PROGRAMMING MANUAL

for

MAZATROL FUSION 640MT 5X
MAZATROL FUSION 640MT Pro
MAZATROL FUSION 640M Pro
MAZATROL MATRIX
Tool Tip Point Control Function

MANUAL No. : H734PB0027E

Serial No. :

Before using this machine and equipment, fully understand the contents of this manual to ensure proper operation. Should any questions arise, please ask the nearest Technical Center or Technology Center.

IMPORTANT NOTICE

1. **Be sure to observe the safety precautions described in this manual and the contents of the safety plates on the machine and equipment. Failure may cause serious personal injury or material damage. Please replace any missing safety plates as soon as possible.**
2. **No modifications are to be performed that will affect operation safety. If such modifications are required, please contact the nearest Technical Center or Technology Center.**
3. **For the purpose of explaining the operation of the machine and equipment, some illustrations may not include safety features such as covers, doors, etc. Before operation, make sure all such items are in place.**
4. **This manual was considered complete and accurate at the time of publication, however, due to our desire to constantly improve the quality and specification of all our products, it is subject to change or modification. If you have any questions, please contact the nearest Technical Center or Technology Center.**
5. **Always keep this manual near the machinery for immediate use.**
6. **If a new manual is required, please order from the nearest Technical Center or Technology Center with the manual No. or the machine name, serial No. and manual name.**

Issued by *Manual Publication Section, Yamazaki Mazak Corporation*, Japan

11. 2006

CONTENTS

1 FUNCTION OUTLINE

The tool-tip point control function moves the tool on a five-axis control machine (of tool tilting, table tilting, or tool and table tilting type) so that the tool tip point describes the path as programmed in the coordinate system of the workpiece according to the particular position of the rotational axis.

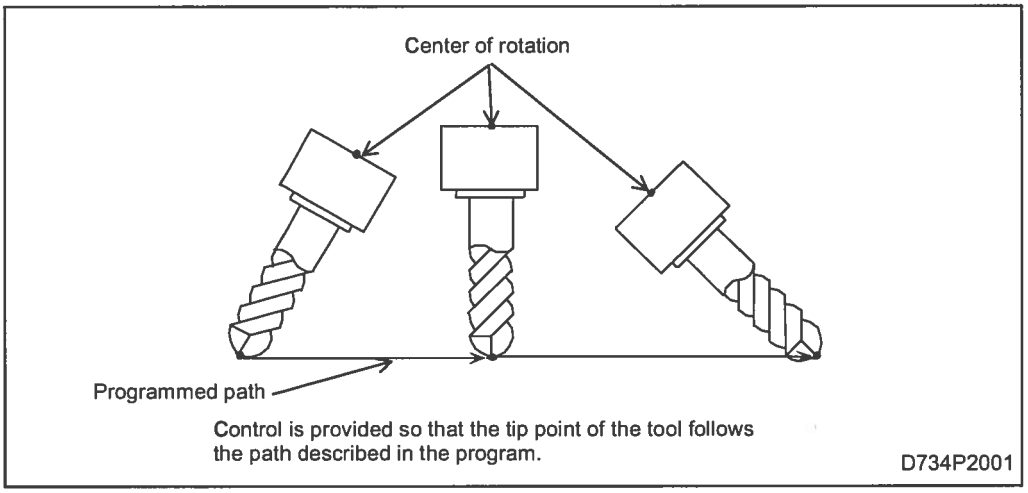


Fig. 1-1

- This function is valid only for five-axis control machines.
- If the required option is not added, giving a tool tip point control command will result in an alarm.

- NOTE -

2 DETAILED DESCRIPTION

The tool tip point control function conducts control so that the center of the tool tip moves along the path programmed either in the workpiece coordinate system for tool tilting type, or in the coordinate system of the workpiece (namely, in a system rotating with the workpiece) for table tilting type.

2-1 Programming Format

Two types of programming formats are available for tool tip point control: type 1 for programming only a tool length offset, and type 2 for programming a tool length offset and the direction (attitude) of the tool axis.

1. Tool tip point control ON

<Type 1>

`G43.4 (Xx Yy Zz Aa Bb Cc) Hh;` ..... Tool tip point control type 1 ON

<Type 2>

`G43.5 (Xx Yy Zz) Ii Jj Kk Hh;` ..... Tool tip point control type 2 ON

x, y, z : Orthogonal coordinate axis motion command
a, b, c : Rotational axis motion command
i, j, k : Direction of tool axis (Position vector from tip point to center of rotation of the tool)
h : Tool length offset number

Note 1: The startup axis movement is executed in the currently active mode of movement.

Note 2: If no (orthogonal or rotational) axis motion command is given in the same block, an independent startup movement occurs according to the particular tool length offset.

Note 3: Do not give any rotational axis motion command in the mode of tool tip point control type 2 (G43.5). Such commands will be simply ignored.

Note 4: An argument of zero (0) for vector components (I, J, K) can be omitted in the mode of G43.5. The position vector of the preceding block will be kept intact if all the three components are omitted.

Note 5: Do not give a command that reverses the direction of the tool with respect to the workpiece. Otherwise an alarm will be caused (**ILLEGAL NUMBER INPUT**).

Note 6: Do not give a (rotational) C-axis motion command with the tool axis being parallel to the axis of the turntable. Otherwise an alarm will be caused (**ILLEGAL NUMBER INPUT**).

