction of the second rotation axis
19688	Inclination angle when the second rotation axis is inclined
19689	Rotation direction of the second rotation axis
19690	Rotation angle when the second rotation axis is a hypothetical axis
19696#0	Whether the first rotation axis is an ordinary rotation axis/hypothetical axis
19696#1	Whether the second rotation axis is an ordinary rotation axis/hypothetical axis
19697	Tool axis direction
19698	Reference angle RA
19699	Reference angle RB
19700	Rotary table position (X-axis of the basic three axes)
19701	Rotary table position (Y-axis of the basic three axes)
19702	Rotary table position (Z-axis of the basic three axes)
19703	Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)
19704	Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)
19705	Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)

In the case that bit 0 (RCM) of parameter No. 11200 is 1, Workpiece setting error compensation must be commanded after AICC(AI contouring control) is already active.(G05.1 Q1 is commanded or the parameter SHP(No.1604#0) is 1.) And, the following parameters for Acceleration/Deceleration are needed.

Parameter No.	Description

1432	Maximum cutting feedrate in the acceleration/deceleration before interpolation mode
1660	Maximum allowable acceleration rate in acceleration/deceleration before interpolation
1671	Maximum allowable acceleration rate for rapid traverse in acceleration/deceleration before interpolation
1672	Acceleration/deceleration change time for rapid traverse in bell-shaped acceleration/deceleration before interpolation
1737	Allowable acceleration rate for each axis when the deceleration function based on acceleration/deceleration under AI contour control is used
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode
1772	Acceleration/deceleration change time in bell-shaped acceleration/deceleration before interpolation
1783	Allowable speed difference in speed determination based on a corner speed difference

NOTE

1. If any of the parameters above is set incorrectly when bit 0 (RCM) of parameter No. **11200** is set to 1, alarm P/S0438 is issued.
2. Depending on the machine configuration, it may be physically impossible to orient the tool in the compensation direction. In such a case, alarm DS0030 is issued.
3. In a block for starting workpiece setting error compensation, the absolute coordinate on a rotation axis is changed considering the workpiece setting error. At this time, depending on the machine configuration, a rotation axis for orienting the tool in the tool direction as viewed from the workpiece setting coordinate system may be absent. In such a case, alarm PS0438 is issued.
4. The option for AI contour control I or AI contour control II is required. Moreover, be sure to set the following parameters:
 - (1) Bit 1 (LRP) of parameter No. **1401=1**: Linear rapid traverse
 - (2) Bit 5 (FRP) of parameter No. 19501=1: Uses acceleration/deceleration before interpolation for rapid traverse.
 - (3) Parameter No. **1671**: Maximum allowable acceleration rate for rapid traverse in acceleration/deceleration before interpolation
 - (4) Parameter No. **1660**: Maximum allowable acceleration rate in acceleration/deceleration before interpolation If the parameters above are not set, alarm P/S0438 is issued.
5. When TCP (Tool center point control) or G53.1 of TWP (Tilted working plane command) is used during Workpiece setting error compensation, Tool direction compensation is needed (Parameter RCM(No.**11200#0**) is
 1. Otherwise, in the case of TCP, the alarm PS5421, in the case of G53.1 of TWP, the alarm PS5458 occurs.

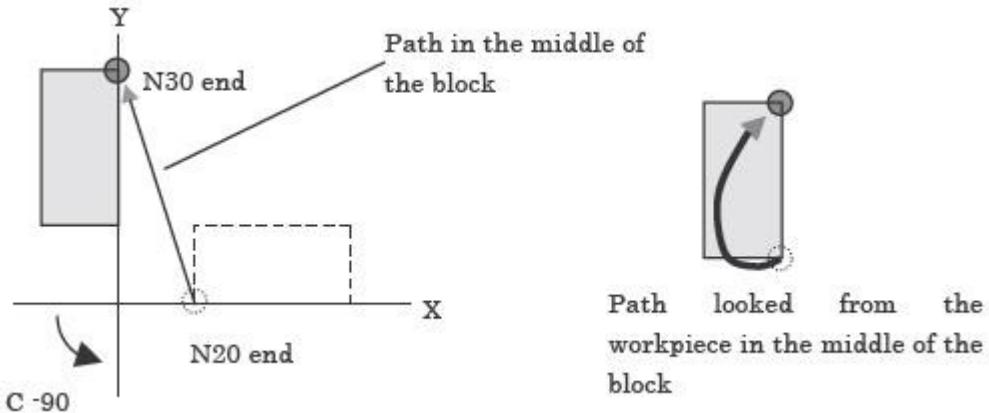
If linear interpolation or circular interpolation is specified on a machine of table rotation type or composite type, linear interpolation or circular interpolation is performed as viewed from the workpiece on the table.

Example:

The machine is a table rotation type with A axis(master axis) around X axis and C axis (slave axis) around Z axis (when A=0).

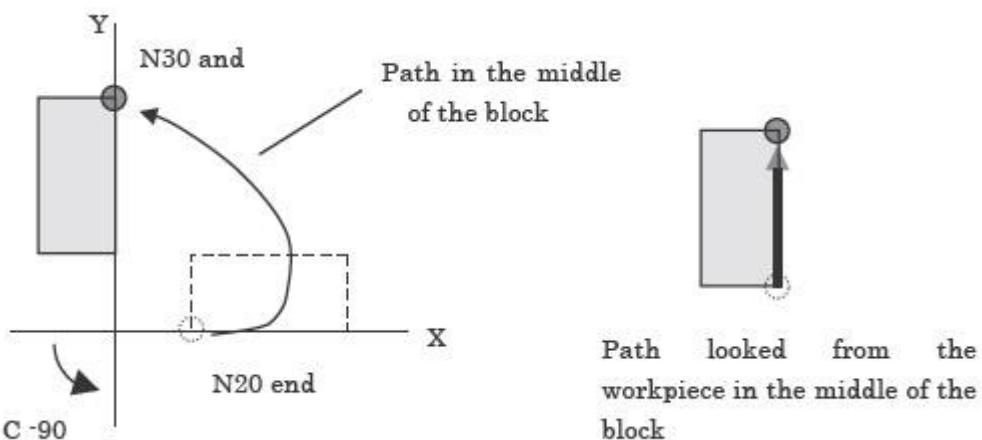
Firstly, suppose that Workpiece setting error is 0 and the following program is commanded.

O1	
N10 G55	(G55 is X50.0 Y0.0 Z0.0.)
N20 X0 Y0 Z0 A0 C0	
N30 G01 X - 50.0 Y150.0 C - 90.0 F100.0	



Next, suppose that Workpiece setting error has $\Delta c=5.0$, which is the error around Z axis, and N15,N16 are added to O2 as follows :

O2	
N10 G55	
N15 G05.1 Q1	
N16 G54.4 P1	
N20 X0 Y0 Z0 A0 C0	
N30 G01 X - 50.0 Y150.0 C - 90.0 F100.0	



The path looked from the workpiece in the middle of the block is linear.

2.3.7 "Rotation axis closer to the tool" and "rotation axis closer to the workpiece" on a 5-axis machine

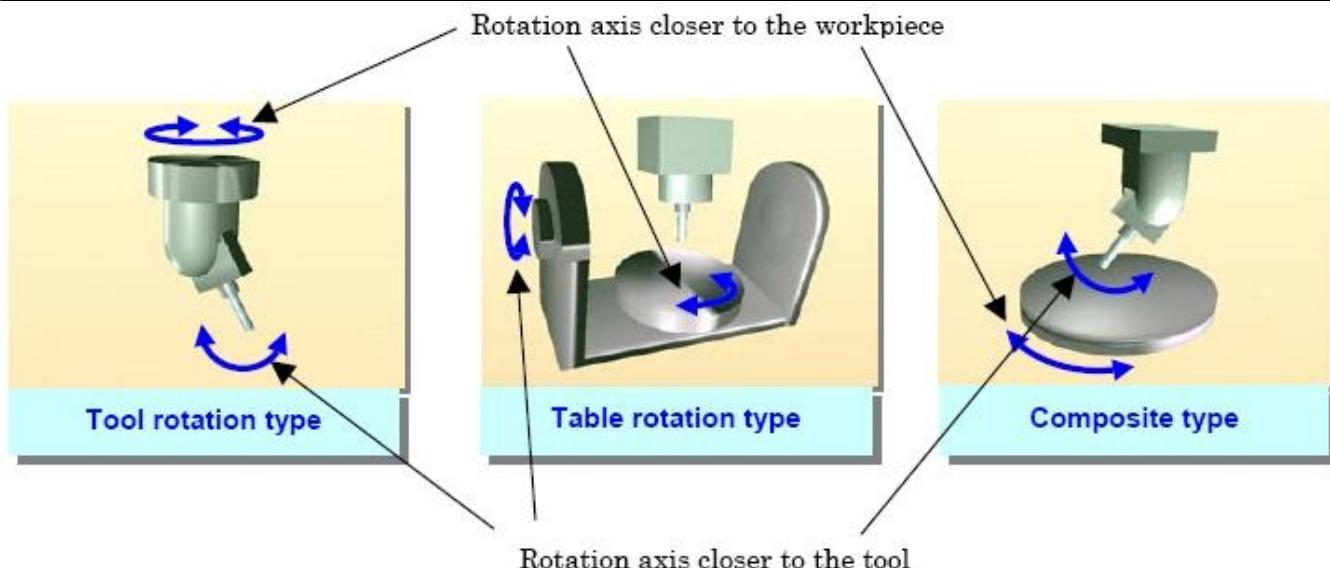
When tool direction compensation is performed on a 5-axis machine, a singular point and singular point posture need to be considered. Here, a "rotation axis closer to the tool" and "rotation axis closer to the workpiece", which are used in a description of a singular point and singular point posture provided later, are explained.

On a 5-axis machine with two rotation axes, one rotation axis functions to tilt the tool toward the workpiece. This rotation axis is referred to as a "rotation axis closer to the tool".

The other rotation axis is referred to as a "rotation axis closer to the workpiece". Depending on the type of mechanical section, the rotation axis closer to the tool and rotation axis closer to the workpiece are determined as indicated in Table 1.

Table 1: "Rotation axis closer to the tool" and "rotation axis closer to the workpiece"

Type of mechanical section (No. 19680)	Rotation axis closer to the tool	Rotation axis closer to the workpiece
Tool rotation type (2)	Slave axis	Master axis
Table rotation type (12)	Master axis	Slave axis
Composite type (21)	Tool rotation axis	Table rotation axis



2.3.8 Singular point and singular point posture on a 5-axis machine

A tool posture is uniquely determined when the angles of the two rotation axes are determined. Usually, however, a combination of the angles of the two rotation axes for achieving a certain tool posture is not determined uniquely.

In particular, such a tool posture that the angle of the rotation axis closer to the tool is arbitrary is referred to as a "singular point posture". Moreover, such an angle of the rotation axis closer to the tool that the tool posture becomes a singular point posture is referred to as a "singular point (or singular point angle)". When the angle of the rotation axis closer to the tool is a singular point, the center of the rotation axis closer to the workpiece and the tool posture (tool direction) are parallel with each other.

Example:

When the C-axis (about the Z-axis) is the master axis, the B-axis is the slave axis (about the Y-axis), and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type, the singular point is $B = \dots, 0 \text{ deg}, 180 \text{ deg}, \dots$. At this time, an arbitrary C-axis angle represents a singular point posture (in the same

direction as the Z-axis or in the direction opposite to the Z-axis).

If the angle of the B-axis is $B = 0$ (singular point) as shown in Figure 2.5, for example, an arbitrary C-axis angle represents the same tool posture (singular point posture).

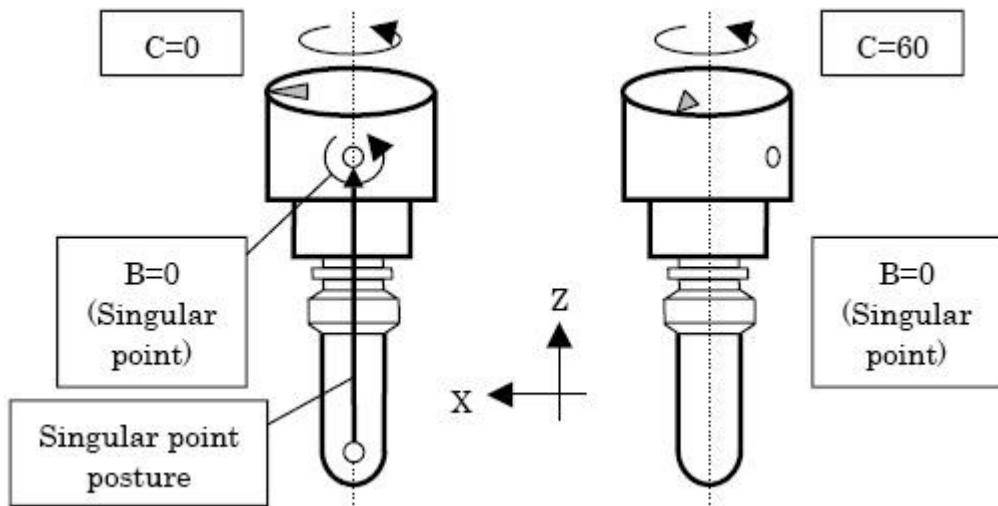


Figure 2.5: Singular point and singular point attitude

When the reference tool axis direction is inclined (parameter No. 19698, No. 19699), or the rotation axis is an angular axis (parameter No. 19682, No. 19683, No. 19687, No. 19688), for example, no singular point and singular point posture exist on some machines.

2.3.9 Conditions to decide that Tool is in singular posture

When the angle between the tool and the singular posture is less than the parameter No.11204, it is decided that the tool is in singular posture.

In the descriptions below, the description 'when the tool is in singular...' means that 'it is decided that the tool is in singular posture....'.

It is recommended that the following value or a little larger than that is set to the parameter No.11204.

- Max(Maximum federate (parameter No.1420) or Rapid traverse(parameter No.1432) of rotary axes) / 15000

2.3.10 Movement in the case that Tool direction is compensated

In the case that the tool direction is compensated(Parameter RCM (No.11200#0)=1), the compensation is performed in every interpolation, and the rotation axes may move to positions which are different from the commanded positions.

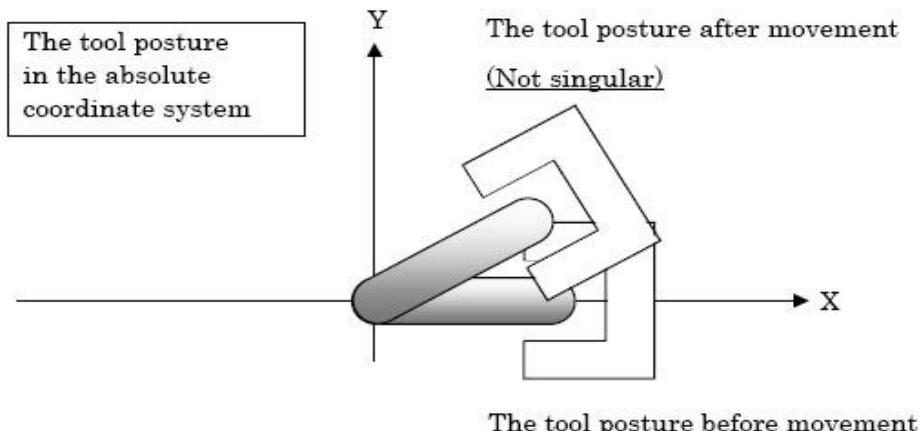
In general, there are two pairs of rotation axes positions in the region between 0 degree to 360 degree to turn the tool in a certain direction.

And in the case that the tool is in singular point posture, the position of the rotation axis closer to the workpiece is not determined uniquely.

How the positions are determined is described as follows :

1. In the case that the current machine position is singular and the position after movement in real time is also singular.
 - 1)-(1) When the absolute position after movement in real time is not singular, the rotation axis closer to the workpiece does not move.

Example: The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.



The tool posture before movement

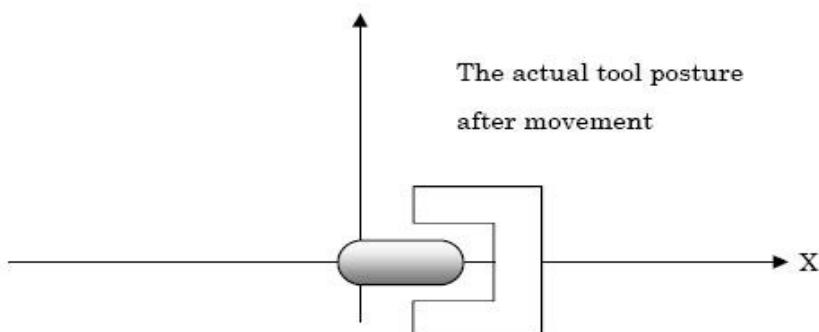
The calculated tool posture after movement

(Singular)

The tool posture before movement

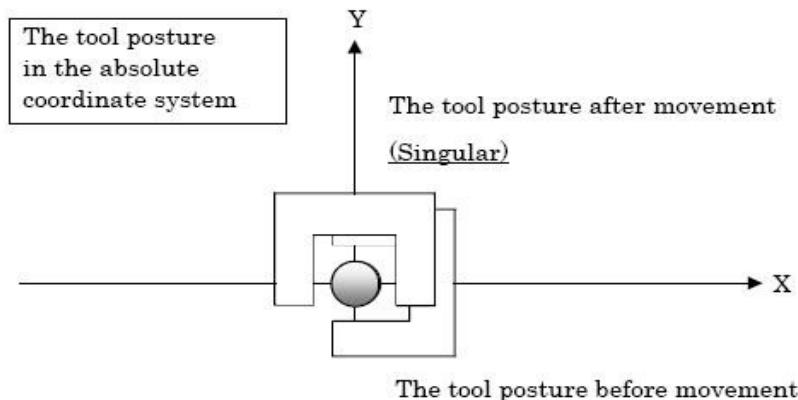
(Singular)

In this case, the rotation axis about the Z-axis (the rotation axis closer to the workpiece) does not move. (Only the rotation axis closer to the tool moves.)

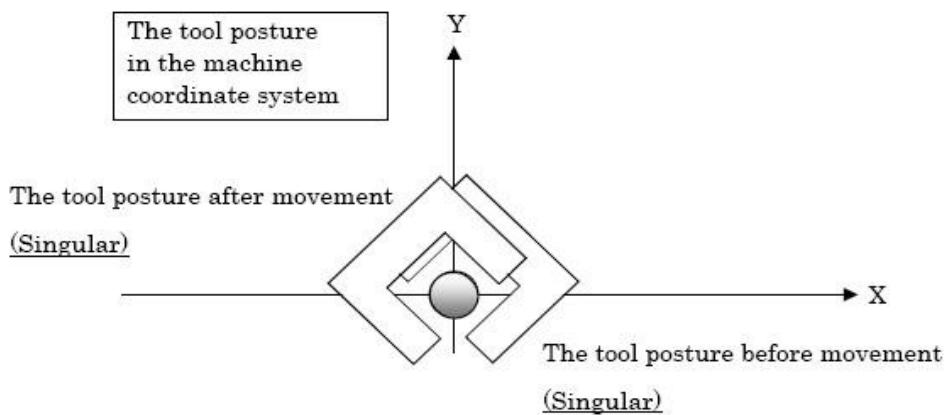


If the parameter No.11204 is set to the value except 0, the tools both before and after movement may be in singular point posture like the above example. In this case, the rotation axis closer to the workpiece does not move.

1)-(2) When the absolute position after movement in real time is singular, the rotation axis closer to the workpiece moves as commanded. Example: The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.