2. Tool tip point control OFF (cancellation)

`G49 ;` ..... Tool tip point control OFF

Other G-codes in group 8

G43/G44 (Tool length offset in the plus/minus direction)

Note 1: It depends upon the setting of a parameter whether or not the cancellation causes an axis motion (in the currently active mode) of canceling the tool length offset.

Note 2: The cancellation command G49 must be given with no other instruction codes.

2-2 Programming Coordinate System

In the mode of tool tip point control, specify in a sequence of motion blocks the ending positions of the tool tip point in the programming coordinate system.

Two types of coordinate system are available (according to the particular parameter setting) for describing the motion of the tool tip point: a table coordinate system (system fixed to the table) or a workpiece coordinate system.

2-2-1 Programming with a table coordinate system

With the relevant parameter set to "0", selecting the tool tip point control mode includes establishment of a programming coordinate system by fixing the current workpiece coordinate system to the table. The table coordinate system will rotate as the table rotates. It will not change in position, however, as the direction of the tool axis changes. Subsequent X-, Y- and Z-axis motion commands will be executed with respect to the table coordinate system.

The initial state of the table coordinate system refers to the current table position, or is to be specified by a table rotation command given in the block of G43.4 or G43.5.

2-2-2 Programming with a workpiece coordinate system

With the relevant parameter set to "1", the current workpiece coordinate system is to be used for describing the movement of the tool tip position. The coordinate system in this case will not rotate as the table rotates.

Subsequent X-, Y- and Z-axis commands will be of linear motion with respect to the table (workpiece). Specify the ending positions along the orthogonal axes always taking into consideration the particular angle of the table rotation.

2-3 Startup

2-3-1 Independent startup

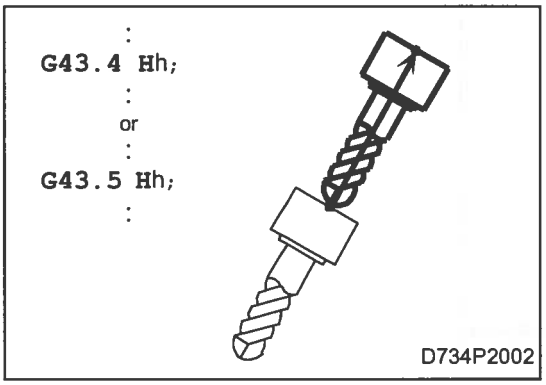
<Type 1>

```
G43.4 Hh;
```

<Type 2>

```
G43.5 Hh;
```

In accordance with the current system of workpiece coordinates (at the time of selecting the tool tip point control mode), a shifting motion occurs in the direction of the tool axis through the particular distance of length offset.



2-3-2 Startup with a motion command

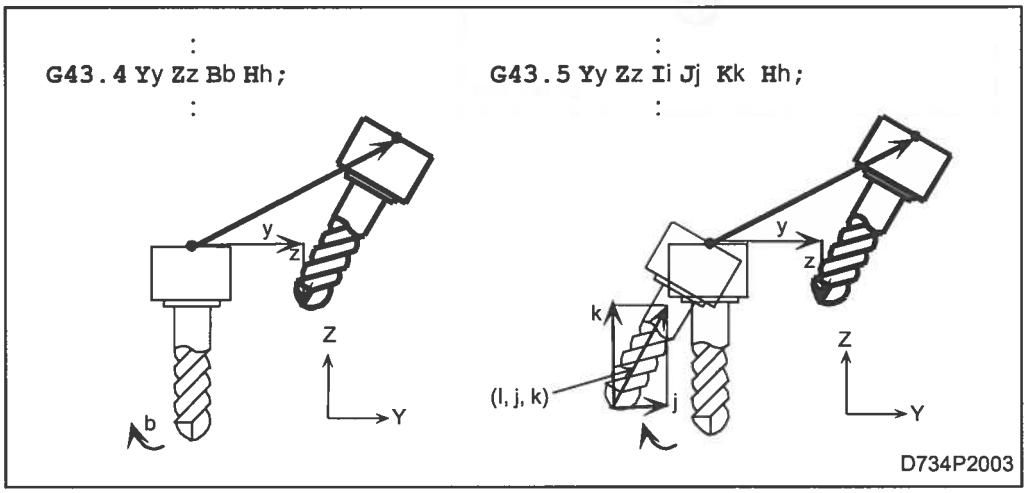
<Type 1>

```
G43.4 Xx Yy Zz Aa Bb Cc Hh;
```

<Type 2>

```
G43.5 Xx Yy Zz Ii Jj Kk Hh;
```

In accordance with the rotation specified (in a rotational axis motion command or position vector components), a linear axis movement occurs with a shifting motion in the direction of the tool axis through the particular distance of length offset.



2-4 Cancellation

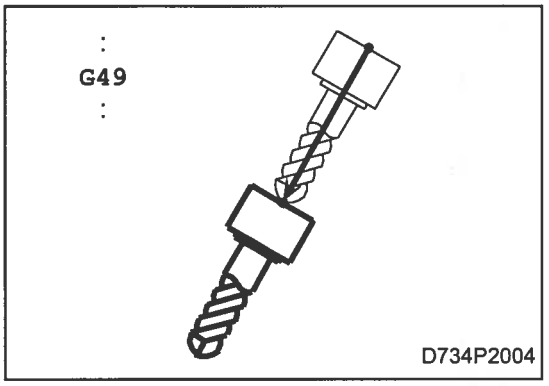
2-4-1 Independent cancellation

<Type 1> and <Type 2>

A parameter is provided for the selection of whether or not the cancellation includes the corresponding axis movement.

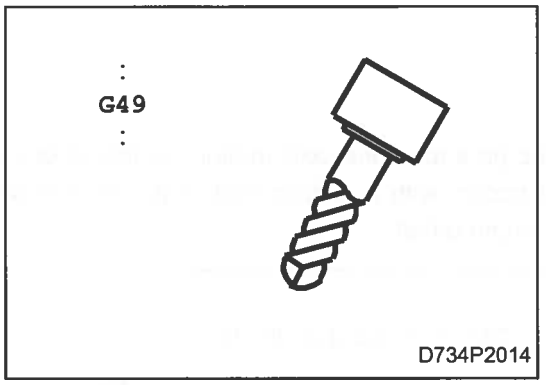
1. Cancellation with axis movement

The tool tip point control mode is cancelled with a shifting motion in the direction of the tool axis for canceling the particular amount of length offset.



2. Cancellation without axis movement

The tool tip point control mode is cancelled without any axis motions being caused by the cancellation block.



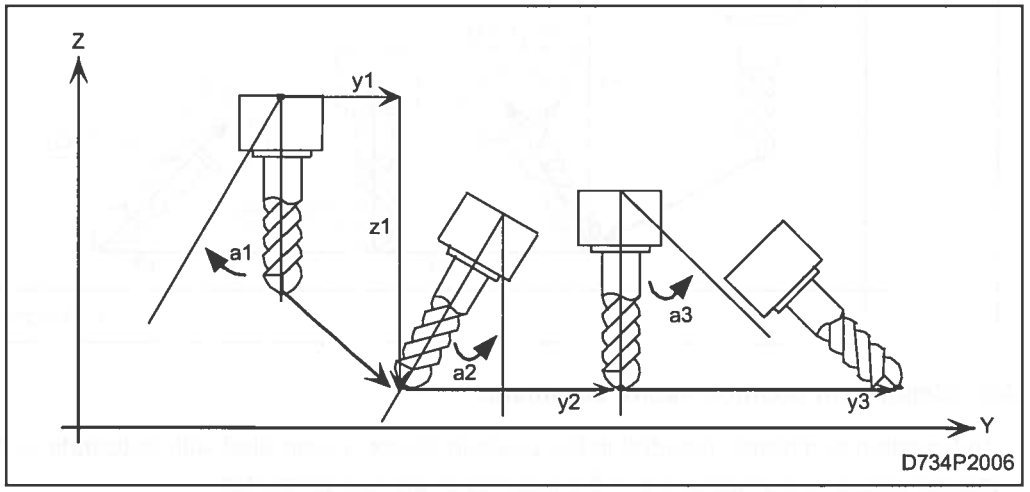
2-5 Operation in the Mode of Tool Tip Point Control

2-5-1 Tool tip point control type 1

1. Orthogonal and rotational axis motion commands given in one block

The motion is controlled in order that the center of the tool tip may describe the programmed path.

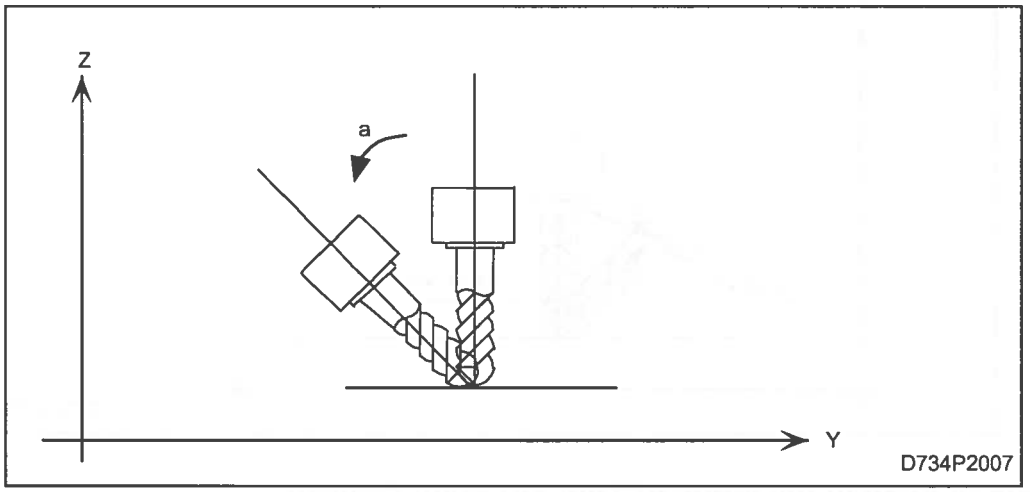
```
  :
G91 G43.4 Yy1 Zz1 Aa1 Hh;
Yy2 Aa2;
Yy3 Aa3;
  :
```



2. An independent rotational axis motion command

The explicit command of rotation is executed with automatic orthogonal axis movements in order not to move the position of the tool tip center.

```
G43.4 Hh;
  :
Aa;
  :
```