Assume that the rotation error around the Z-axis exists, and the tool posture after the compensation of tool direction becomes like following figure.

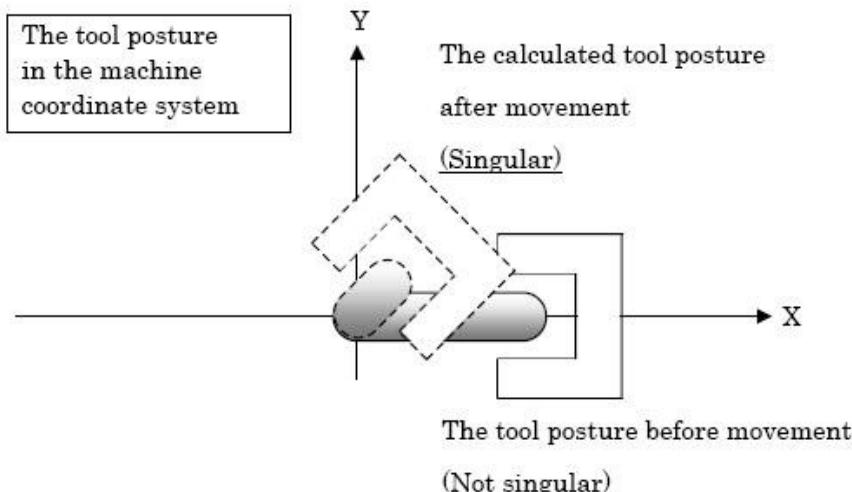


In this case, the rotation axis closer to the workpiece moves as commanded.

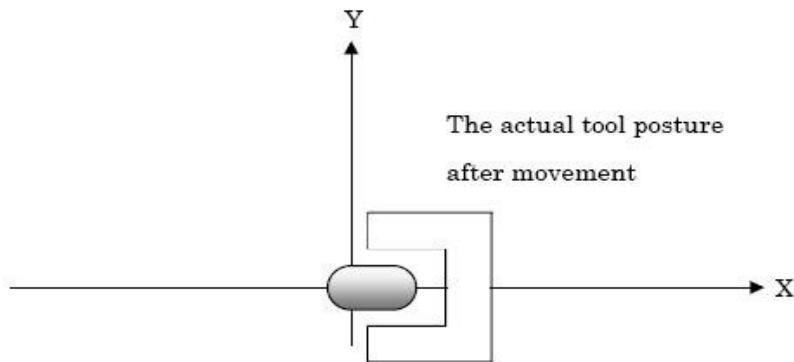
2. In the case that the current machine position is not singular and the position after movement in real time is singular.

: The rotation axis closer to workpiece does not move.

Example: The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type. Assume that the tool posture after the compensation of tool direction becomes like following figure.



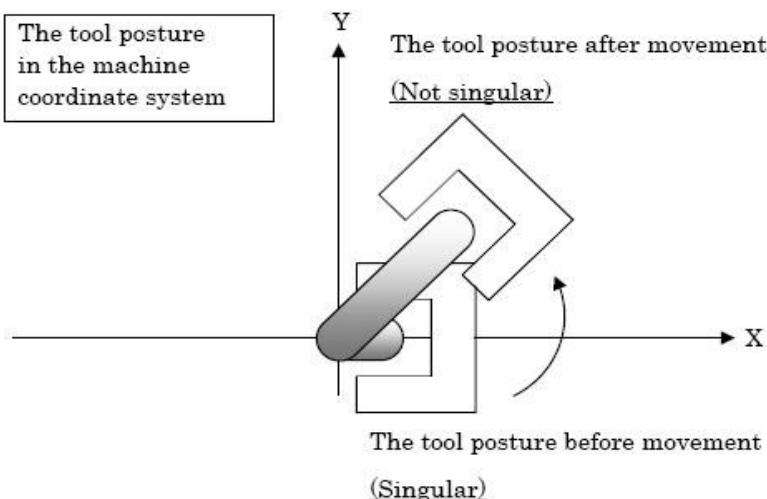
In this case, the rotation axis about the Z-axis (the rotation axis closer to the workpiece) does not move. (Only the rotation axis closer to the tool moves.)



3. In the case that the current machine position is singular and the position after movement in real time is not singular.
: In order to position the tool to the correct direction, there are two pairs of solutions of rotation axes angles within 0 – 360deg. One solution with which the rotation axis closer to workpiece moves shorter is selected. And, the rotation axes move to the positions of the selected solution.

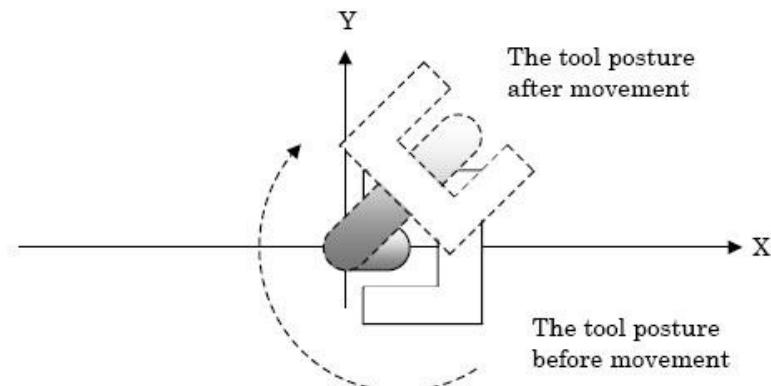
Example: The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.

Assume that the tool posture after the compensation of tool direction becomes like following figure.



In this case, the rotation axes move to the positions with which the rotation axis about the Z-axis (the rotation axis closer to workpiece) moves shorter. (The tool moves in the direction of the arrow in the above figure.)

The tool does not move in the direction of the arrow in the following figure.



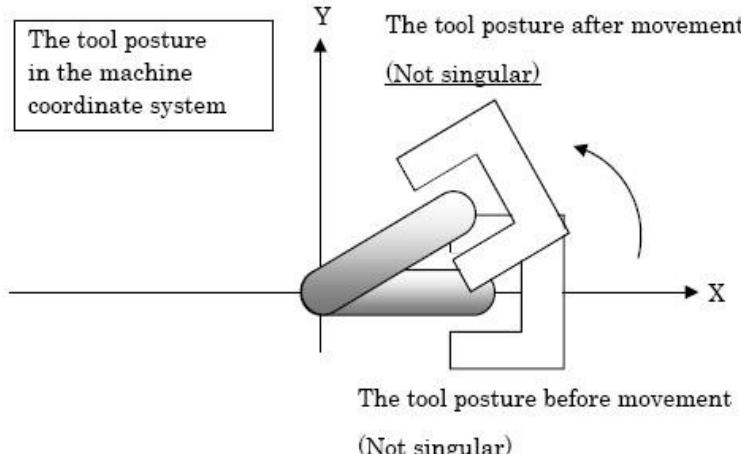
4. In the case that the current machine position is not singular and the position after movement in real time is not singular.

2.3.10 Movement in the case that Tool direction is compensated

: In order to position the tool to the correct direction, there are two pairs of solutions of rotation axes angles within 0 – 360deg. One solution with which the rotation axes don't pass the singular position is selected. And, the rotation axes move to the positions of the selected solution.

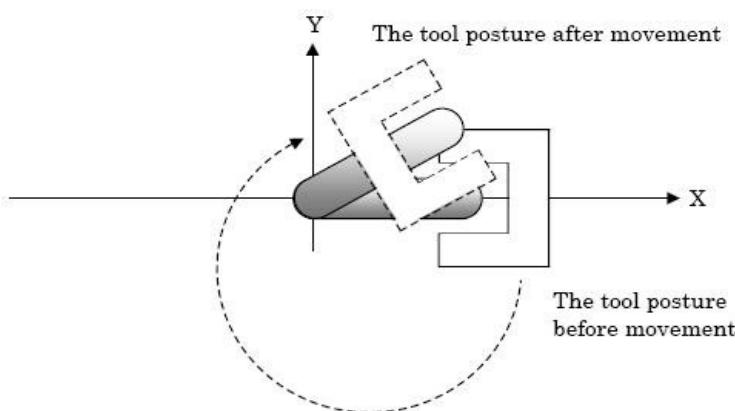
Example: The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.

Assume that the tool posture after the compensation of tool direction becomes like following figure.



In this case, the rotation axes move to the positions with which the rotation axes don't pass the singular position. (The tool moves in the direction of the arrow in the above figure.)

The tool does not move in the direction of the arrow in the following figure.



2.3.11 When the tool posture is closer to a singular point posture on a 5-axis machine

If tool direction compensation is performed on a 5-axis, and the tool posture gets closer to a singular point posture during execution of a block, the rotation axis closer to the workpiece may make a large motion.

And, when the parameter No.11204(Angle to decide singular posture) has a value, the rotary axis closer to the tool may rapidly move by the angle. Therefore, so large a value should not be set to the parameter No.11204.

2.3.12 Notes in the case that rotary axes have movable range

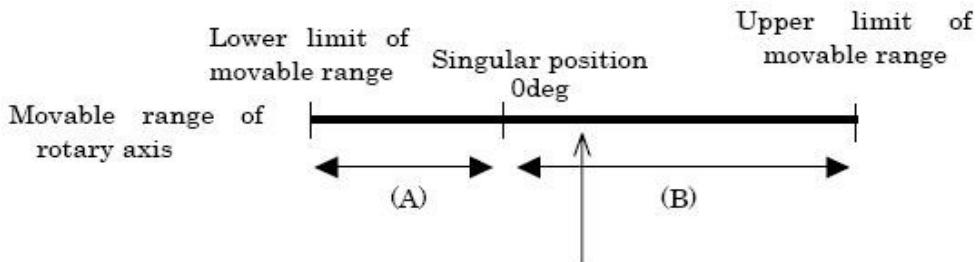
When the tool direction is compensated(Parameter RCM (No.11200#0)=1) on a 5-axis machine, there can be the case that the machine position does not pass the singular posture during a progress of work at all.

In this case, in accordance with 4) described above, the solution with which the rotary axes don't pass the singular position is always selected. Therefore, in this case, the movement direction(area) depends where the tool firstly exists in comparison with the singular position, because the tool position is always selected so that the rotary axes don't pass the singular position.

In the case that rotary axes have movable range and a singular position exists in that range, Workpiece setting

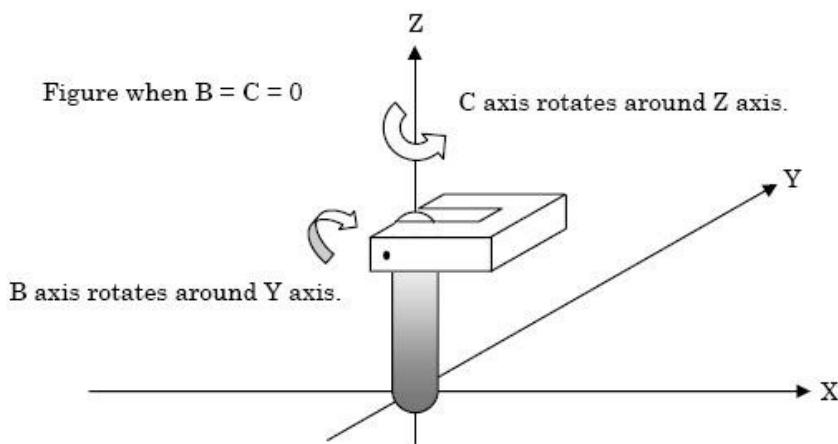
error compensation must be activated after the rotary axes have been moved to the range where the rotary axes should move, that is, the range (A) between the lower limit and the singular position or the range(B) between the upper limit and the singular position.

Generally speaking, if (A) and (B) are different, it is recommended that the larger one is selected.

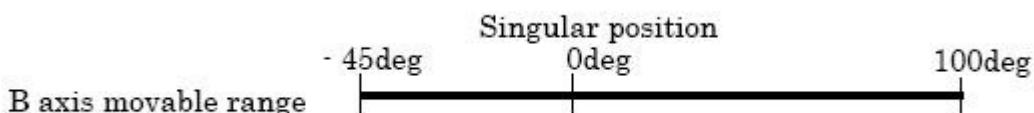


Workpiece setting error compensation must be activated after the rotary axes have been moved to the range where the rotary axes should move

The following is an example that the movement direction(area) depends on where the tool firstly exists in comparison with the singular position. The Master axis is C axis around Z axis, and the Slave axis is B axis around Y axis(when C=0). Tool direction is +Z when B=C=0. Here, suppose B axis' movable range is -45deg to +100deg.

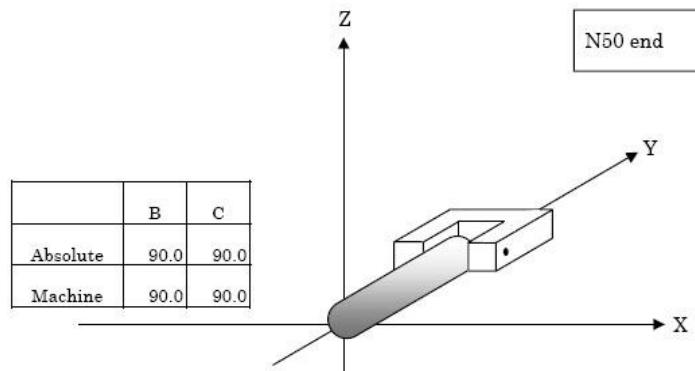
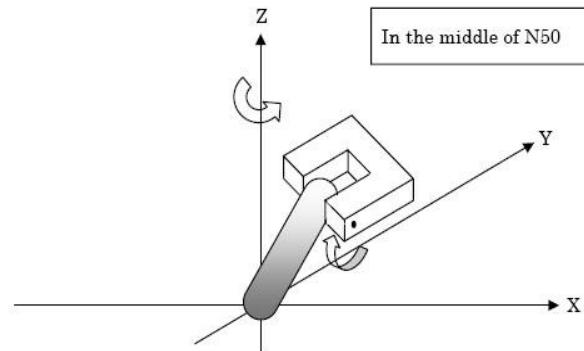
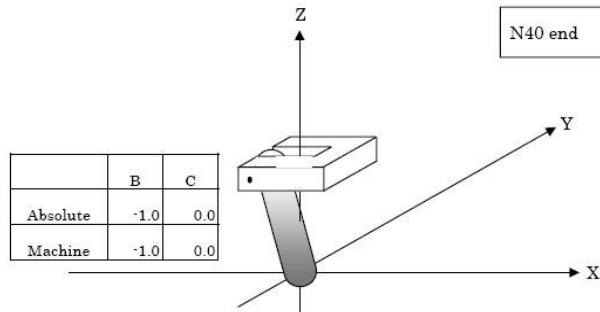


When $B=0$, the tool posture does not change even if C axis moves. That means the position $B=0$ is singular position.



Firstly, suppose the following program without Workpiece setting error.

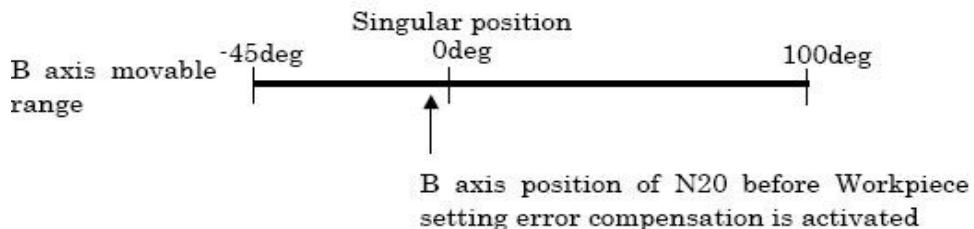
Program without Workpiece setting error

O1**N10 G5.1 Q1****N20 G90 G01 B-1.0 C0 F1000****N30 G43.4 H1****N40 X0 Y0 Z0****N50 B90.0 C90.0**

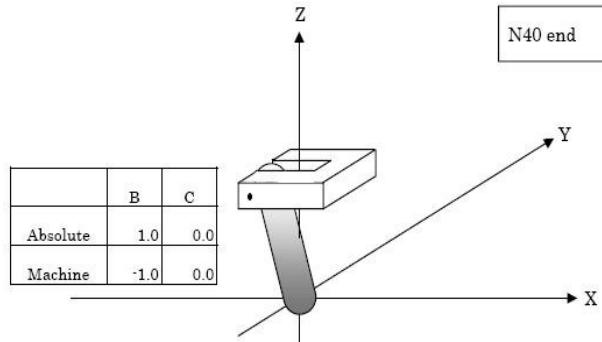
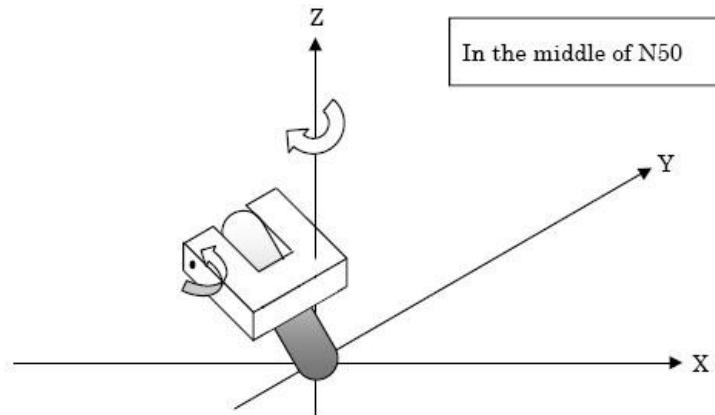
At N50, Machine position moves to B90.0 and C90.0, as commanded.

Next, suppose there is the error -2.0deg around Y axis and Workpiece setting error $\Delta b = -2.000$ is set, and the block N25 is added as follows :

On O2, B axis position is -1.0 before Workpiece setting error compensation is activated, which is between the lower limit of movable range and the singular position of B axis.

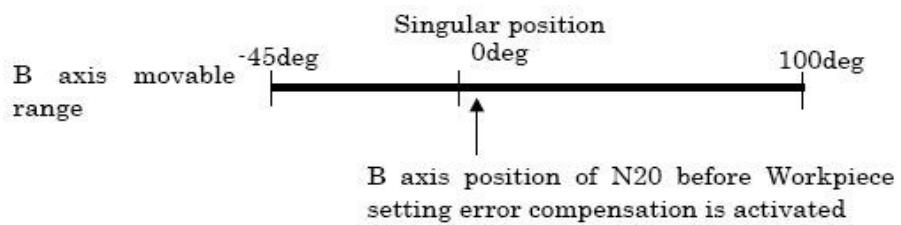
**O2****N10 G5.1 Q1****N20 G90 G01 B-1.0 C0 F1000 ;**

B axis machine position is between the lower limit and the singular position

N25 G54.4 P1**N30 G43.4 H1****N40 X0 Y0 Z0****N50 B90.0 C90.0**

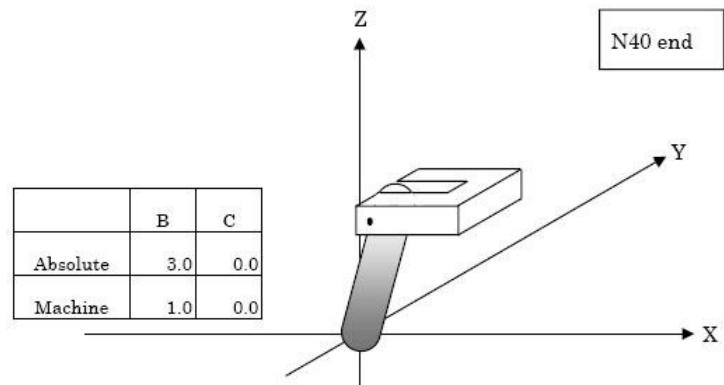
During N50, Machine position does not pass the singular position at all. So, B axis moves from the starting position(-1.0) to the position(B=90.0,C=90.0). As the result, B axis moves over the lower limit of B axis movable range.