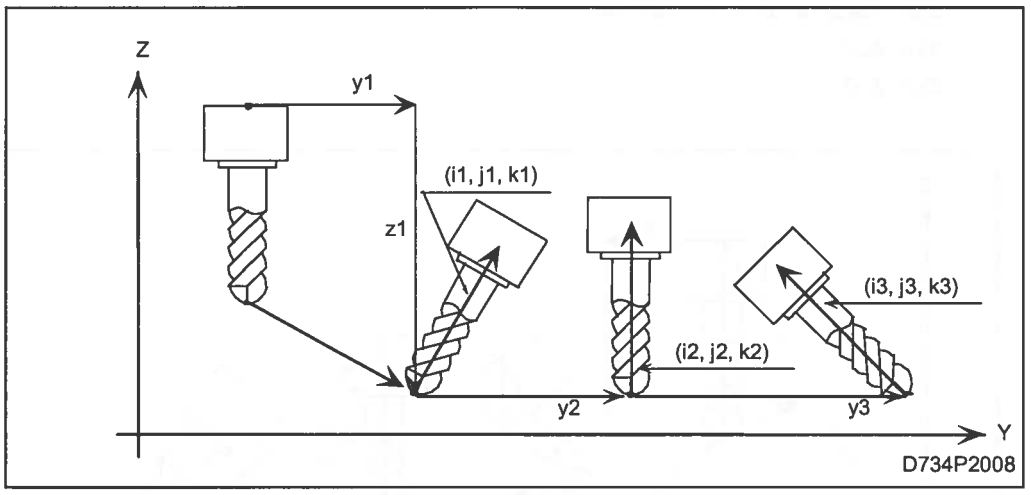


2-5-2 Tool tip point control type 2

1. Orthogonal axis motion and position vector commands given in one block

The motion is controlled in order that the center of the tool tip may describe the programmed path.

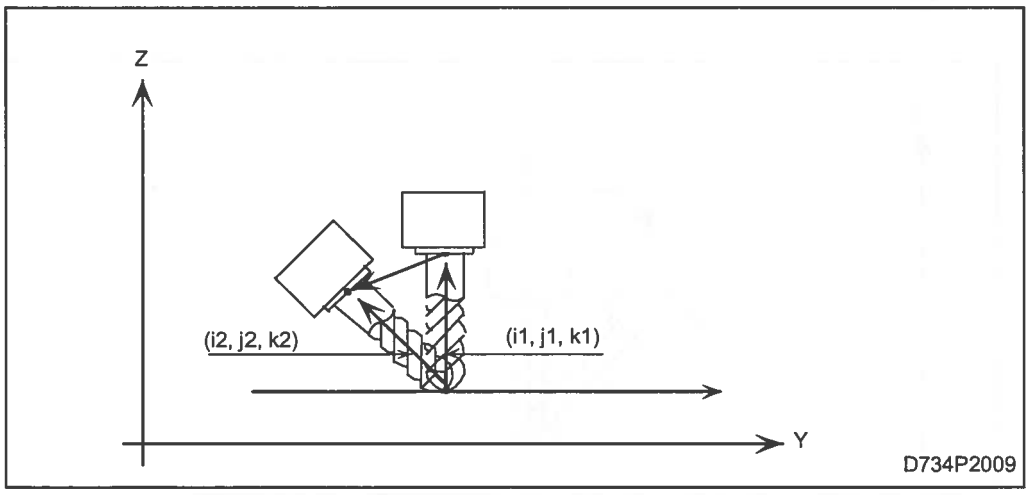
```
G91 G43.5 Yy1 Zz1 Ii1 Jj1 Kk1;
Yy2 Ii2 Jj2 Kk2;
Yy3 Ii3 Jj3 Kk3;
```



2. An independent position vector command

The rotation command included in the position vector is executed with automatic orthogonal axis movements in order not to move the position of the tool tip center.

```
  :
G43.5 Ii1 Jj1 Kk1 Hh;
  :
  :
Ii2 Jj2 Kk2;
  :
```

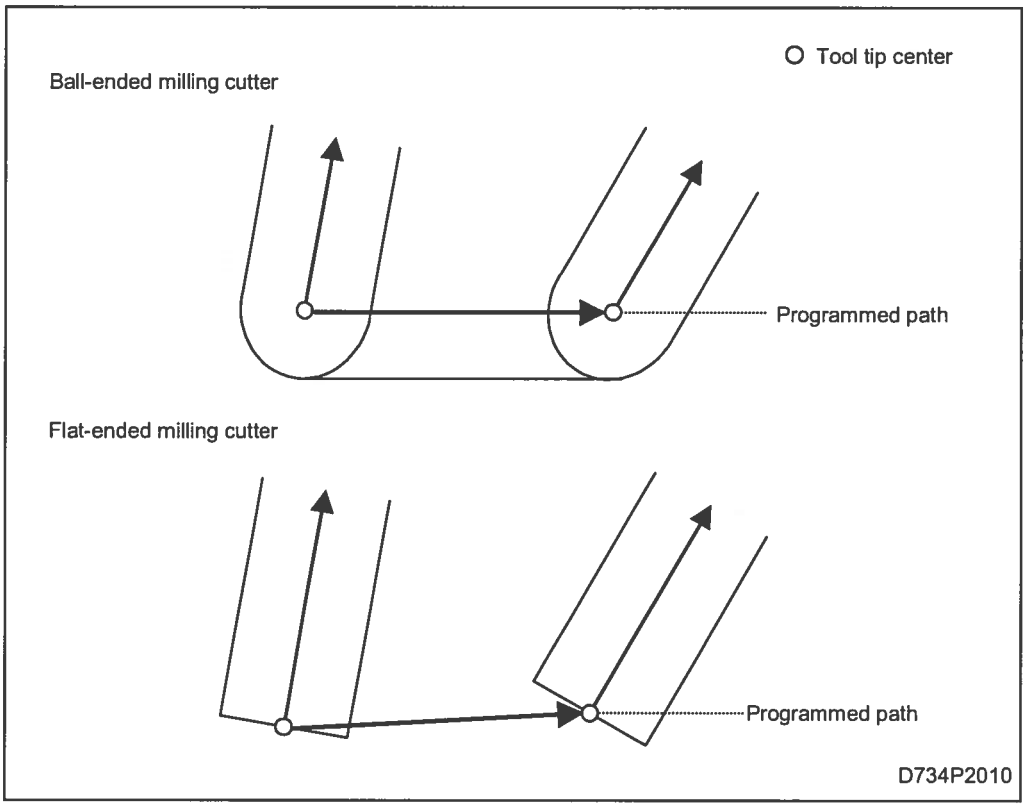


2-6 Rate of Feed in the Mode of Tool Tip Point Control

The feed is controlled so that the center of the tool tip may move at the specified rate.

2-7 Positions to be Programmed

Describe in the program the movement of the center of the tool tip.



- NOTE -

3 RELATIONSHIP TO OTHER FUNCTIONS

3-1 Relationship to Other Preparatory Functions

The tables below indicate the compatibility of each G-code with the tool tip point control function in the two columns on the right.

[1] Is the G-code available in the tool tip point control mode?
O: Yes, ×: No.

[2] Is the tool tip point control mode selectable under the condition of the G-code?
O: Yes, ×: No (— for non-modal codes)

1. Functions for the M640MT 5X, M640MT Pro and MATRIX(Series T)

Function	G-code	[1]	[2]
Positioning	G00	O	O
Linear interpolation	G01	O	O
Threading with C-axis interpolation	G01.1	×	×
Circular interpolation	G02/G03	×	×
Helical interpolation	G02/G03	×	×
Spiral interpolation	G02.1/G03.1	×	×
Dwell	G04	O	—
Cylindrical interpolation	G07.1	×	×
Exact-stop check	G09	O	—
Data setting	G10	×	—
Data setting cancel	G11	×	—
Polar coordinate interpolation	G12.1	×	×
Polar coordinate interpolation OFF	G13.1	×	×
Plane selection	G16 to G19	×	—
Inch/Metric data input	G20/G21	×	O
Stored stroke check ON/OFF	G22/G23	O	O
Reference point return check	G27	×	—
Return to reference point	G28	×	—
Return from reference point	G29	×	—
Return to 2., 3. and 4. reference point	G30	×	—
Return to floating reference point	G30.1	×	—
Skip function	G31	×	—
Threading (straight, taper)	G32, G33	×	×
Variable-lead threading	G34	×	×
Automatic tool correction	G36/G37	×	—
Preparation/Calculation for measurement	G36.5/G37.5	×	—
Nose radius compensation OFF/left/right/automatic	G40/G41/G42/G46	×	×
Setting of coordinate system/max. spindle speed	G50/G92	×	—
Polygonal machining OFF/ON	G50.2/G51.2	×	×
Local coordinate system selection	G52	×	—
Machine coordinate system selection	G53	×	—
MAZATROL coordinate system OFF/ON	G52.5/G53.5	×	—
Selection of Workpiece coordinate system 1 to 6	G54 to G59	×	×
Exact-stop check mode	G61	O	O

Function	G-code	[1]	[2]
Geometry compensation	G61.1	×	×
Automatic corner override mode	G62	×	×
Cutting mode	G64	O	O
User macro simple call	G65	×	×
User macro modal call	G66	×	×
User macro modal call OFF	G67	×	×
Subprogram call	M98/M99	O	—
Mirror image for opposite turret ON/OFF	G68/G69	×	×
Programmed coordinates rotation ON/OFF	G68.5/G69.5	×	×
Fixed cycle	G70 to G89	×	×
Absolute/Incremental data input	G90/G91	O	O
Asynchronous feed (feed per minute)	G94	O	O
Synchronous feed (feed per revolution)	G95	×	×
Spindle control for constant cutting speed ON/OFF	G96/G97	×	×
2 processes in 1 program	G109	O	—
Cross machining control axis selection ON/OFF	G110/G111	×	×
Output of M/S/T/B command to opposite system	G112	O	—
Hob milling mode cancel	G113	O	×
Hob milling mode	G114.3	×	×
Swiss type machining mode ON/OFF	G120/G121	×	×
Polar coordinate input ON/OFF	G122/G123	×	×
Radius data input for X-axis ON/OFF	G122.1/G123.1	×	O