Therefore, suppose the program O3 in which B axis position is changed to 1.0, which is between the singular position and the upper limit, before Workpiece setting error compensation is activated. Then, the range between the singular position and the upper limit of B axis movable range is the range where B axis should move.

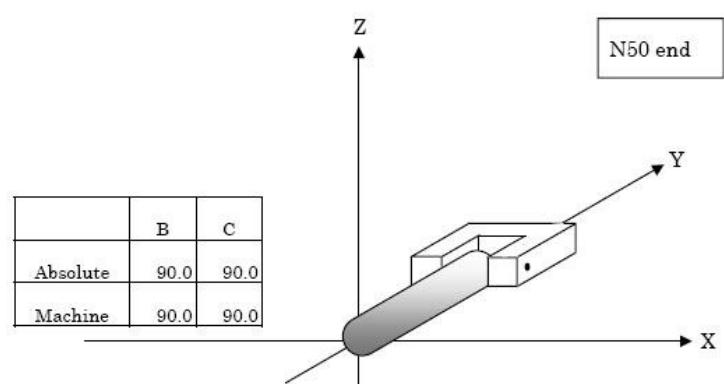
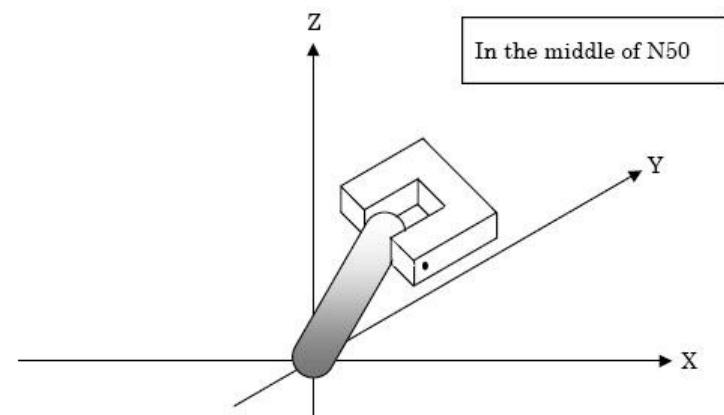


O3**N10 G5.1 Q1****N20 G90 G01 B1.0 C0 F1000 ;****N25 G54.4 P1****N30 G43.4 H1****N40 X0 Y0 Z0**

B axis machine position is between the upper limit and the singular position

**N50 B90.0 C90.0**

In the middle of N50



This time, machine position moves to B90.0,C90.0. As the result, B axis does not move over the lower limit of B axis movable range.

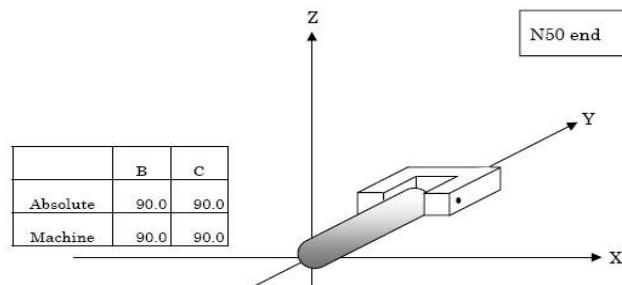
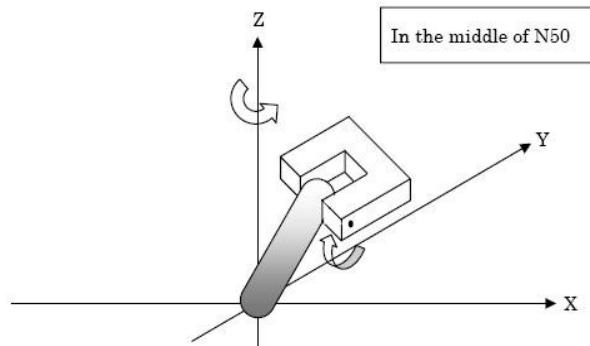
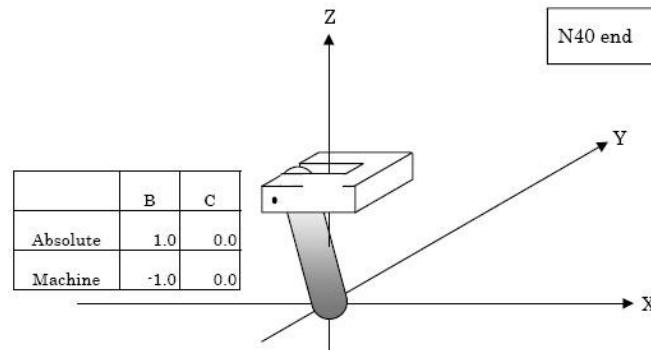
In O2, the case that B axis moves over the limit of movable range is the case that machine position does not make the singular point posture during N50.

When parameter No.11204 has 0, whether the tool posture is a singular point posture or not is judged strictly. So when the tool posture is almost a singular point posture, the tool posture is not regarded as a singular point posture.

When parameter No.11204 has a proper value and the tool posture is regarded as a singular point posture, there is the possibility that the position passes the singular position.

In the above example, when parameter No.11204 has 0.1, the machine position come to make the singular point posture during N50 in O2, and B axis passes the singular position and moves to B90.0,C90.

O2	O2 is executed with the parameter No.11204=0.1.
N10 G5.1 Q1	
N20 G90 G01 B-1.0 C0 F1000 ;	B axis machine position is between the lower limit and the singular position
N25 G54.4 P1	
N30 G43.4 H1	
N40 X0 Y0 Z0	
N50 B90.0 C90.0	



At N50, the machine position moves to B90.0,C90.0. As the result, B axis does not move over the lower limit of B axis movable range.

2.3.13 Absolute position display

Whether absolute coordinates in the workpiece setting error mode are to be displayed in the workpiece coordinate system or workpiece setting coordinate system can be chosen by using bit 6 (DAK) of parameter No. 3106.

2.3.14 Rapid traverse rate for hole machining in a hole machining canned cycle

Rapid traverse for hole machining in a hole machining canned cycle is performed according to the cutting feedrate specified in parameter No. 5412. If this parameter is set to 0, the dry run feedrate is used.

2.3.15 Custom macro variables

To system variables #5041 through #5048 (current position on each axis), the coordinates in the workpiece coordinate system are assigned.

2.3.16 Reset

The workpiece setting error compensation mode is canceled by resetting the CNC in the workpiece setting error compensation mode.

If bit 2 (D3R) of parameter No. 5400 is set to 1, however, the workpiece setting error compensation mode can be canceled by specifying only G54.4 P0. If this setting is made, the workpiece setting error compensation mode is not canceled even when the CNC is reset with a reset operation or the input signal ERS, ESP, or RRW from the PMC.

2.3.17 Rigid tapping

"Positional deviation Z" displayed on the spindle adjustment screen during rigid tapping indicates a value related to the long axis.

2.3.18 Range of rotary axes movements

In the case that Tilted working plane command or Type II of Tool center point control for 5-axis machining or Type II of Tool radius compensation for 5-axis machining is used, there are the parameters No.19741～No.19744 to limit the range of rotary axes movements. When tool direction is compensated (Parameter RCM (No.11200#0)=1), the rotary axes are compensated after the range of rotary axes movements are limited. Therefore, when the rotary axes are compensated, the rotary axes positions may be out of the range of rotary axes movements.

Therefore, in the case that the rotary axes have the range of movements, set the stored stroke limit check to prevent a movement over the range.

2.4 Examples

2.4.1 Example 1

O1 represents a program that cuts each side of a square.

O1 ;	
N10 G55 ;	Set coordinate system
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;	Move to initial position
N30 G01 G43 H01 Z40.0 F500. ;	Start tool length compensation H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 ;	Z-axis height of machining plane is 20.0.
N50 X150.0 ;	
N60 Y150.0 ;	
N70 X50.0 ;	
N80 Y50.0 ;	
N90 X0 Y0 Z40.0 ;	
N100 G49 Z300.0 ;	Cancel tool length compensation Move to initial position on Z-axis
N110 M30	

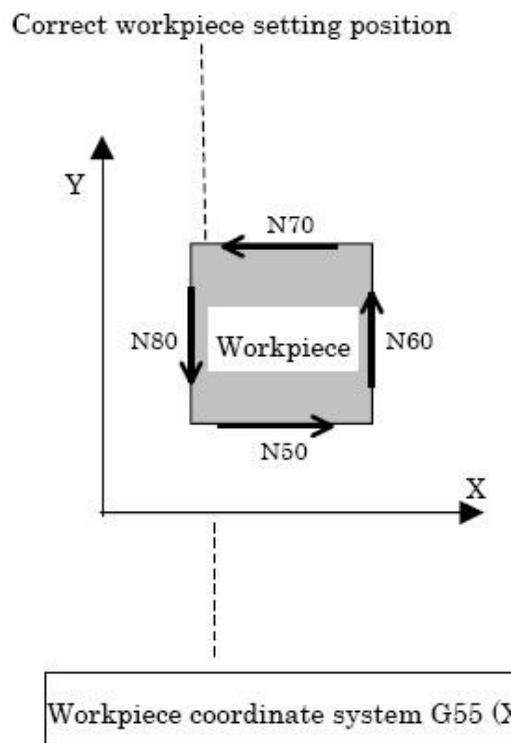


Figure 2.6: Operation without workpiece setting error

Suppose that the workpiece is displaced from the "correct workpiece setting position" as shown in Figure 2.7.

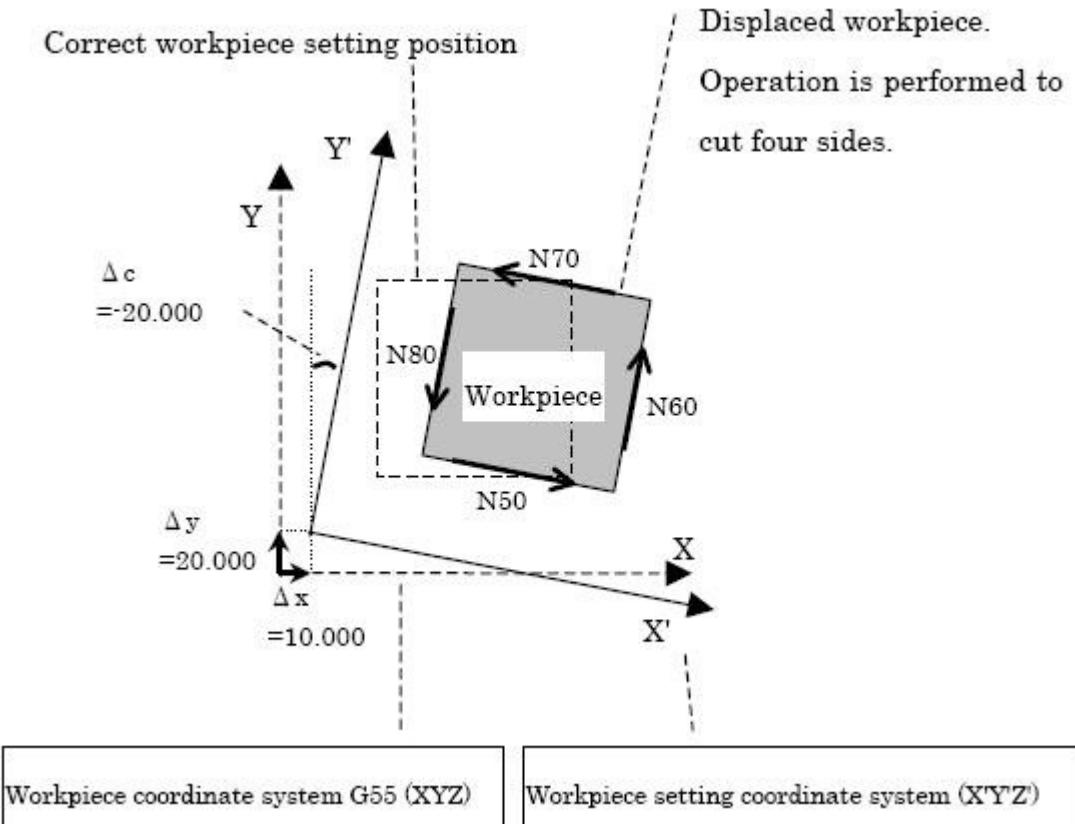


Figure 2.7: Operation with workpiece setting error

The workpiece coordinate system, when rotated by -20.000 deg about the Z-axis, shifted by 10.000 in the X direction, and shifted by 20.000 in the Y direction, is to match the workpiece setting coordinate system.

At this time, set the following workpiece setting error values: (Workpiece setting error No. 01)

x 10.000
y 20.000
z 0.000
a 0.000
b 0.000
c -20.000

To validate the workpiece setting error, add N15 and N115 to O1 as indicated below.

O1 ;	
N10 G55 ;	Set coordinate system
N15 G54.4 P1	Workpiece setting error compensation mode ON
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;	Move to initial position
N30 G01 G43 H01 Z40.0 F500. ;	Start tool length compensation H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 ;	Z-axis height of machining plane is 20.0.
N50 X150.0 ;	
N60 Y150.0 ;	
N70 X50.0 ;	
N80 Y50.0 ;	
N90 X0 Y0 Z40.0 ;	
N100 G49 Z300.0 ;	Cancel tool length compensation Move to initial position on Z-axis
N115 G54.4 P0 ;	Workpiece setting error compensation mode OFF
N110 M30	

When O1 is executed, the tool moves to cut each side of the displaced workpiece as shown by the solid lines in Figure 2.7.

2.4.2 Example 2

O2 is a program for cutting each side of a square by using tool center point control.

The machine is of tool rotation type, the C-axis is the master rotation axis and rotates about the Z-axis, and the B-axis is the slave axis and rotates about the Y axis. For cutting on the plane normal to the movement direction, the tool is tilted 45 deg relative to the +Z direction.

O2 ;	
N10 G55 ;	Set coordinate system
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;	Move to initial position
N30 G01 G43.4 H01 Z40.0 F500. ;	Start tool center point control H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 B45.0 C-90.0 ;	Z-axis height of machining plane is 20.0.
N50 X150.0 ;	
N60 C0.0 ;	
N70 Y150.0 ;	
N80 C90.0 ;	
N90 X50.0 ;	
N100 C180.0 ;	
N110 Y50.0 ;	
N120 X0 Y0 Z40.0 B0.0 C0.0 ;	
N130 G49 Z300.0 ;	Cancel tool length compensation Move to initial position on Z-axis

N140 M30;

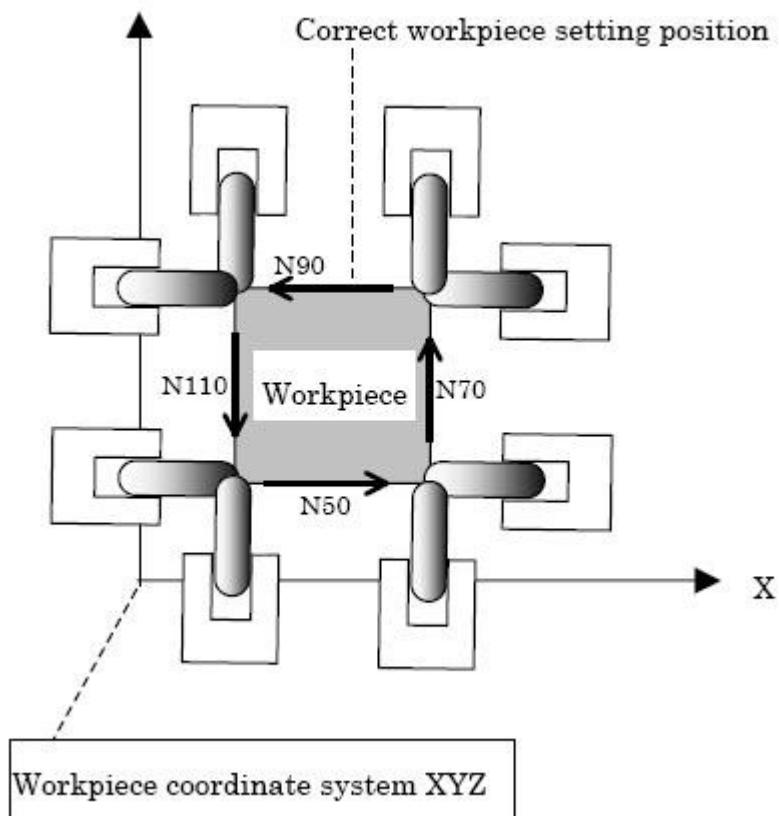


Figure 2.8: Operation of TCP without workpiece setting error

Suppose that the workpiece is displaced from the correct workpiece setting position as with example 1, and set a workpiece setting error in the same way as in Example 1.

Moreover, add N15,N16,N135 and N136 to O2 as indicated below to validate the workpiece setting error as in the case of Example 1. (AI contour control must be active because tool direction compensation is performed.)

O2 ;	
N10 G55 ;	Set coordinate system
N15 G05.1 Q1 ;	AI contour control ON
N16 G54.4 P1 ;	Workpiece setting error compensation ON
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;	Move to initial position
N30 G01 G43.4 H01 Z40.0 F500. ;	Start tool center point control H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 B45.0 C-90.0 ;	Z-axis height of machining plane is 20.0.
N50 X150.0 ;	
N60 C0.0 ;	
N70 Y150.0 ;	
N80 C90.0 ;	
N90 X50.0 ;	
N100 C180.0 ;	

N110 Y50.0 ;	
N120 X0 Y0 Z40.0 B0.0 C0.0 ;	
N130 G49 Z300.0 ;	Cancel tool length compensation Move to initial position on Z-axis
N135 G54.4 P0 ;	Workpiece setting error compensation OFF
N136 G05.1 Q0	AI contour control OFF
N140 M30;	

Set bit 0 (RCM) of parameter No. 11200 to 1 to perform tool direction compensation, then execute O2. When the tool moves to cut each side of the workpiece with the tool tip, the tool is tilted 45 deg relative to the +Z direction on the plane normal to the movement direction as shown in Figure 2.9.

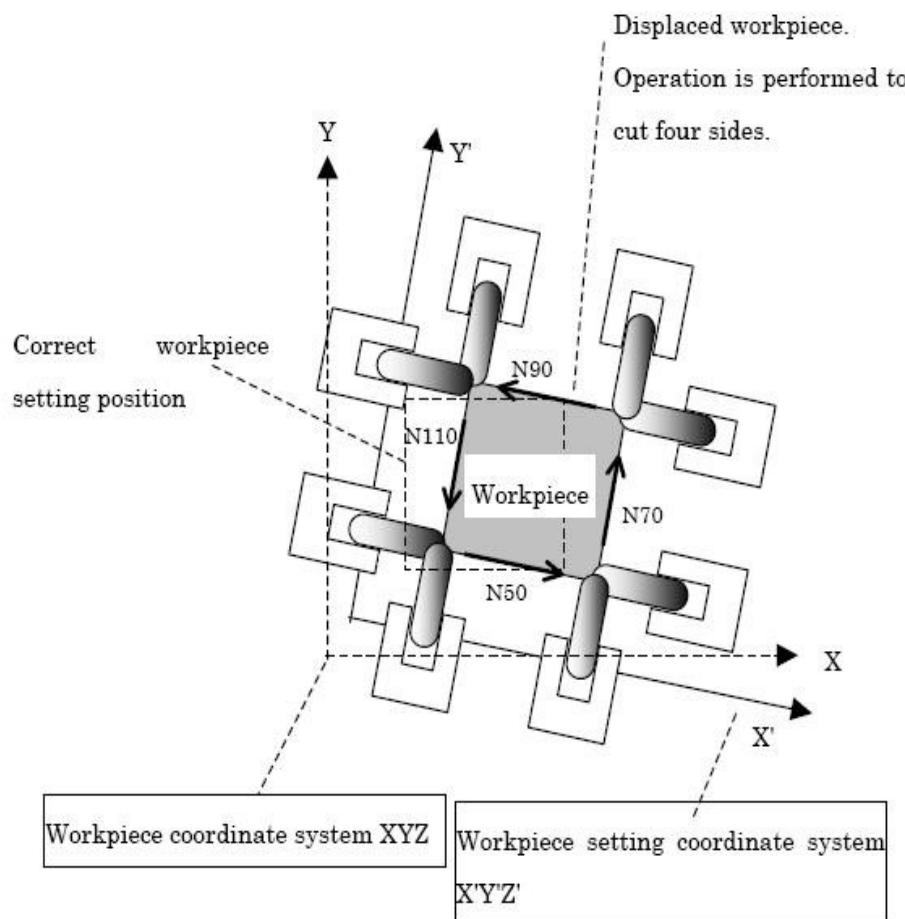


Figure 2.9: Operation of TCP with workpiece setting error

2.5 Restrictions

2.5.1 Modal G code that cannot be used

In a modal G code state listed below, workpiece setting error compensation can be specified.

If workpiece setting error compensation is specified in a modal state not listed below, alarm PS0439 is issued.

- G00 Positioning
- G01 Linear interpolation
- G13.1 Polar coordinate interpolation mode cancel

- G15 Polar coordinate command cancel
- G17 Plane selection (XY)
- G18 Plane selection (ZX)
- G19 Plane selection (YZ)
- G20 Inch input
- G21 Metric input
- G23 Stored stroke check function OFF
- G25 Spindle speed fluctuation detection OFF
- G40 Cutter compensation/tool-nose radius compensation/cutter compensation for
- 5-axis machining cancel
- G49 (G49.1) Tool length compensation cancel
- G50 Scaling cancel
- G50.1 Programmable mirror image cancel
- G50.2 Polygon turning cancel
- G54 to G59, G54.1 Workpiece coordinate system selection
- G64 Cutting mode
- G67 Custom macro modal call cancel
- G80 Canned cycle cancel
- G80.5 (G80.8) Electronic gear box synchronization cancel
- G97 Constant surface speed control cancel

Milling

- G40.1 Normal direction control cancel
- G54.2 P0 Rotary table dynamic fixture offset cancel
- G69 Coordinate system rotation/three-dimensional coordinate conversion/tilted working plane command cancel
- G90 Absolute programming
- G91 Incremental programming
- G94 Feed per minute
- G95 Feed per revolution
- G98 Canned cycle initial level return
- G99 Canned cycle R point level return

Turning

- G69 Mirror image for double turret OFF/balanced cutting mode cancel
- G69.1 Coordinate system rotation/three-dimensional coordinate conversion/tilted working plane command cancel
- G90 Absolute programming (for G code system B and C)
- G91 Incremental programming (for G code system B and C)
- G94 Feed per minute (for G code system B and C)
- G95 Feed per revolution (for G code system B and C)
- G98 Canned cycle initial level return (for G code system B and C)
- G99 Canned cycle R point level return (for G code system B and C)

2.5.2 General Restrictions

Manual intervention and handle interruption

Workpiece setting error compensation is not applicable to the amounts of manual intervention and handle interruption.

Positioning in the machine coordinate system

Workpiece setting error compensation is not applicable to positioning by a G code such as G28, G30, and G53 in the machine coordinate system.

Rapid traverse command

When using workpiece setting error compensation, specify linear rapid traverse (by setting set bit 1 (LRP) of parameter No. **1401** to 1).

Relationships with other modal commands

The commands for the functions listed below must be nested with a workpiece setting error compensation command and must be placed between G54.4 P_{_} and G54.4 P0.

Namely, specify G54.4 P_{_} when the mode of a desired function below is off. Next, turn on then off the mode of the function, then specify G54.4 P0.

- Cutter compensation (G40, G41, G41.2, G41.3, G41.4, G41.5, G41.6, G42, G42.2, G42.4, G42.5, G42.6)
- Tool length compensation (G43, G43.1, G43.4, G43.5, G44, G49)
- Tool offset expansion/reduction (G45, G46, G47, G48)
- Programmable mirror image (G50.1, G51.1)
- Scaling (G51)
- Coordinate system rotation/three-dimensional coordinate conversion/tilted working plane command (G68, G68.2)
- Canned cycle

PMC axis control

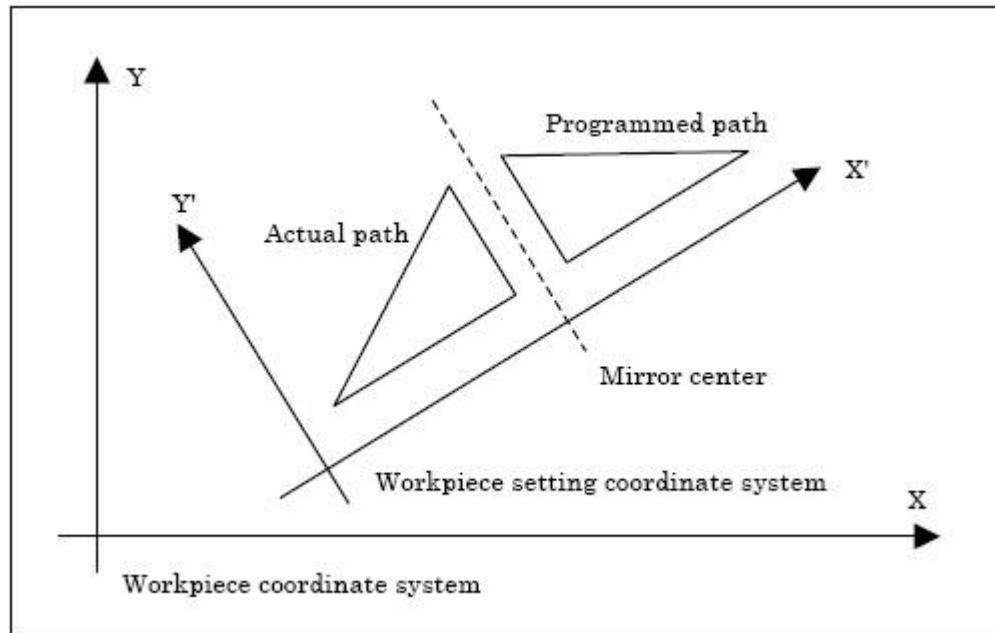
If PMC axis control is exercised in the workpiece setting error compensation mode, workpiece setting error compensation is not applied to movement based on PMC axis control.

Movement of start-up and cancel of Tool length offset

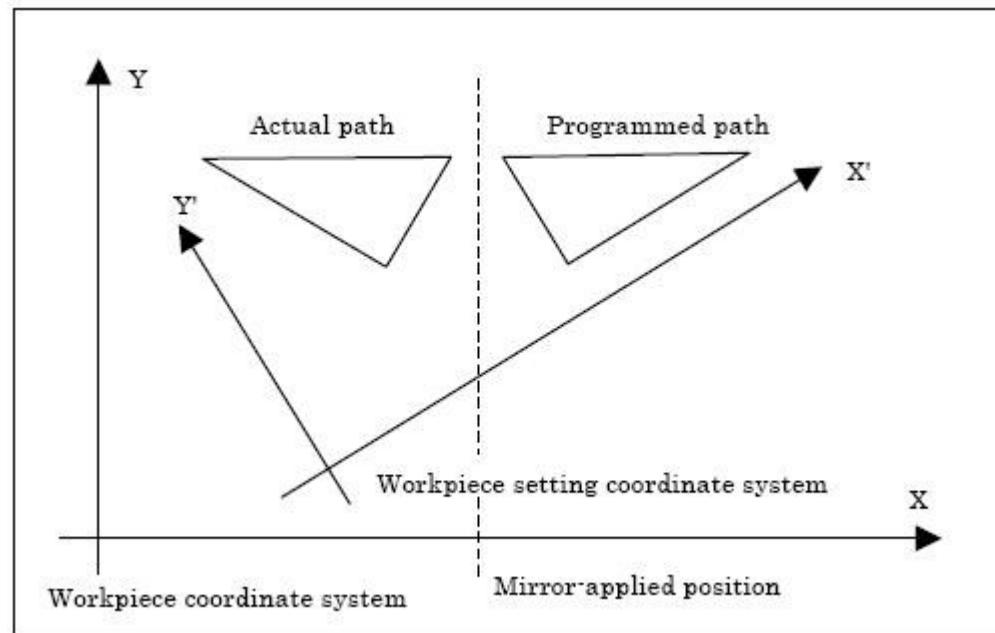
When Tool length offset(including Tool center point control) is commanded during Workpiece setting error compensation, Parameter TOS(No.5006#6) must be 1 (Tool length offset is done by shift of coordinates system). If TOS is 0, the alarm P/S0438 occurs.

Mirror image

If workpiece setting error compensation and programmable mirror image are used at the same time, programmable mirror image is applied to the coordinates in the workpiece setting coordinate system, then workpiece setting error compensation is applied.



If workpiece setting error compensation and external mirror image (mirror image based on a mirror image signal or setting) are used at the same time, workpiece setting error compensation is first applied, then external mirror image is applied.



Stroke limit check before movement

In the mode of Workpiece setting error compensation, Stroke limit check before movement is not available.

Feedrate override

When tool direction is compensated (Parameter RCM (No.11200#0)=1), the tool direction is compensated to the movement which is modified by the override signals. If a large movement of rotary axis occurs for the compensation, the federate of the rotary axis is clamped by the speed which is generated by calculation that the parameters No.11202, No.11203 override the maximum cutting federate/rapid traverse.

While rotary axes move with this override, even when the override signals change, the actual federate may not

change, and, even when feedhold is done, the actual rotary axes movements may not stop.

In the case that Tool posture control is used with Type-2 of Tool center point control for 5-axis machining

In the case that Tool posture control is used with Type-2 of Tool center point control for 5-axis machining, when the angle between a tool posture and the singular posture is less than the parameter No. **19738**, the tool posture at the end of the block is changed so that the tool posture passes the singular posture. But, while Workpiece setting error compensation is active, this change is not done.

2.5.3 Restrictions on 5-axis machines

Particularly when tool direction compensation is performed with a 5-axis machine of rotation table type or composite type (when bit 0 (RCM) of parameter No. 11200 is set to 1), the restrictions described below are imposed in addition to "Restrictions (general)" above.

Manual intervention

Do not perform manual intervention in the workpiece setting error compensation mode. Otherwise, an alarm is issued.

Hypothetical axis used for a table rotation axis

If a hypothetical axis is used as a table rotation axis, compensation is performed with the angle of the table rotation axis set to 0 degree.

Acceleration/deceleration at a corner

When a command for linear interpolation is specified, linear interpolation is performed as viewed from the workpiece on the table. So, even when the command specifies a linear interpolation, the control point may make a curved motion. This means that a corner operation may be performed with some commands.

So, if a small value is set as an allowable feedrate difference (parameter No. 1783) or as an allowable acceleration/deceleration rate (parameter No. **1660** and No. 1737) for a corner, the tool may be decelerated.

G codes that must not be specified

Among the "Specifiable G codes" in "Restrictions (general)" above, the G codes listed below must not be specified when tool direction compensation is performed with a 5-axis machine of table rotation type or composite type. If any of these G codes is specified, alarm PS0439 is issued.

G06.2 NURBS interpolation
G28 Return to reference position
G29 Return from reference position
G30 Return to 2nd, 3rd, or 4th reference position
G43.1 Tool length compensation in tool axis direction
G53 Machine coordinate system selection

Milling

G95 Feed per revolution

Turning

G95 Feed per revolution (for G code system B and C)

Modal G codes not usable when workpiece setting error compensation is specified

Among the "Modal G codes usable when workpiece setting error compensation is specified" in "Restrictions (general)" above, the G codes listed below must not be specified with a 5-axis machine of table rotation type or composite type in the modal G code state indicated below. If workpiece setting error compensation is specified in the modal G code state below, alarm PS0439 is issued.

Milling

G95 Feed per revolution

Turning

G95 Feed per revolution (for G code system B and C)

Command for an axis not related to 5-axis machining

Those axes that are not related to 5-axis machining must not be specified. Otherwise, alarm PS0439 is issued.

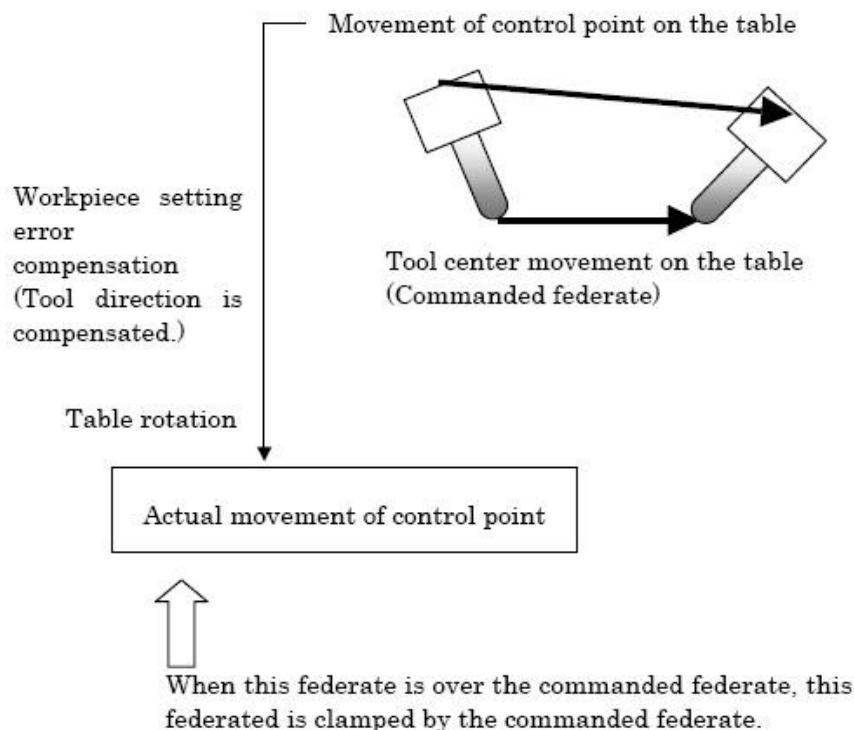
Tool center point control / Tilted working plane command

Before specifying tool center point control or tilted working plane command in the workpiece setting error compensation mode, be sure to set the table rotation axis position (absolute position in the workpiece coordinate system) to 0. Otherwise, alarm PS0439 is issued.

Feedrate during Workpiece setting error compensation

In the case that Workpiece setting error compensation is done on a table rotation type machine or combination type machine, a process similar to Tool center point control is treated so that movements on the table is controlled.

Then, movements of the control point is calculated so that the commanded federate is applied to the movement of Tool center. As the result, the federate of the control point may exceed the commanded federate. In such a case, the federate of the control point is clamped by the commanded federate.



In reverse, when the actual federate of control point is less than the commanded federate, the control point may move with the small federate.

IV MANUAL OPERATIONS

1 Manual Intervention during Tool Center Point

This function enables the use of manual intervention during tool center point control is available. Then 3-dimensional manual feed is available.

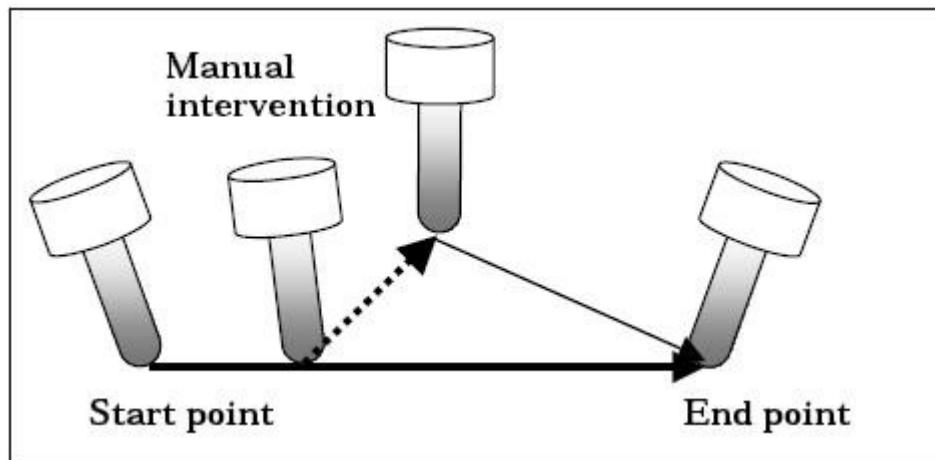
As for tool center point control and 3-dimensional manual feed, refer to USER'S MANUAL (B-63944EN).

1.1 Explanation

1.1.1 Tool rotation type machine

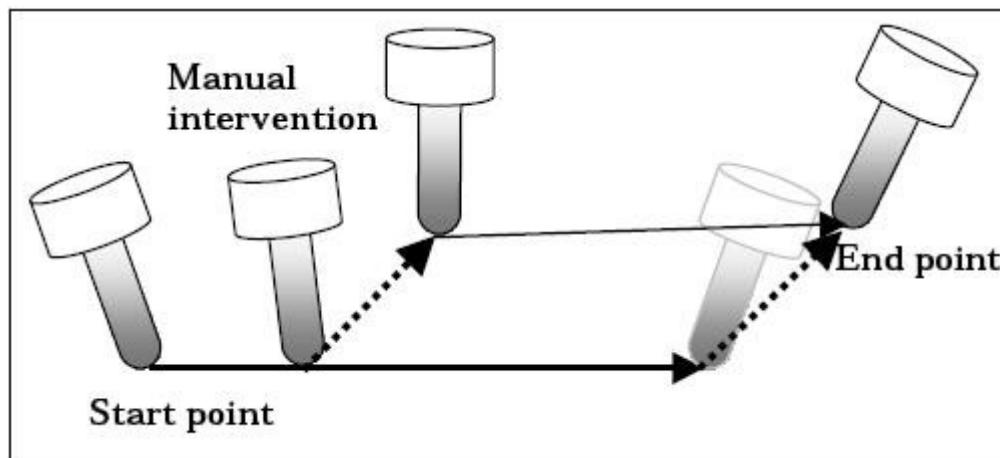
When manual absolute turns on (*ABSM=0)

Restarting after manual intervention in manual absolute on, the tool moves to end point of the program command. (In the case of absolute command (G90))



When manual absolute turns off (*ABSM=1)

Restarting after manual intervention in manual absolute off, the tool moves to the end position that is added manual intervention to, and manual intervention is added to later end positions also.