2. Functions for the M640M Pro and MATRIX(Series M)

Function	G-code	[1]	[2]
Positioning	G00	O	O
Linear interpolation	G01	O	O
Circular interpolation	G02/G03	Note 3	×
Helical interpolation	G02/G03	×	×
Spiral interpolation	G02.1/G03.1	×	×
Dwell	G04	O	—
High-speed machining mode	G05	×	×
Fine-spline interpolation	G06.1	×	×
NURBS interpolation	G06.2	×	×
Virtual-axis interpolation	G07	×	×
Exact-stop check	G09	O	—
Programmed parameter input	G10	×	—
Plane selection	G17 to G19	O	—
Inch/Metric data input	G20/G21	×	O
Pre-move stroke check ON/OFF	G22/G23	O	O
Reference point return check	G27	×	—
Return to reference point	G28	×	—
Return from reference point	G29	×	—
Return to 2., 3. and 4. reference point	G30	×	—
Skip function	G31	×	—
Multi-step skip function 1 to 3	G31.1 to G31.3	×	—

Function	G-code	[1]	[2]
Threading	G33	×	×
Automatic tool length measurement	G37	×	—
Tool radius compensation	G38 to G42	×	—
Tool length offset	G43/G44/G49	Note 1	Note 2
Tool position offset	G45 to G48	×	×
Scaling OFF/ON	G50/G51	×	×
G-code mirror image OFF/ON	G50.1/G51.1	×	×
Local coordinate system selection	G52	×	—
Machine coordinate system selection	G53	×	—
Selection of workpiece coordinate systems	G54 to G59	×	O
Selection of additional workpiece coordinate systems	G54.1	×	O
One-way positioning	G60	×	—
Exact-stop check mode	G61	O	O
Shape correction	G61.1	×	×
Automatic corner override	G62	×	×
Tapping mode	G63	×	×
Cutting mode	G64	O	O
User macro simple call	G65	×	×
User macro modal call A	G66	×	×
User macro modal call B	G66.1	×	×
User macro modal call OFF	G67	×	×
Subprogram call	M98/M99	O	—
Programmed coordinates rotation ON/OFF	G68/G69	×	×
Fixed cycle	G70 to G89	×	×
Absolute/Incremental data input	G90/G91	O	O
Machine coordinate system setting	G92	×	—
Workpiece coordinate system rotation	G92.5	×	—
Inverse time feed	G93	O	O
Asynchronous feed (feed per minute)	G94	O	O
Synchronous feed (feed per revolution)	G95	×	×

Note 1: The tool tip point control mode is replaced by the tool length offset mode.

Note 2: The tool length offset mode is replaced by the tool tip point control mode.

Note 3: Available only for "Type 1" (G43.4) and "Programming with a workpiece coordinate system" (**F85**, bit 2 = 1).
Moreover, commands for the rotational axes B and C cannot be given in the mode of circular interpolation.

3-2 On the Use of the MAZATROL Tool Data (for the M640M Pro and MATRIX(Series M))

The data settings of the **TOOL DATA** display (prepared for the execution of MAZATROL programs) can also be used for the tool tip point control. The table below indicates those usage patterns [1] to [4] of the externally stored tool offset data items which are applied to the tool tip point control according to the settings of the relevant parameters (**F93** bit 3 and **F94** bit 7).

Table 3-1 Tool offset data items used according to the parameter settings

Pattern	Data items used (Display and Data item names)		Parameter		Programming method
			F94 bit 7	**F93** bit 3	
[1]	**TOOL OFFSET**	Offset data items	0	0	G43.4/G43.5 with H-code
[2]	**TOOL DATA**	LENGTH	1	1	T-code
		LENGTH + LENG. No. LENGTH + LENG. CO.			T-code + H-code
[3]	**TOOL DATA**	LENG. No. LENG. CO.	1	0	G43.4/G43.5 with H-code
[4]	**TOOL OFFSET** + **TOOL DATA**	Offset data items + LENGTH	0	1	G43.4/G43.5 with H-code + T-code

3-3 Change between Tip Control and Length Offset (for the M640M Pro Only)

3-3-1 Immediate change between G43.4/G43.5 and G43/G44 (without using G49)

An immediate change of one mode for the other with a motion command added causes the "control point" to be moved to the specified position. The control point here refers to the real tool tip point in the case of G43.4 and G43.5 or, for G43 and G44, to an imaginary tip point which corresponds to a B-axis angle of 0°.
An independent change (without axis motion commands) does not include any change at all in axis positions, indeed, but causes the POSITION values (indicated with respect to the workpiece coordinate system) to be changed because of the above-mentioned difference in the meaning of the control point unless the current B-axis position is 0° (see Subsection 3-3-2 for details).

Shown below is a programming example with an explanatory figure for the use of the MAZATROL tool data (stored on the **TOOL DATA** display).

```
N01  T01 T02 M6              ATC includes selection of length offset function.  (Note 1)
N02  G01 X_Y_Z_F_
...                                Machining with length offset function (G43)
...
N10  G43.4 Xx1 Yy1 Zz1 Bb1   Tool tip point control ON                           (Note 2)
N11  G01 X_Y_Z_B_C_
...                                Machining with tip point control (G43.4)
...
N20  G43 Xx2 Yy2 Zz2         Tool length offset ON                             (Note 2)
N21  G01 X_Y_Z_B0
...                                Machining with length offset function (G43)
...
```