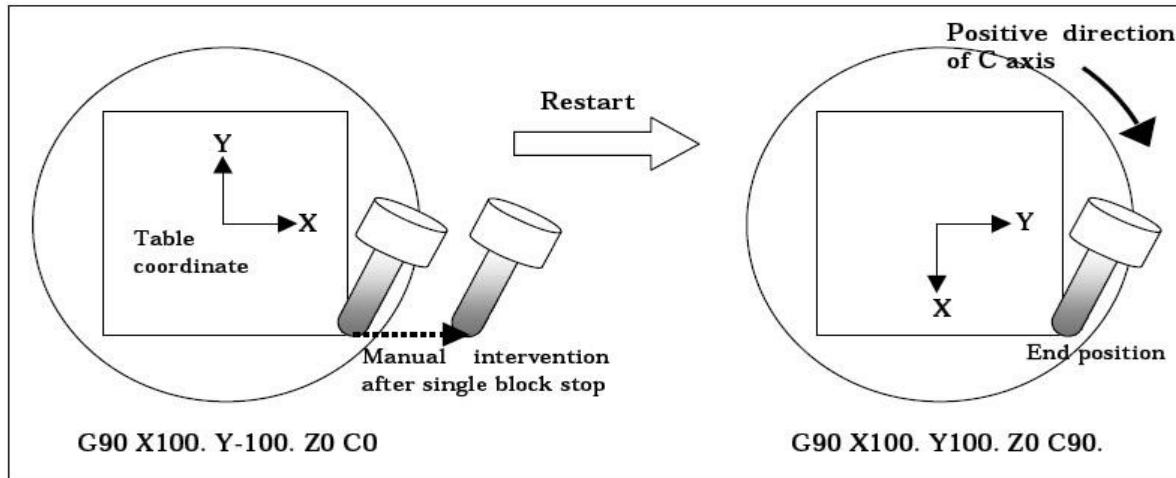
NOTE

When manual intervention to rotary axis is executed, tool center point is shifted by the amount of manual intervention.

1.2 Table rotation type and Mixed type machine

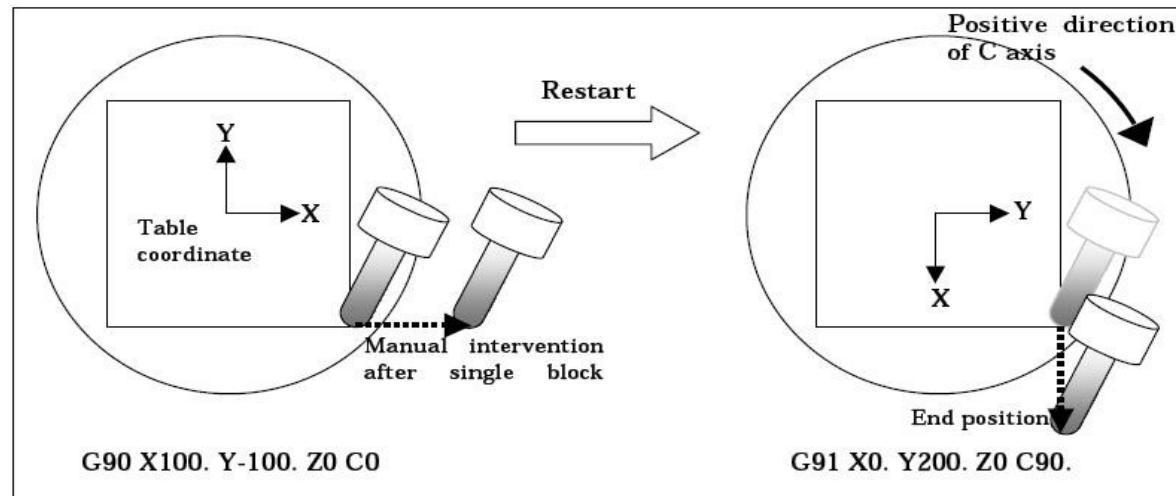
When manual absolute turns on (*ABSM=0)

Restarting after manual intervention in manual absolute on, the tool moves to end point of the program command.
(In the case of absolute command (G90))



When manual absolute turns off (*ABSM=1)

Restarting after manual intervention in manual absolute off, the tool moves on the condition that the amount of manual intervention is kept on "the table coordinate system".



2 Manual Feed for 5-Axis Machining

This function enables the use of the following functions.

- Manual feed for 5-axis machining
 - Tool axis direction handle feed/tool axis direction JOG feed/tool axis direction incremental feed
 - Tool axis right-angle direction handle feed/tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed
 - Tool tip center rotation handle feed/tool tip center rotation JOG feed/tool tip center rotation incremental feed
 - Table vertical direction handle feed/table vertical direction JOG feed/table vertical direction incremental feed
 - Table horizontal direction handle feed/table horizontal direction JOG feed/table horizontal direction incremental feed

A handle interrupt can be generated for each handle feed.

Handle interrupts work according to the corresponding handle feed specifications described hereinafter unless otherwise noted.

- Screen display functions
 - Display of the coordinate of the tool tip
 - Display of pulse values
 - Display of the amount of machine axes movement

NOTE

- 1 To execute 5-axis machining handle feed requires the manual handle feed option. Also, to generate 5-axis machining handle interrupts requires the manual handle interrupt option.
- 2 A 5-axis machining handle interrupt must not be generated when a rotation axis command is being executed during automatic operation.
- 3 Manual feed for 5-axis machining is disabled when the manual reference position return mode is selected.

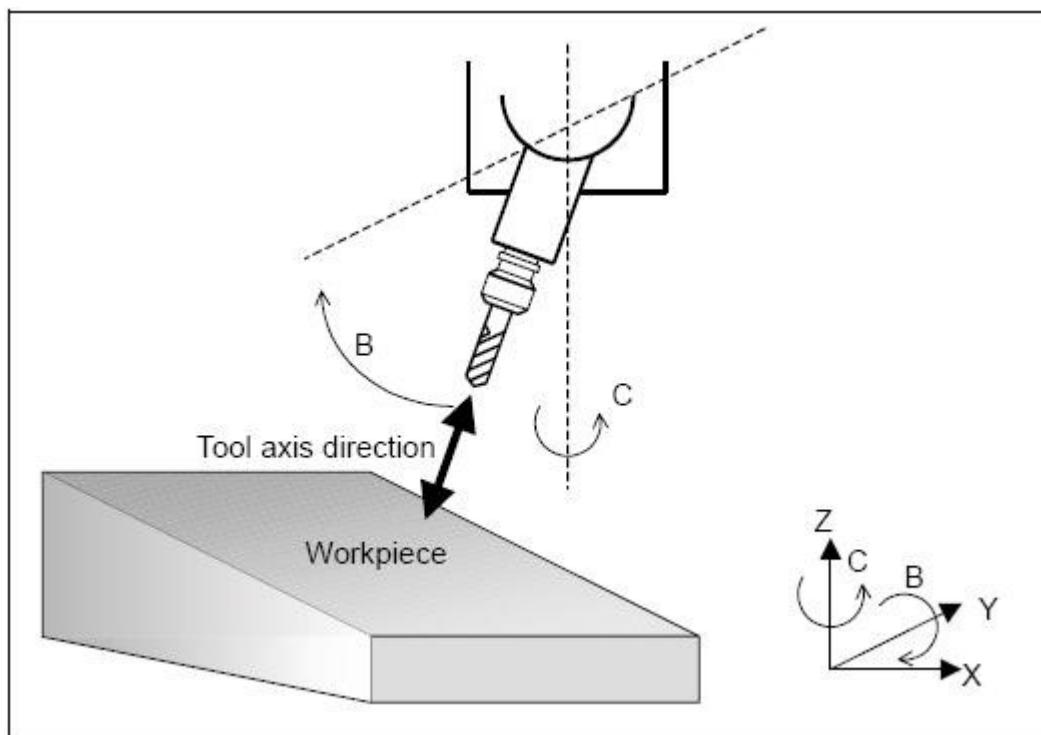
2.1 Tool Axis Direction Handle Feed/Tool Axis Direction JOG Feed/Tool Axis Direction Incremental Feed

In the tool axis direction handle feed, tool axis direction JOG feed, and tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

2.1.1 Explanation

Tool axis direction

The tool axis direction that is taken when all the rotation axes for controlling the tool are at an angle of 0 degree is specified in parameters No.**19697**, No.**19698**, and No.**19699**. As the rotation axes for controlling the tool rotate, the tool axis direction changes according to the rotation axis angle.



Tool axis direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. **12320** is set to 1, the feed direction of the tool axis direction feed in the tilted working plane command mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane command.

Tool axis direction handle feed

The tool axis direction handle feed is enabled when the following four conditions are satisfied:

- (1) Handle mode is selected.
- (2) The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".
- (3) The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis direction handle feed mode effective is set in parameter No.12310.
- (4) The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424).

Handle pulses generated while the clamp feedrate is exceeded are ignored.

Tool axis direction JOG feed/tool axis direction incremental feed

The tool axis direction JOG feed or tool axis direction incremental feed is enabled when the following three conditions are satisfied:

- (1) JOG mode or incremental feed mode is selected.
- (2) The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".

- (3) The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction specified by parameter No. **19697**. (Even when the tool axis direction is slant because of the settings of parameters No. **19698** and No. **19699**, the signal that activates the tool axis direction JOG feed or tool axis direction incremental feed is determined by parameter No. **19697** only.)

Ex.) No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.

- +J3: Tool axis direction +
- -J3: Tool axis direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. **12320** is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

2.2 Tool Axis Right-Angle Direction Handle Feed/Tool Axis Right-Angle Direction JOG Feed/Tool Axis Right-Angle Direction Incremental Feed

In the tool axis right-angle direction handle feed, tool axis direction JOG feed, or tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

If bit 1 (FLL) of parameter No. **12320** is set to 1, the tool or table is moved in the latitude or longitude direction determined by the tool axis direction vector.

2.2.1 Explanation

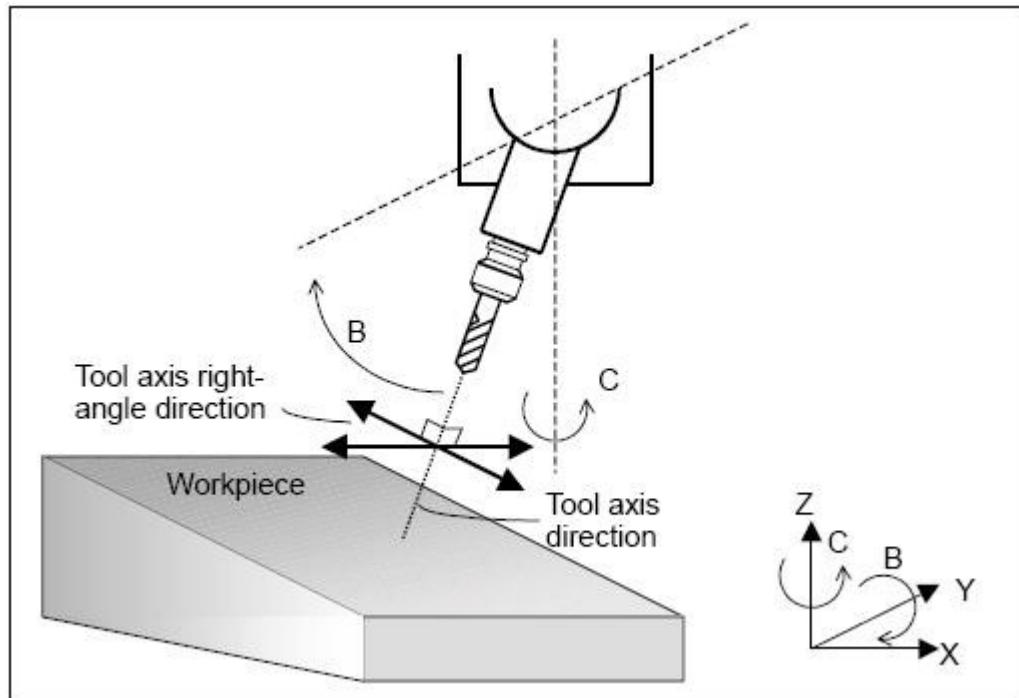
There are two tool axis right-angle directions, which are perpendicular to the tool axis direction (see the previous section).

Parameter No.19697		Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	(The reference tool direction is +X.)	+Y direction	+Z direction
2	(The reference tool direction is +Y.)	+Z direction	+X direction
3	(The reference tool direction is +Z.)	+X direction	+Y direction

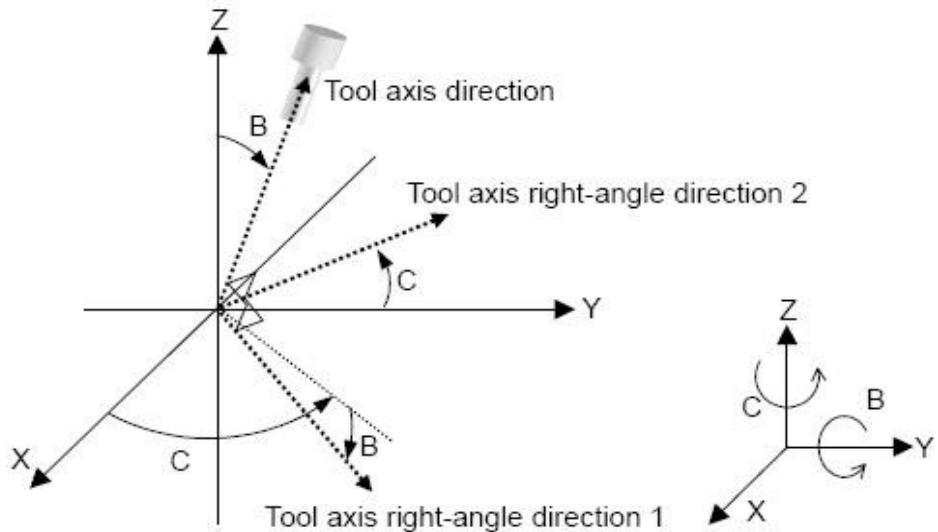
This table shows the tool axis right-angle directions that may be taken when the angles of all the rotation axes for controlling the tool are 0 degree and when parameters No. **19698** and No. **19699** are both set to 0.

When the reference tool axis direction is inclined based on the settings of parameters No. **19698** and No. **19699**, the tool axis right-angle direction is also inclined as much.

As the rotation axes for controlling the tool rotate, the tool axis right-angle direction changes according to the rotation axis angle.



(Example) When the tool rotation axes are B-axis and C-axis and the tool axis direction is the Z-axis direction



Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector (\vec{T}) and normal axis direction vector (\vec{P}) (parameter No.12321) be the tool axis right-angle direction 1 (longitude direction) vector (\vec{R}_1). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

$$\text{Equation: } \vec{R}_1 = \vec{P} \times \vec{T}$$

Let a vector perpendicular to the tool axis direction vector (\vec{T}) and tool axis right-angle direction 1 (longitude direction) vector (\vec{R}_1) be the tool axis right-angle direction 2 (latitude direction) vector (\vec{R}_2).

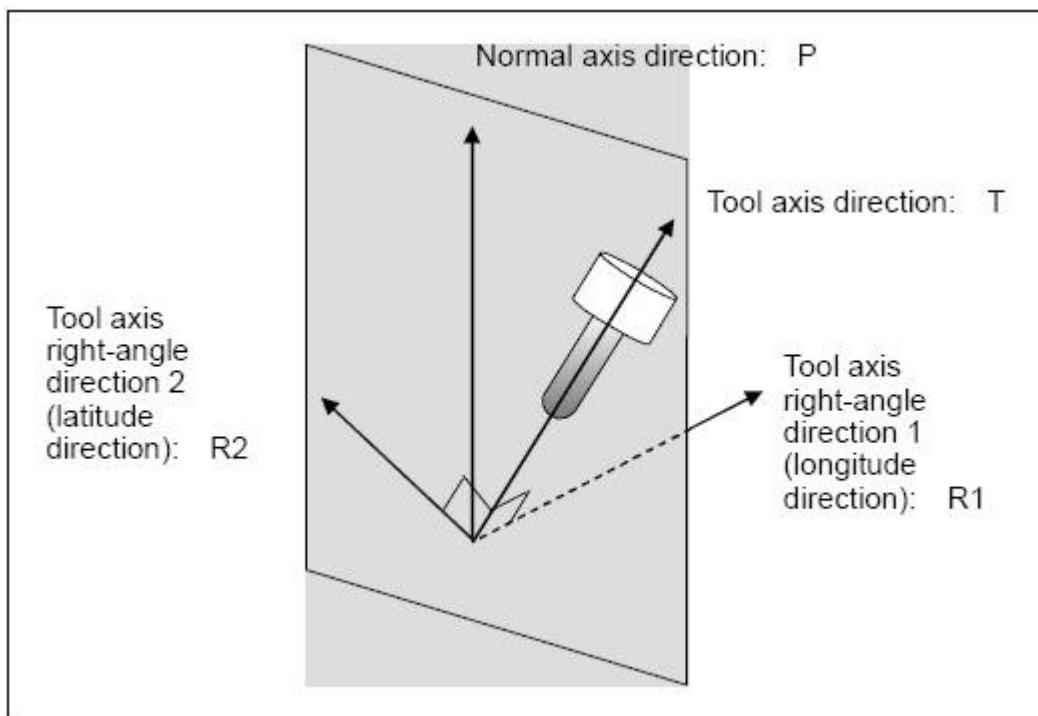
When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

$$\text{Equation: } \vec{R}_2 = \vec{T} \times \vec{R}_1$$

When the tool axis direction vector (\vec{T}) is parallel to the normal axis direction vector (\vec{P}) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No.12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the reference tool axis direction (parameter No. **19697**). If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459 is issued.



Tool axis right-angle direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. **12320** is set to 1, the feed direction of the tool axis right-angle direction feed in the tilted working plane command mode is defined as follows:

Tool axis right-angle direction 1: X direction in the feature coordinate system of the tilted working plane command
 Tool axis right-angle direction 2: Y direction in the feature coordinate system of the tilted working plane command

Tool axis right-angle direction handle feed

The tool axis right-angle direction handle feed is enabled when the following four conditions are satisfied:

- (1) Handle mode is selected.
- (2) The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".

- (3) The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis right-angle direction handle feed mode effective is set in parameter No.12311 or No.12312.
- (4) The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis right-angle direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

Tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed

The tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is enabled when the following three conditions are satisfied:

- (1) JOG mode or incremental feed mode is selected.
- (2) The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".
- (3) The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.**19697**. (Even when the tool axis direction is slant because of the settings of parameters No.**19698** and No.**19699**, the signal that activates the tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is determined by parameter No.**19697** only.)

Ex.) No.19697=3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1: Tool axis right-angle direction 1 +
- -J1: Tool axis right-angle direction 1 -
- +J2: Tool axis right-angle direction 2 +
- -J2: Tool axis right-angle direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. **12320** is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

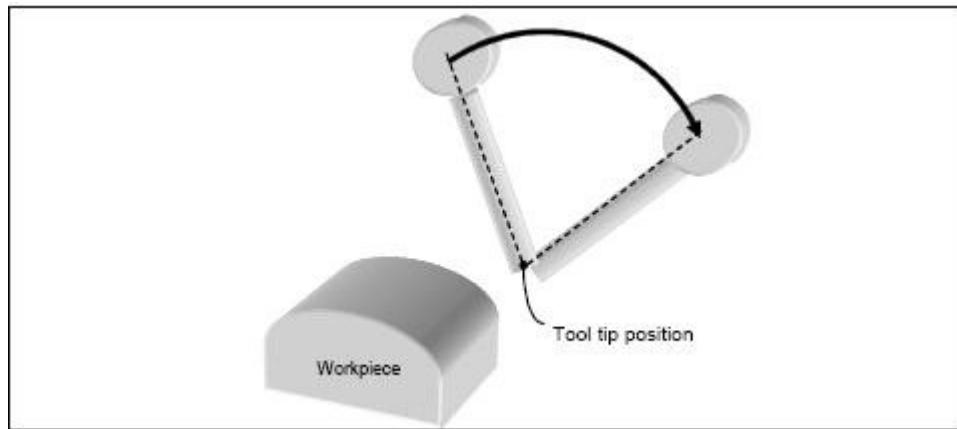
Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424).