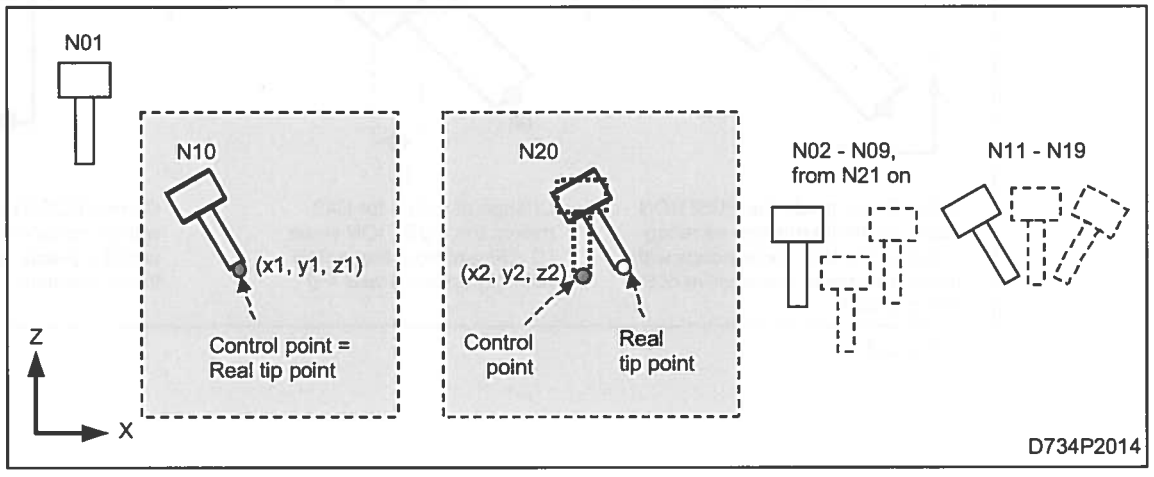


Fig. 3-1

Note 1: When **F93** bit 3 = 1, the length offset function is automatically made active by each ATC operation according to the new tool's LENGTH value on the **TOOL DATA** display.

Note 2: Add an H-code as required to use as the offset amount the sum of the LENGTH value and another related setting (see Table 3-1 for details).

3-3-2 POSITION counting displayed on the screen

An immediate (without using G49) and independent (without axis motion commands) change between G43.4/G43.5 and G43/G44 brings about no actual machine motion (axis movements), but causes the POSITION values on the display to be changed unless the current B-axis position is 0°.