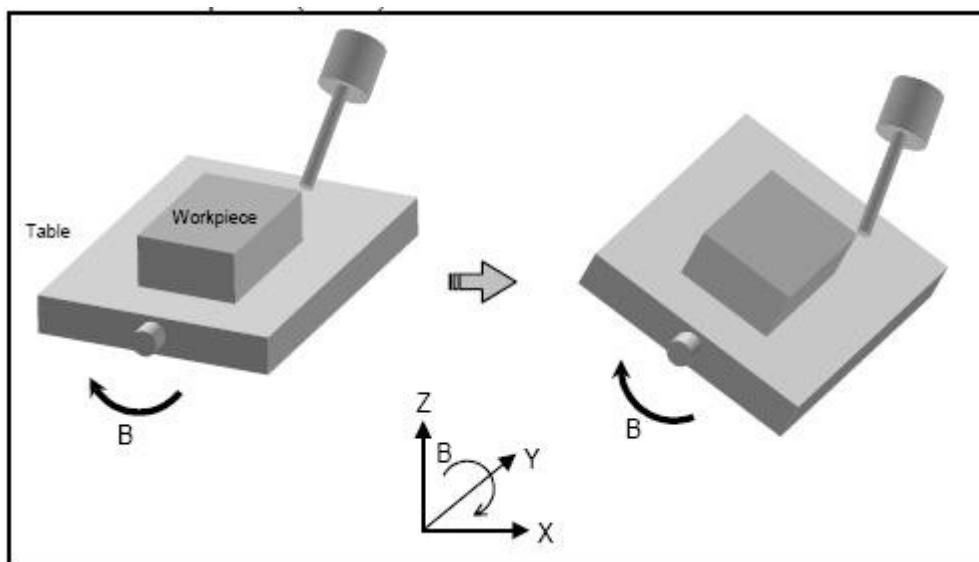
2.3 Tool Tip Center Rotation Handle Feed/Tool Tip Center Rotation JOG Feed/Tool Tip Center Rotation Incremental Feed

In the tool tip center rotation handle feed, tool tip center rotation JOG feed, and tool tip center rotation incremental feed, when a rotary axis is rotated by manual feed, the linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece (table).

- The following figure shows an example where the tool is rotated on the rotation axis. In this case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece.



- The following figure shows an example where the table is rotated on the rotation axis. As in the previous case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece (table).



Tool tip center rotation handle feed

The tool tip center rotation handle feed is enabled when the following four conditions are satisfied:

- (1) Handle mode is selected.
- (2) The tool tip center rotation feed mode signal (RNDH) is set to "1".
- (3) The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool tip center rotation handle feed mode effective is set in parameter No.12313 or No.12314.
- (4) The value of parameter No.12313 or No.12314 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the rotation axis is moved by the amount of rotation. The linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter

2.3 Tool Tip Center Rotation Handle Feed/Tool Tip Center Rotation JOG Feed/Tool Tip Center Rotation
Incremental Feed

No.1424) (of that particular axis). Handle pulses generated while the clamp feedrate is exceeded are ignored.

Tool tip center rotation JOG feed/tool tip center rotation incremental feed

The tool tip center rotation JOG feed or tool tip center rotation incremental feed is enabled when the following three conditions are satisfied:

- (1) JOG mode or incremental feed mode is selected.
- (2) The tool tip center rotation feed mode signal (RNDH) is set to "1".
- (3) The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the rotation axis to be rotated.

Ex.) When the B-axis (4th axis) is rotated

- +J4: Tool tip center rotation feed +
- -J4: Tool tip center rotation feed -

Feedrate

Control is exerted so that the synthetic speed of the linear axes (in the tangential direction) is the dry run rate (parameter No.1410). The manual feedrate override feature is available. If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis).

Selection of the tool length offset value

The tool length in manual feed for 5-axis machining is determined as explained below. Table 3.8.3 (a))

If bit 2 (LOD) of parameter No. **19746** is set to 0, the value set in parameter No. 12318 is assumed to be the tool length. If the LOD parameter is set to 1, and the tool length offset function is performed, the offset data specified for the tool length offset is assumed to be the tool length.

If the LOD parameter is set to 1, and the tool length offset function is not performed, the tool length is determined as follows. If bit 3 (LOZ) of parameter No. **19746** is set to 0, the value set in parameter No. 12318 is assumed to be the tool length in manual feed for 5-axis machining; if LOZ is set to 1, the tool length is assumed to be 0.

		19746#2(LOD)		
		= 0	= 1	
			Tool length offset enabled	Tool length offset canceled
19746#3 (LOZ)	= 0	Parameter No. 12318	Offset data	Parameter No. 12318
	= 1			0

The tool length offset function is enabled when the following two conditions are both satisfied:

- The tool length offset function listed below is enabled (modal code of group 8 except G49)
 - G43 / G44 : Tool length compensation
 - G43.4 / G43.5 : Tool center point control
- The H/D code is other than 0.

If bit 6 (CLR) of parameter No. 3402 is set to 0 not to clear the tool length offset vector, G codes of group 8, and H

codes at the time of a reset, the tool length offset status is maintained when a reset is made in the tool length offset mode.

2.4 Table Vertical Direction Handle Feed/Table Vertical Direction JOG Feed/Table Vertical Direction Incremental Feed

In the table vertical direction handle feed, table vertical direction JOG feed, and table vertical direction incremental feed, the tool is moved in the table vertical direction.

2.4.1 Explanation

Table vertical direction

The table vertical direction is a direction vertical to the table. It is equal to the tool axis direction specified in parameter No. **19697** when all of the rotation axes for controlling the table are at an angle of 0 degree. When the rotation axes for controlling the table rotate, the table vertical direction changes according to the rotation axis angle.

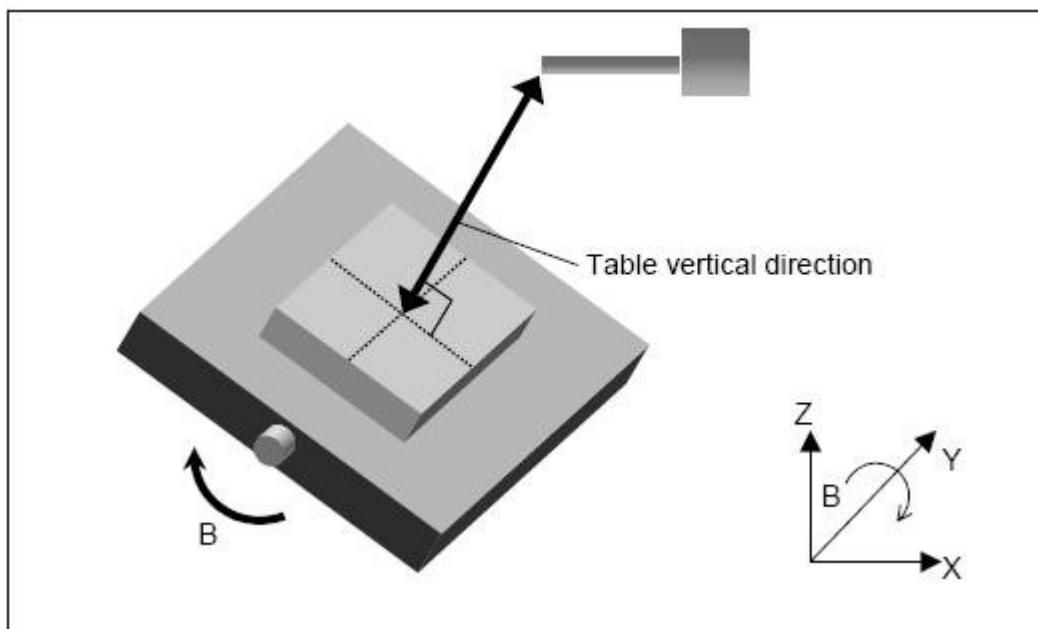


Table-based vertical direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. **12320** is set to 1, the feed direction of the table-based vertical direction feed in the tilted working plane command mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane command.

Table vertical direction handle feed

The table vertical direction handle feed is enabled when the following four conditions are satisfied:

- (1) Handle mode is selected.
- (2) Both the tool axis direction feed mode signal (ALNGH) and the table base signal (TB_BASE) are set to "1".
- (3) The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the table vertical handle feed mode effective is set in parameter No.12310.
- (4) The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table vertical direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

Table vertical direction JOG feed/table vertical direction incremental feed

The table vertical direction JOG feed or table vertical direction incremental feed is enabled when the following three conditions are satisfied:

- (1) JOG mode or incremental feed mode is selected.
- (2) Both the tool axis direction feed mode signal (ALNGH) and the table base signal (TB_BASE) are set to "1".
- (3) The feed axis direction selection signal [+Jn,-Jn (where n = 1 to the number of controlled axes)] is set to "1" for the axis corresponding to the direction specified by parameter No.**19697**.
Ex.) No.**19697** = 3 (+Z-axis direction); Z-axis is the 3rd axis.
 - +J3: Table vertical direction +
 - -J3: Table vertical direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424).

2.5 Table Horizontal Direction Handle Feed/Table Horizontal Direction JOG Feed/Table Horizontal Direction Incremental Feed

In the table horizontal direction handle feed, table horizontal direction JOG feed, and table horizontal direction incremental feed, the tool is moved in the table horizontal direction.

If bit 1 (FLL) of parameter No. **12320** is set to 1, the tool or table is moved in the latitude or longitude direction determined by the table-based vertical direction vector.

2.5.1 Explanation

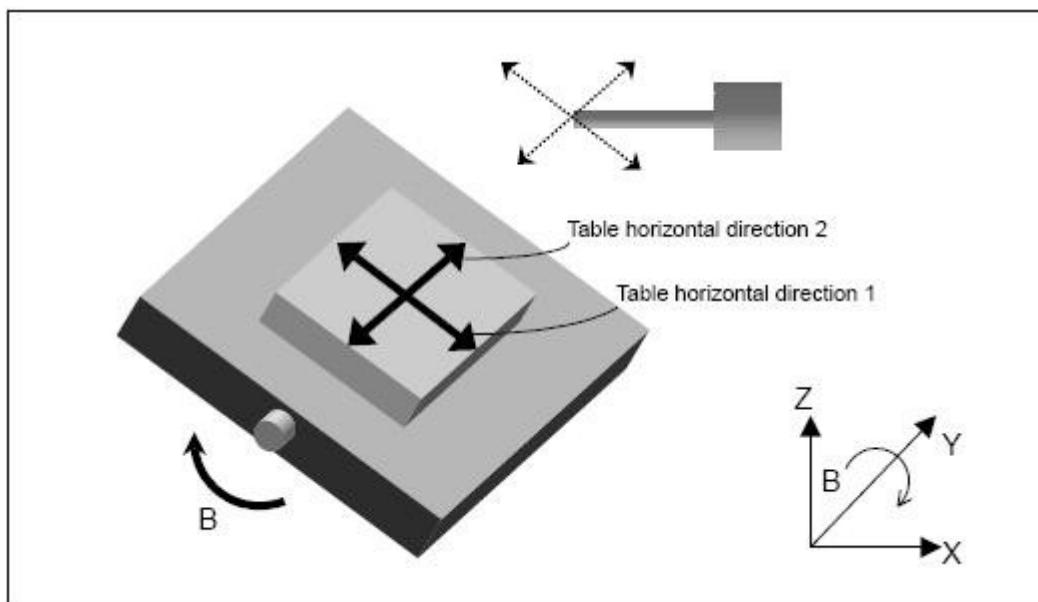
Table horizontal direction

There are two table horizontal directions, which are perpendicular to the table vertical direction (see the previous section).

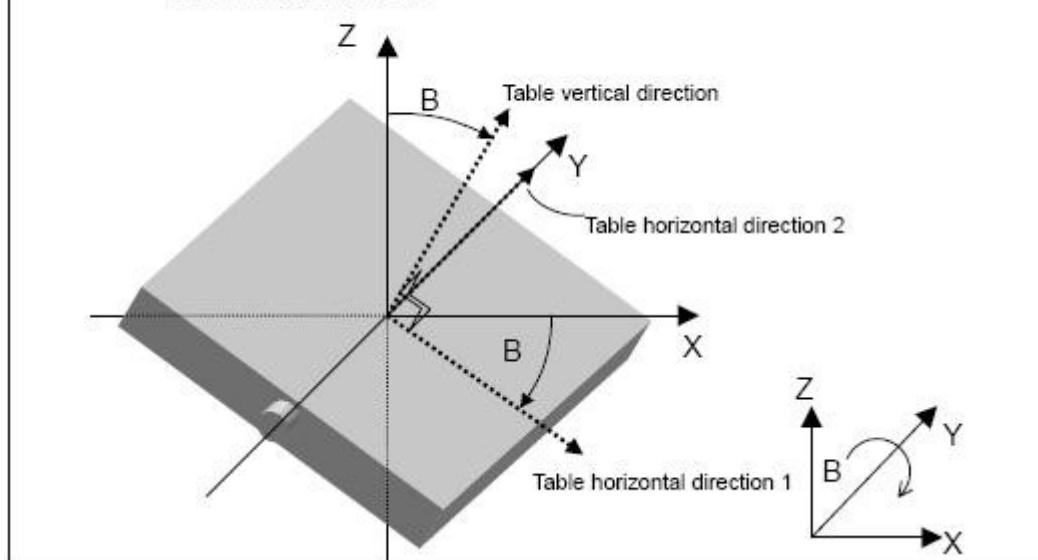
Parameter No.19697		Table horizontal direction 1	Table horizontal direction 2
1	(The reference tool direction is +X.)	+Y direction	+Z direction
2	(The reference tool direction is +Y.)	+Z direction	+X direction
3	(The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the table horizontal directions that may be taken when the angles of all the rotation axes for controlling the table are 0 degree.

As the rotation axes for controlling the table rotate, the table horizontal direction changes according to the rotation axis angle.



(Example) When the table rotation axis is the B-axis, and the table vertical direction is the Z-axis direction



Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector (\vec{T}) and normal axis direction vector (\vec{P}) (parameter No. 12321) be the tool axis right-angle direction 1 (longitude direction) vector (\vec{R}_1). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

$$\text{Equation: } \vec{R}_1 = \vec{P} \times \vec{T}$$

Let a vector perpendicular to the tool axis direction vector (\vec{T}) and tool axis right-angle direction 1 (longitude direction) vector (\vec{R}_1) be the tool axis right-angle direction 2 (latitude direction) vector (\vec{R}_2).

When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

$$\text{Equation: } \vec{R}_2 = \vec{T} \times \vec{R}_1$$

When the tool axis direction vector (\vec{T}) is parallel to the normal axis direction vector (\vec{P}) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No. 12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the tool axis direction.

If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459 is issued.

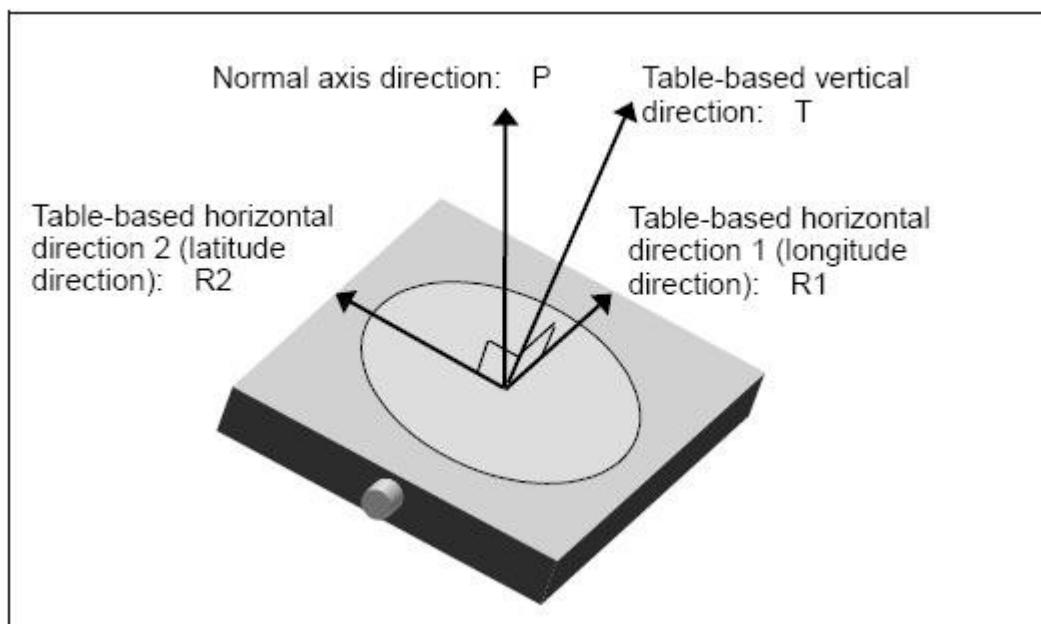


Table-based horizontal direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based horizontal direction feed in

the tilted working plane command mode is defined as follows:

Table-based horizontal direction 1: X direction in the feature coordinate system of the tilted working plane command

Table-based horizontal direction 2: Y direction in the feature coordinate system of the tilted working plane command

Table horizontal direction handle feed

The table horizontal direction handle feed is enabled when the following four conditions are satisfied:

- (1) Handle mode is selected.
- (2) Both the tool axis right-angle direction feed mode signal (RGHTH) and the table base signal (TB_BASE) are set to 1.
- (3) The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the table horizontal direction handle feed mode effective is set in parameter No.12311 or No.12312.
- (4) The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table horizontal direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424).

Handle pulses generated while the clamp feedrate is exceeded are ignored.

Table horizontal direction JOG feed/table horizontal direction incremental feed

The table horizontal direction JOG feed or table horizontal direction incremental feed is enabled when the following three conditions are satisfied:

- (1) JOG mode or incremental feed mode is selected.
- (2) Both the tool axis right-angle direction feed mode signal (RGHTH) and the table base signal (TB_BASE) are set to "1".
- (3) The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697.

Ex.) No.19697 = 3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1: Table horizontal direction 1 +
- -J1: Table horizontal direction 1 -
- +J2: Table horizontal direction 2 +
- -J2: Table horizontal direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) of a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis does not exceed the manual rapid traverse rate (parameter No.1424).

NOTE

1. To perform a handle feed for 5-axis machining, the manual handle feed option is required. To perform a handle interrupt for 5-axis machining, the manual handle interrupt option is required.
2. When a handle interrupt for 5-axis machining is performed, rotation axis command execution must not be in progress in automatic operation.
3. When the manual reference position return mode is selected, manual feed for 5-axis machining is not enabled.
4. When the offset value specified for the tool length offset function is used for tool center point rotation feed (when bit 2 (LOD) of parameter No. **19746** is set to 1), the controlled point should generally be shifted. (Set bit 5 (SVC) of parameter No. 19665 to 1.) In this case, specify the tool length with a radius value.

V APPENDIX

1 Parameters

2 Parameters

NOTE: The power must be turned off before operation is continued.

1260 Amount of a shift per one rotation of a rotation axis

Input type	Parameter input
Data type	Real axis
Unit of data	Degree
Minimum unit of data	Depends on the increment system of the applied axis
Valid data range	0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B)) (When the increment system is IS-B, 0.0 to +999999.999)
	Set the amount of a shift per one rotation of a rotation axis. For the rotation axis used for cylindrical interpolation, set the standard value.

#7	#6	#5	#4	#3	#2	#1	#0
1401						LRP	

1 LRP Positioning (G00)
 0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
 1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
 Set to 1 for:
 Three dimensional coordinate conversion
 Tool center point control command

#7	#6	#5	#4	#3	#2	#1	#0
1403			HTG				

5 HTG The feedrate for helical interpolation/helical involute interpolation/three-dimensional circular interpolation is:
 0: Tangential speed of arc
 Specified using the feedrate along the tangent to an arc/involute curve/three-dimensional arc
 1: Synthetic speed of the linear axis speed and tangential speed
 Specified using the feedrate along axes including a linear axis (specified axes other than the circular interpolation axis in the case of three-dimensional circular interpolation)

1660	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis
	<p>Input type Parameter input</p> <p>Data type Real axis</p> <p>Unit of data mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)</p> <p>Minimum unit of data Depends on the increment system of the applied axis</p> <p>Valid data range Refer to the standard parameter setting table (D) (When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.) Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for each axis. If a value greater than 100000.0 is set, the value is clamped to 100000.0. If 0 is set, the specification of 100000.0 is assumed. If 0 is set for all axes, however, acceleration/deceleration before interpolation is not performed than a maximum allowable acceleration rate set for another axis by a factor or 2 or more, the feedrate at a corner where the direction of travel abruptly changes can decrease temporarily.</p>
1671	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis, or maximum allowable reference acceleration rate in optimum torque acceleration/deceleration
	<p>Input type Parameter input</p> <p>Data type Real axis</p> <p>Unit of data mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)</p> <p>Minimum unit of data Depends on the increment system of the applied axis</p> <p>Valid data range Refer to the standard parameter setting table (D) (When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.) (1) Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for linear rapid traverse. If a value greater than 100000.0, the value is clamped to 100000.0. If 0 is set, the specification of the following is assumed: 1000.0 mm/sec/sec 100.0 inch/sec/sec 100.0 degrees/sec/sec If 0 is specified for all axes, however, acceleration/deceleration before interpolation is not performed. (2) Maximum allowable reference acceleration rate in optimum torque acceleration/deceleration</p>

1672

Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse, or acceleration change time of bell-shaped acceleration/deceleration in optimum torque acceleration/deceleration

Input type Parameter input

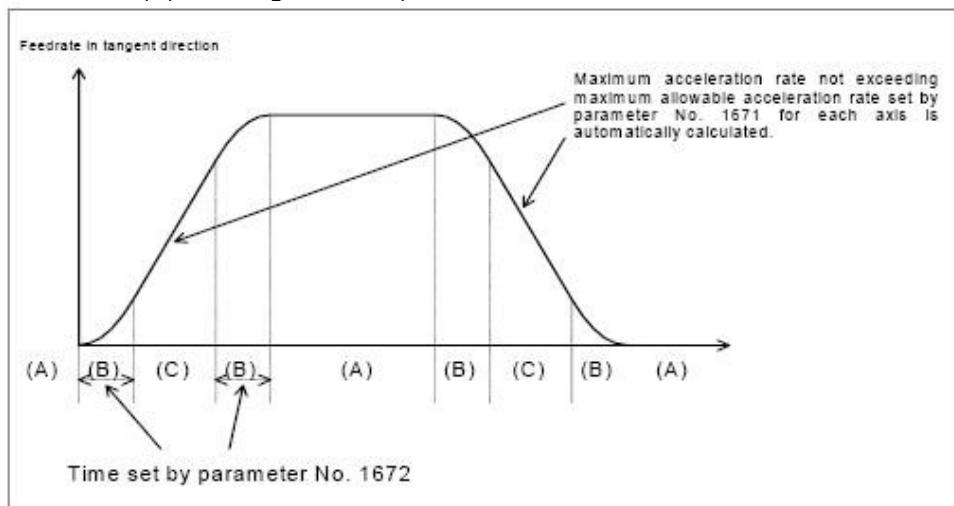
Data type Real axis

Unit of data mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)

Minimum unit of data Depends on the increment system of the applied axis

Valid data range 0 to 200

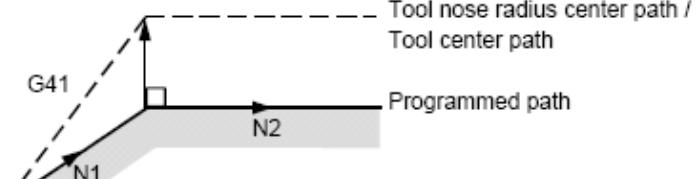
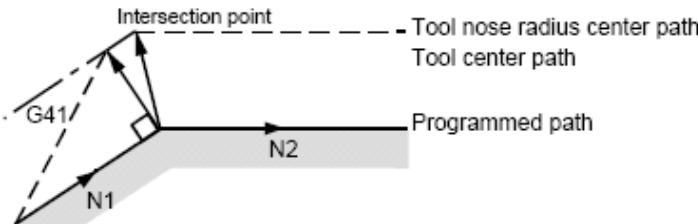
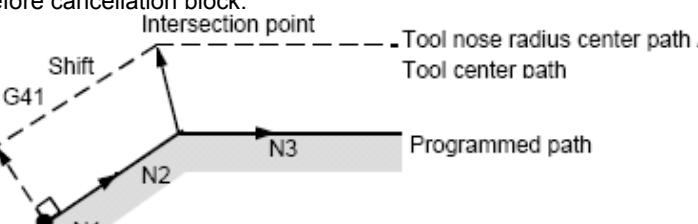
- Set an acceleration change time of bell-shaped acceleration/ deceleration for linear rapid traverse (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. **1671**: time of (B) in the figure below).
- Set an acceleration change time of bell-shaped acceleration/ deceleration in optimum torque acceleration/deceleration (time for changing from the state of constant feedrate (A) to the state of acceleration/deceleration (C) at the acceleration rate calculated from optimum torque acceleration/deceleration: time of (B) in the figure below).



	#7	#6	#5	#4	#3	#2	#1	#0
5003							SUV	SUP
#0	SUP	These bits are used to specify the type of startup/cancellation of cutter compensation or tool nose radius compensation.						
#1	SUV							

NOTE

When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS16i-T is performed.

SUV	SUP	Type	Operation
0	0	A	<p>A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output.</p>  <p>The diagram shows a corner of a workpiece being machined. A dashed line labeled "Tool nose radius center path / Tool center path" extends from the corner. A solid line labeled "Programmed path" follows the corner. A right-angle bracket indicates the direction of compensation. Labels N1 and N2 mark the blocks before and after the corner respectively. A dashed line labeled G41 indicates the tool's path relative to the programmed path.</p>
0	1	B	<p>A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output.</p>  <p>The diagram shows a corner of a workpiece being machined. A dashed line labeled "Tool nose radius center path / Tool center path" extends from the corner. A solid line labeled "Programmed path" follows the corner. A right-angle bracket indicates the direction of compensation. Labels N1 and N2 mark the blocks before and after the corner respectively. A dashed line labeled G41 indicates the tool's path relative to the programmed path. An additional vector labeled "Intersection point" is shown originating from the corner, perpendicular to the compensation vector.</p>
1	0 1	C	<p>When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.</p>  <p>The diagram shows a corner of a workpiece being machined. A dashed line labeled "Tool nose radius center path / Tool center path" extends from the corner. A solid line labeled "Programmed path" follows the corner. A right-angle bracket indicates the direction of compensation. Labels N1, N2, and N3 mark the blocks before and after the corner respectively. A dashed line labeled G41 indicates the tool's path relative to the programmed path. A vector labeled "Shift" is shown originating from the corner, perpendicular to the compensation vector.</p> <p>When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.</p>

#7	#6	#5	#4	#3	#2	#1	#0
----	----	----	----	----	----	----	----

11221								MTW
--------------	--	--	--	--	--	--	--	------------

0 MTW Multiple tilted working plane command is:

- 0: Not used
- 1: used

11262	Angle to judge singular posture (Cutting Point Command)
--------------	---

Input type Parameter input

Data type Real path

Unit of data Degree

Minimum unit of data Depends on the increment system of the reference axis

Valid data range 9 digit of the minimum data unit of data (refer to standard parameter setting table A)

(When the increment system is IS-B, -999999.999 - +999999.999)

The tool posture is considered to be near singular posture if the angle from the direction of tool length offset to the perpendicular to cutting surface is smaller than this parameter.

In the case of near singular posture, "Vector from the program point to Tool center point " is replaced by "Vector from the cutting point to Tool center point " immediately before becoming near singular posture.

#7 #6 #5 #4 #3 #2 #1 #0

12320						JFR	FLL	TWD
--------------	--	--	--	--	--	------------	------------	------------

#0 TWD The directions of 5-axis machining manual feed (other than tool tip center rotation feed) when the tilted working plane command is issued are:

- 0: Same as those not in the tilted working plane command. That is, the directions are:
 - Tool axis normal direction 1 (table-based horizontal direction 1)
 - Tool axis normal direction 2 (table-based horizontal direction 2)
 - Tool axis direction (table-based vertical direction)
- 1: X, Y, and Z directions in the feature coordinate system.

#1 FLL The directions of tool axis normal direction feed or table-based horizontal direction feed in the 5-axis machining manual feed mode are:

- 0: Tool axis normal direction 1 (table-based horizontal direction 1) and tool axis normal direction 2 (table-based horizontal direction 2).
- 1: Longitude direction and latitude direction.

FLL (12320#1)	TWD (12320#0)	Directions of 5-axis machining manual feed
0	0	Conventional directions
0	1	When the tilted working plane command is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Conventional directions
1	0	Longitude direction and latitude direction
1	1	When the tilted working plane command is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Longitude direction and latitude direction

#2 JFR As the feedrate of 5-axis machining jog feed or incremental feed:

- 0: The dry run rate (parameter No. 1410) is used.
- 1: The jog feedrate (parameter No. 1423) is used.

#7 #6 #5 #4 #3 #2 #1 #0

19501						FRP	
--------------	--	--	--	--	--	------------	--

#5 FRP The feedrate for helical interpolation/helical involute interpolation/three-dimensional circular interpolation is:

- 0: Specified using the feedrate along the tangent to an arc/involute curve/three-dimensional arc
- 1: Specified using the feedrate along axes including a linear axis (specified axes other than the circular interpolation axis in the case of three-dimensional circular interpolation)

19587	Tolerance of rotary axes for nano smoothing for 5-axis machining							
--------------	--	--	--	--	--	--	--	--

Input type Setting input
 Data type Real axis
 Unit of data degree (input unit)
 Minimum unit of data Depend on the increment system of the applied axis
 Valid data range 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
 (When the increment system is IS-B, 0.0 - +999999.999)
 Tolerances of rotary axes for nano smoothing for 5-axis machining are specified.
 Only values for rotary axes that are commanded in nano smoothing for 5-axis machining are valid.
 The minimum increment is applied as tolerance if this parameter is 0.

	#7	#6	#5	#4	#3	#2	#1	#0
19604								TPC

#0 TPC In the case that there is no address P at the start of Tool center point control for 5-axis machining (G43.4/G43.5),
 0: Tool posture control does not work
 1: Tool posture control works

	#7	#6	#5	#4	#3	#2	#1	#0
19605	TIT							

#7 TIT In the case of a tool rotation type machine (parameter(No.**19680**)=2), when tool center point control and inverse time feed are simultaneously used,
 0: Inverse time feed adjusts under tool center point control
 1: Move as a tool axis direction tool length compensation

	#7	#6	#5	#4	#3	#2	#1	#0
19607						SPG		

#2 SPG When cutter compensation for 5-axis machining is performed with a machine containing a table rotary axis, the G codes to be used are:
 0: Made.
 1: Not made.

	#7	#6	#5	#4	#3	#2	#1	#0
19608			PRI			DET		
# 5	PRI	Among multiple end point candidates that exist when a movement is made on a rotation axis by a command such as I, J, and K when a slanted surface machining command is specified under tool tip point control for 5-axis machining (type 2) or cutter compensation for 5-axis (type 2): 0: A combination in which the master (first rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the table (second rotation axis) makes a smaller angular movement is selected for a machine of composite type. 1: A combination in which the slave (second rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the tool (first rotation axis) makes a smaller angular movement is selected for a machine of composite type.						
#2	DET	When the programming coordinate system is fastened to the table in tool tip point control for 5-axis machining or cutter compensation for 5-axis, the relative position and absolute position of a specified path are: 0: Displayed in the programming coordinate system (fastened to the table). 1: Displayed in the workpiece coordinate system (not fastened to the table).						

19632	Distance from a programmed point (pivot point) to tool tip position (cutting point)
Input type	Setting input
Data type	Real path
Unit of data	Mm, inch (input unit)
Minimum unit of data	Depend on the increment system of the reference axis
Valid data range	9 digit of minimum unit of data (refer to the standard parameter setting table (A)) (When the increment system is IS-B, -999999.999 to +999999.999) Set the distance from a programmed point to actual cutting point to allow vector calculation for three-dimensional cutter compensation at the tool tip. If this parameter is set to 0, the three-dimensional cutter compensation function cannot be performed at the tool tip.

NOTE

When changing the setting of this parameter, make the change before turning on the three-dimensional cutter compensation mode.

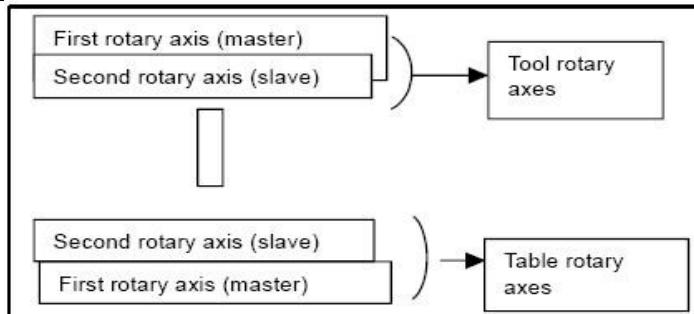
19635	Angle for determination in interference checks in three-dimensional cutter compensation
Input type	Setting input
Data type	Real path
Unit of data	degree (input unit)
Minimum unit of data	Depend on the increment system of the reference axis
Valid data range	9 digit of minimum unit of data (refer to the standard parameter setting table (A)) (When the increment system is IS-B, -999999.999 to +999999.999) In three-dimensional cutter compensation, if the difference in angle between two tool vectors is greater than or equal to the setting in this parameter, the tool direction is regarded as having changed. If 0 is set, 45 degrees is assumed. Let two tool vectors be Va and Vb. If the difference in angle is α degrees or greater as shown in the figure below, the tool vector is regarded as having changed.

19680 Mechanical unit type

Input type Setting input
 Data type Byte path
 Valid data range 0 to 21

Specify the type of the mechanical unit

19680	Mechanical unit type	Controlled rotation axis	Master and slave
0		Mechanism having no rotation axis	
2	Tool rotation type	Two rotation axis of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
12	Table rotation type	Two rotation axis of the table	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.



NOTE

A hypothetical axis is also counted as a controlled rotary axis.

<Hypothetical axis>

In some cases, it is convenient to use an imaginary rotary axis whose angle is fixed to a certain value. For example, suppose that a tool is mounted in a tilted manner through an attachment.

In such a case, the rotary axis considered hypothetically is a hypothetical axis. Bits 0 and 1 of parameter No. **19696** determine whether each rotary axis is an ordinary rotary axis or a hypothetical axis.

+

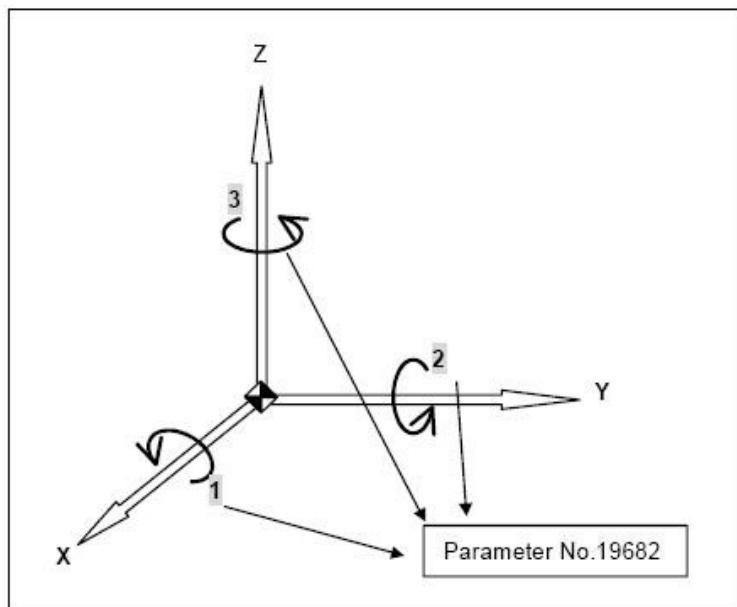
19681 Controlled-axis number for the first rotation axis

Input type Parameter input
 Data type Byte path
 Valid data range 0 to number of controlled axis

Set the controlled-axis number for the first rotation axis.
 For a hypothetical axis (when bit 0 (IA1) of parameter No. **19696** is 1), set 0.

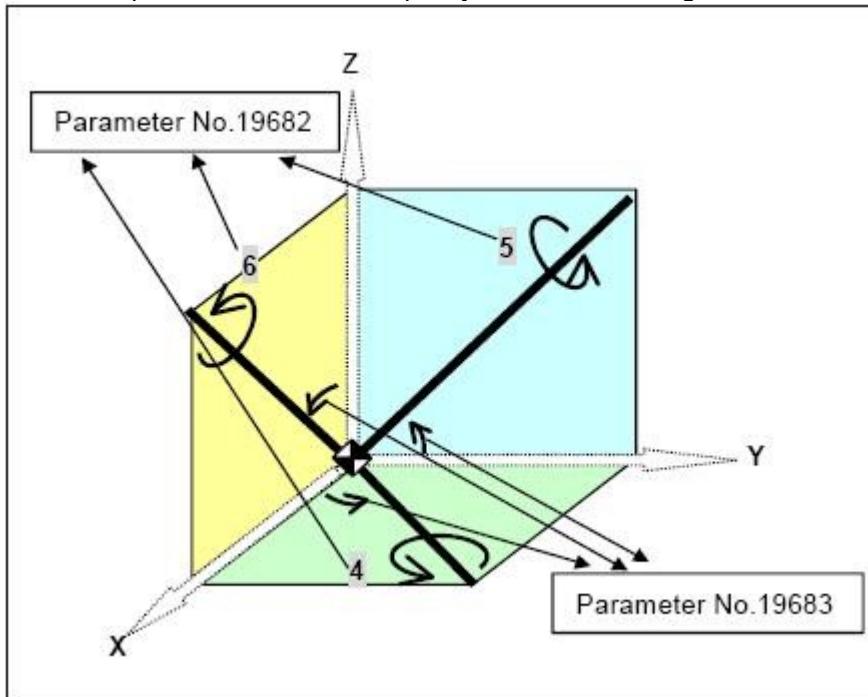
19682 Axis direction of the first rotation axis

Input type Parameter input
Data type Byte path
Valid data range 0 to 6
Specify the axis direction of the first rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis
4: On an axis tilted a certain angle from the X-axis to the positive Y-axis
5: On an axis tilted a certain angle from the Y-axis to the positive Z-axis
6: On an axis tilted a certain angle from the Z-axis to the positive X-axis
(A value 4 to 6 is to be set when the inclined rotation axis control function is used.)



19683 Inclination angle when the first rotation axis is an inclined axis

Input type	Parameter input
Data type	Real path
Unit of data	Degree
Minimum unit of data	The increment system of the reference axis is to be followed.
Valid data range	Nine digits of the least input increment (see standard parameter setting table (A).) (-999999.999 to +999999.999 for IS-B) When a value 1 to 3 is set in parameter No. 19682 , set 0 degrees. When a value 4 to 6 is set in parameter No. 19682 , specify the inclination angle.



19684 Rotation direction the first rotation axis

Input type	Parameter input
Data type	Byte path
Valid data range	0 to 1
	Set the direction in which the first rotation axis rotates as a mechanical motion when a positive move command is issued. 0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (right-hand thread rotation) 1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (left-hand thread rotation) Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19685	Rotation angle when the first rotation axis is a hypothetical axis
Input type	Parameter input
Data type	Real path
Unit of data	Degree
Minimum unit of data	Depend on the increment system of the reference axis
Valid data range	9 digit of minimum unit of data (refer to standard parameter setting table (A)) (When the increment system is IS-B, -999999.999 to +999999.999) When the first rotation axis is a hypothetical axis (bit 0 (IA1) of parameter No. 19696 is 1), set the rotation angle.
19686	Controlled-axis number for the second rotation axis
Input type	Parameter input
Data type	Byte path
Valid data range	0 to number of controlled axis Set the controlled-axis number for the first rotation axis. For a hypothetical axis (when bit 1 (IA2) of parameter No. 19696 is 1), set 0.
19687	Axis direction of the second rotation axis
Input type	Parameter input
Data type	Byte path
Valid data range	0 to 6 Specify the axis direction of the second rotation axis. 1: On X-axis 2: On Y-axis 3: On Z-axis 4: On an axis tilted a certain angle from the X-axis from the positive X-axis to positive Y-axis 5: On an axis tilted a certain angle from the Y-axis from the positive Y-axis to positive Z-axis 6: On an axis tilted a certain angle from the Z-axis from the positive Z-axis to positive X-axis (A value 4 to 6 is to be set when the inclined rotation axis control function is used.)
19688	Inclination angle when the second rotation axis is an inclined axis
Input type	Parameter input
Data type	Real path
Unit of data	Degree
Minimum unit of data	The increment system of the reference axis is to be followed.
Valid data range	Nine digits of the least input increment (see standard parameter setting table (A).) (-999999.999 to +999999.999 for IS-B) When a value 1 to 3 is set in parameter No. 19687 , set 0 degrees. When a value 4 to 6 is set in parameter No. 19687 , specify the inclination angle.

19689	Rotation direction the second rotation axis
--------------	---

Input type Parameter input
 Data type Byte path
 Valid data range 0 to 1
 Set the direction in which the second rotation axis rotates as a mechanical motion when a positive move command is issued.
 2: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. **19687** (right-hand thread rotation)
 3: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. **19687** (left-hand thread rotation)
 Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19690	Rotation angle when the second rotation axis is a hypothetical axis
--------------	---

Input type Parameter input
 Data type Real path
 Unit of data Degree
 Minimum unit of data Depend on the increment system of the reference axis
 Valid data range 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -999999.999 to +999999.999)
 When the second rotation axis is a hypothetical axis (bit 1 (IA2) of parameter No. **19696** is 1), set the rotation angle.

		#7	#6	#5	#4	#3	#2	#1	#0
19696		RFC	WKP			NPC		IA2	IA1
#0	IA1								
#1	IA2								
#3	NPC								
#5	WKP								

If IA1 is 1, set 0 as the controlled-axis number for the first rotation axis (parameter No. **19681**).
 Also, set parameter Nos. **19682** to **19685** on the assumption that there is a rotation axis.
 0: The first rotation axis is an ordinary rotation axis.
 1: The first rotation axis is a hypothetical axis.
 If IA2 is 1, set 0 as the controlled-axis number for the second rotation axis (parameter No. **19686**).
 Also, set parameter Nos. **19687** to **19690** on the assumption that there is a rotation axis.
 0: The second rotation axis is an ordinary rotation axis.
 1: The second rotation axis is a hypothetical axis.
 In Tool posture control for Tool center point control type II, when the change of tool posture at the block end is not done with the parameters (No. **19738**, **19739**), even if the tool posture passes the singular posture,
 0: Program is executed without change of tool posture
 1: The alarm P/S5421 occurs
 For a 5-axis machine having a table rotation axis, as the programming coordinate system for tool tip point control for 5-axis machining or cutter compensation for 5-axis machining:

NOTE

For cutter compensation for 5-axis machining, the setting of this parameter is used only when bit 4 (TBP) of parameter No. **19746** is set to 1.

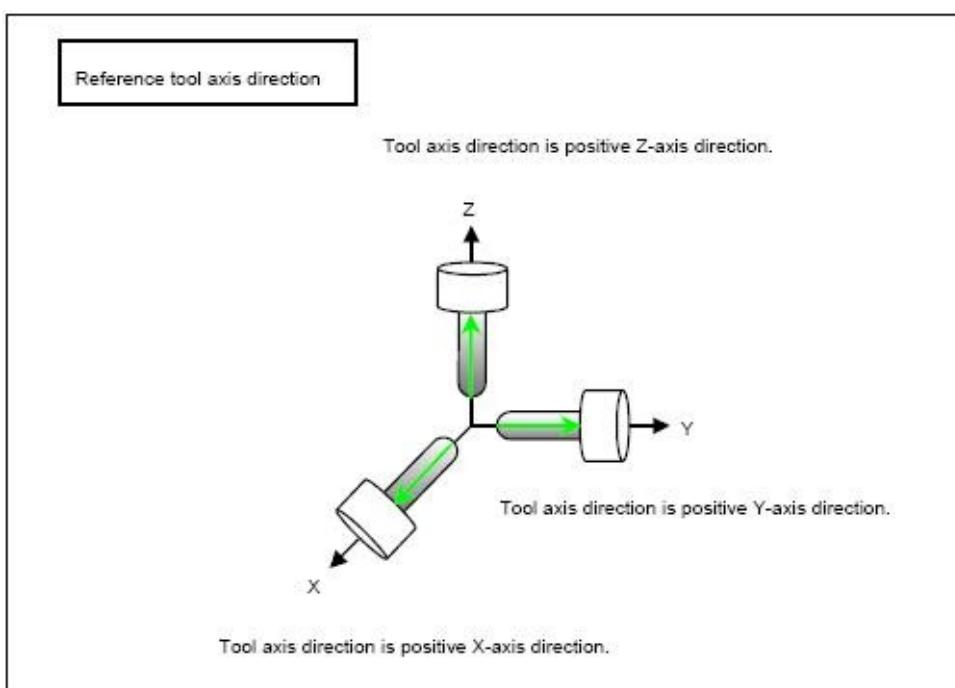
- 0: The table coordinate system (coordinate system fixed on the rotary table) is used.
 1: The workpiece coordinate system is used.

- # 6 RFC In tool center point control for 5-axis machining, when a command that does not move the tool center point with respect to the workpiece is issued, the feedrate of the rotation axis is:
 0: The maximum cutting feedrate (parameter No. **1422**).
 1: A specified feedrate.

19697 Reference tool axis direction

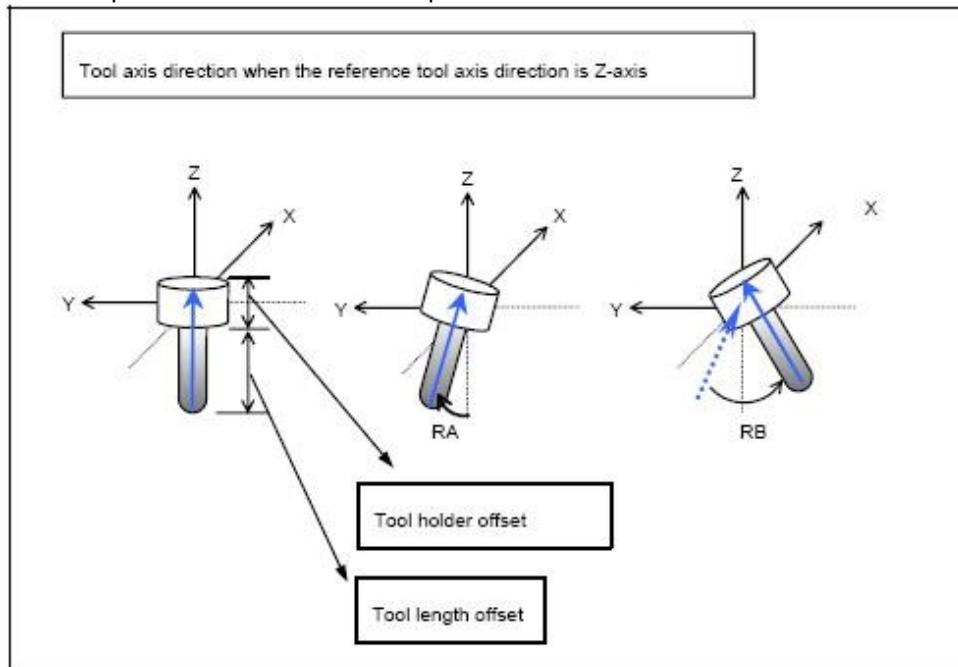
Input type Parameter input
 Data type Byte path
 Valid data range 0 to 3
 Set the tool axis direction in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees. Also, set the tool axis direction in the machine coordinate system in a mechanism in which only the rotation axes for controlling the table are present (there is no rotation axis for controlling the tool).
 1: Positive X-axis direction
 2: Positive Y-axis direction
 3: Positive Z-axis direction

When the reference tool axis direction is neither the X-, Y-, nor Z-axis direction, set the reference direction in this parameter, then set appropriate angles as the reference angle RA and reference angle RB (parameter Nos. 19698 and 19699).



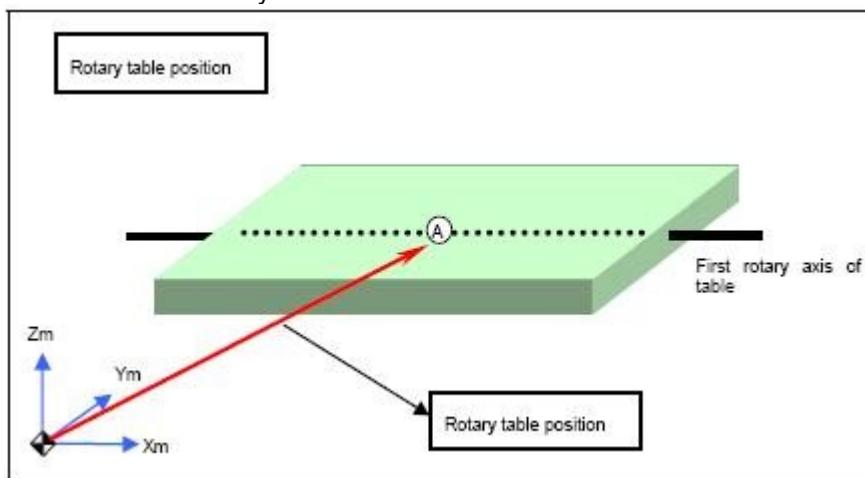
19698	Angle when the reference tool axis direction is tilted (reference angle RA)
19699	Angle when the reference tool axis direction is tilted (reference angle RB)

Input type	Parameter input
Data type	Real path
Unit of data	Degree
Minimum unit of data	Depend on the increment system of the reference axis
Valid data range	9 digit of minimum unit of data (refer to standard parameter setting)
	When the reference tool axis direction (parameter No. 19697) is set to 1, the tool axis is tilted the RA degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction, then the tool axis is tilted the RB degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction.
	When the reference tool axis direction (parameter No. 19697) is set to 2, the tool axis is tilted the RA degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction, then the tool axis is tilted the RB degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction.
	When the reference tool axis direction (parameter No. 19697) is set to 3, the tool axis is tilted the RA degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction, then the tool axis is tilted the RB degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction.



19700	Rotary table position (X-axis of the basic three axes)
19701	Rotary table position (Y-axis of the basic three axes)
19702	Rotary table position (Z-axis of the basic three axes)

Input type Parameter input
 Data type Real path
 Unit of data mm, inch (machine unit)
 Minimum unit of data Depend on the increment system of the applied axis
 Valid data range 9 digit of minimum unit of data (refer to standard parameter setting table (A))
 (When the increment system is IS-B, -999999.999 to +999999.999)
 Set these parameters when parameter No. 19680 is set to 12 or 21. The vector from the origin of the machine coordinate system to point A on the first rotation axis of the table is set as the rotary table position in the machine coordinate system.

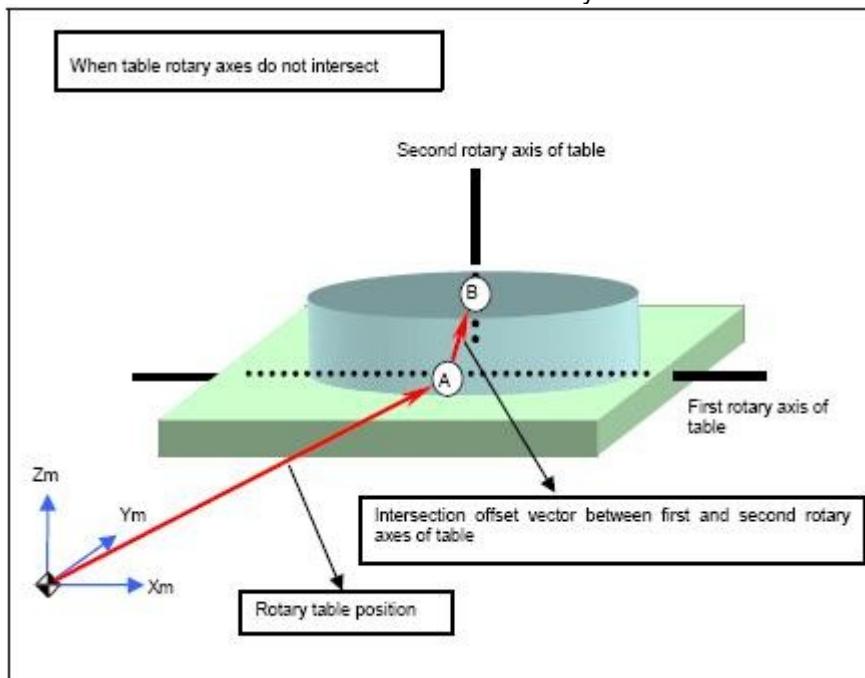


NOTE

As point A, set a position that is easy to measure on the first rotary axis of the table. Set a radius value.

19703	Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)
19704	Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)
19705	Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)

Input type	Parameter input
Data type	Real path
Unit of data	mm, inch (machine unit)
Minimum unit of data	Depend on the increment system of the applied axis
Valid data range	<p>9 digit of minimum unit of data (refer to standard parameter setting table (A)) (When the increment system is IS-B, -999999.999 to +999999.999)</p> <p>Set these parameters when the first rotation axis and second rotation axis of the table do not intersect. These parameters are valid when parameter No. 19680 is set to 12. When the rotation axes for controlling the table are all at 0 degrees, the vector from point A to point B on the second rotation axis of the table is set as the intersection offset vector in the machine coordinate system.</p>



NOTE

As point B, set a position that is easy to measure on the second rotary axis of the table. Set a radius value.

19738	Angle check if tool posture is near singular posture or not
--------------	---

19739	Angle to decide that the tool posture at block end is not changed
--------------	---

19741	Upper limit of the movement range of the first rotation axis
--------------	--

19742	Lower limit of the movement range of the first rotation axis
--------------	--

19743	Upper limit of the movement range of the second rotation axis
--------------	---

19744	Lower limit of the movement range of the second rotation axis
--------------	---

	#7	#6	#5	#4	#3	#2	#1	#0
19746		CRS		TPB				
# 4	TPB	For a 5-axis machine having a table rotation axis, as the programming coordinate system for cutter compensation for 5-axis machining: 0: The workpiece coordinate system is used. 1: The setting of bit 5 (WKP) of parameter No. 19696 is used.						
# 6	CRS	In tool tip point control for 5-axis machining, when the deviation from the path during movement at the specified cutting feedrate or rapid traverse rate is determined to exceed the limit: 0: The feedrate or rapid traverse rate is not decreased. 1: The feedrate or rapid traverse rate is controlled so that the limit of the deviation from the path set in the parameter for the cutting feed or rapid traverse is not exceeded. In the rapid traverse mode, the rapid traverse rate is decreased so that the deviation from the path does not exceed the limit specified in parameter No. 19751 . In the cutting feed mode, the cutting feedrate is decreased so that the deviation from the path does not exceed the limit specified in parameter No. 19752 .						

19751	Limit of the deviation from the path (for rapid traverse)
--------------	---

Input type	Parameter input
Data type	Real path
Unit of data	mm, inch (machine unit)
Minimum of data	Depend on the increment system of the reference axis
Valid data range	9 digit of minimum unit of data (refer to standard parameter setting table (A)) This parameter sets the limit of the deviation from the path in the rapid traverse mode in tool tip point control for 5-axis machining. If the tool moves at the specified rate, the deviation from the path may exceed the value specified in this parameter. In this case, the rate is decreased so that the tool moves along the path. This parameter is valid when bit 6 (CRS) of parameter No. 19746 is set to 1. When 0 is set, the least input increment is assumed to be the limit of the deviation from the path. If a negative value is set, the rapid traverse rate is not decreased.

NOTE

The error generated after the rate is decreased may be smaller than the value set in this parameter depending on the calculation error.

19752 Limit of the deviation from the path (for cutting feed)

Input type	Parameter input
Data type	Real path
Unit of data	mm, inch (machine unit)
Minimum of data	Depend on the increment system of the reference axis
Valid data range	9 digit of minimum unit of data (refer to standard parameter setting table (A)) This parameter sets the limit of the deviation from the path in the rapid traverse mode in tool tip point control for 5-axis machining. If the tool moves at the specified rate, the deviation from the path may exceed the value specified in this parameter. In this case, the rate is decreased so that the tool moves along the path. This parameter is valid when bit 6 (CRS) of parameter No. 19746 is set to 1. When 0 is set, the least input increment is assumed to be the limit of the deviation from the path. If a negative value is set, the rapid traverse rate is not decreased.

NOTE

The error generated after the rate is decreased may be smaller than the value set in this parameter depending on the calculation error.

#7	#6	#5	#4	#3	#2	#1	#0
----	----	----	----	----	----	----	----

19754			INZ					
# 5	INZ							

In tool center point control and 3-dimensional cutter compensation, in case that the programming coordinate system is table coordinate system, the table coordinate system is fixed to the table;
0: when these functions are started.
1: in the state that table rotary axes positions are 0.

3 Documentation Reference

	Specification	Title-
1.	B-63944EN/02	User's Manual
2.	A-799294E	Tool posture control of tool center point control for 5-axis machining
3.	A-90921E	Inverse time feed under tool center point control
4.	A-90501E	Cutting point command
5.	A-90865E	Tilted Working Plane command with guidance
6.	A-90003E	Specifying tilted working plane execution
7.	A-90770E	Manual intervention during tool center point
8.	A-90557EN//01	Additional information for cutter radius compensation / Nose-R compensation
9.	A-79964	Nano smoothing for 5-axis machining
10.	A-79985E	NURBS interpolation for 5-axis machining
11.	A-796344E	Specification of workpiece setting error compensation
12.	A-91055E	Tool Posture Control for G02/G03 (Preliminary)
13.	A-90952E	Improvement about table coordinate system setting in tool center point control and 3D cutter compensation
14.	A-90776E	Expansion of axis move command in tool center point control.

4 Document History

Date	Version	Changes
23.10.2007	02	- Change of Company name - Minor Corrections
28.03.2007	01	Initial release