1. Change of G43/G44 for G43.4/G43.5

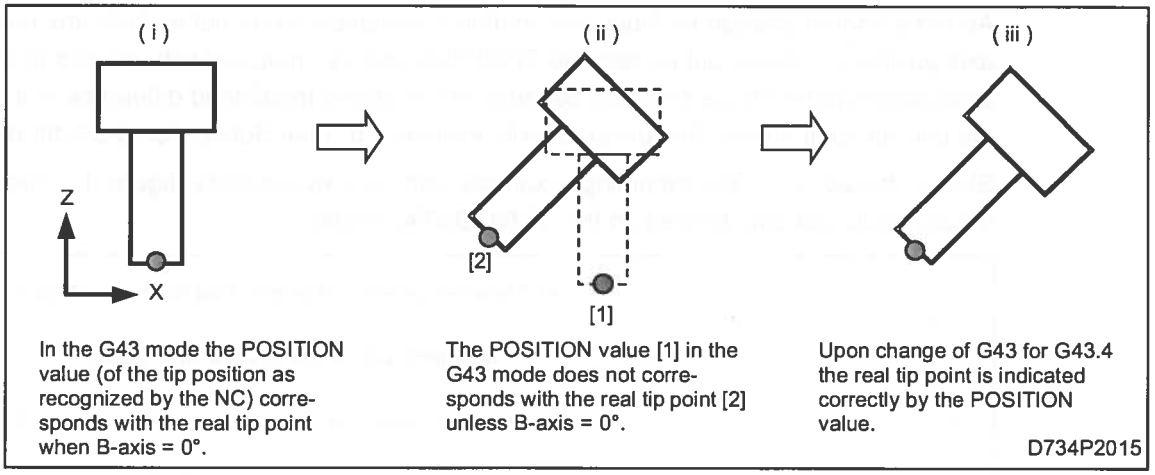


Fig. 3-2

2. Change of G43.4/G43.5 for G43/G44

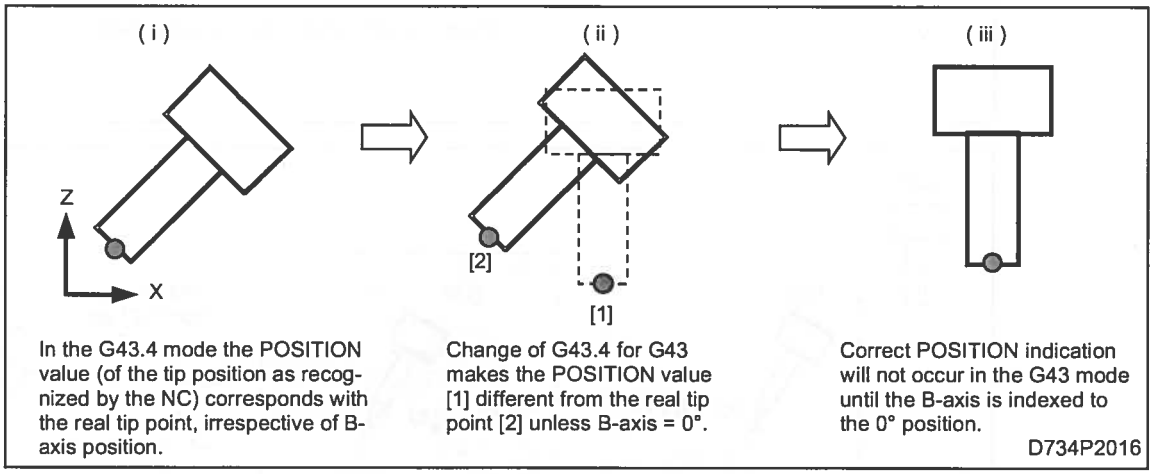


Fig. 3-3

3-3-3 Change between G43.4/G43.5 and G43/G44 by means of G49

The execution of the cancellation command G49 includes axis movements for canceling the offset amount in general (see Note 4 below). Add, therefore, a motion command as required for a safety position before the G49 command given as a means of mode change.

Shown below is a programming example with an explanatory figure for the use of the MAZATROL tool data (stored on the **TOOL DATA** display).

```
N01  T01 T02 M6              ATC includes selection of length offset function.   (Note 1)
N02  G01 X_Y_Z_F_
...                             Machining with length offset function (G43)
...
N08  G0 Xx1 Yy1 Zz1          Positioning to a safety position for canceling the length offset function.
N09  G49                        Canceling the length offset function in the safety position.
N10  G0 Xx2 Yy2 Zz2 Bb2 Cc2  Positioning to a safety position for selecting the tip point control.
N11  G43.4                      Selecting the tip point control in the safety position.   (Note 1, 2)
N12  G01 X_Y_Z_B_C_
...                             Machining with tip point control (G43.4)
...
N18  G0 Xx3 Yy3 Zz3 Bb3 Cc3  Positioning to a safety position for canceling the tip point control.
N19  G49                        Canceling the tip point control in the safety position.   (Note 4)
N20  G0 Xx4 Yy4 Zz4 Bb4 Cc4  Positioning to a safety position for selecting the length offset function.
N21  G43                        Selecting the length offset function in the safety position.   (Note 1, 3)
N22  G01 X_Y_Z_B0
...                             Machining with length offset function (G43)
...
```

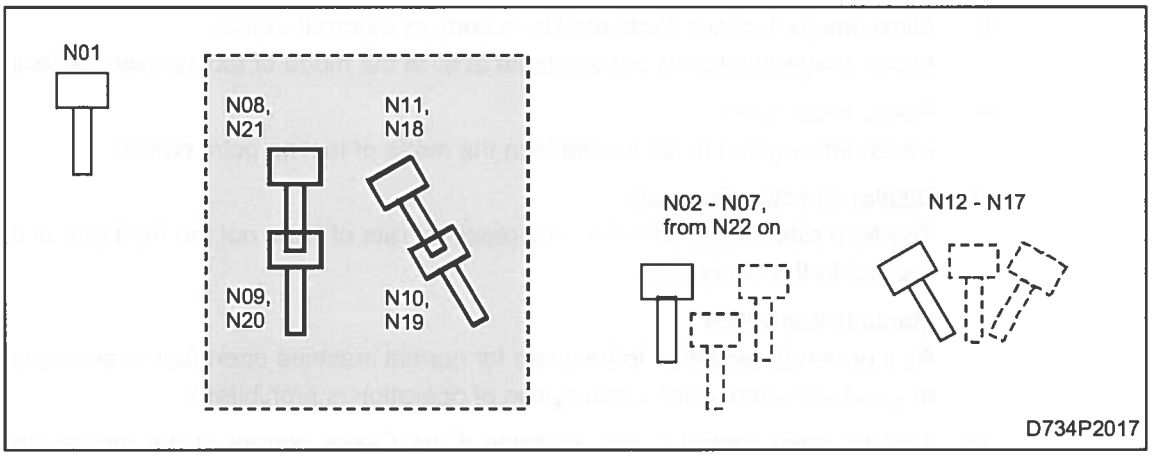


Fig. 3-4

Note 1: When **F93** bit 3 = 1, the length offset function is automatically made active by each ATC operation according to the new tool's LENGTH value on the **TOOL DATA** display.

Note 2: Add an H-code as required to use as the offset amount the sum of the LENGTH value and another related setting (see Table 3-1 for details).

Note 3: Irrespective of the related parameters (**F94** bit 7 and **F93** bit 3), the execution of a G49 command always clears the currently used offset amount. Do not fail, therefore, to give a G43 command or a tool change command (T_T_M6) again as required to replace the tip point control when the automatic selection of the length offset function is used (with **F93** bit 3 = 1).

Note 4: Axis movements for canceling the offset amount do not occur by the execution of an independent command of G49 unless **F114** bit 1 = 0.

3-4 Restrictions

1. The selection code of tool tip point control (G43.4/G43.5) must not be given together with any other G-code.
2. Calculation of the machining time
   The machining time cannot be calculated accurately for a program containing tool tip point control commands.
3. Tracing
   It is the movement of the tool tip point that is traced in the mode of tool tip point control.
4. Tool path check
   Tool path check function cannot be applied to a program containing tool tip point control commands.
5. Restart
   As for the relationship between tool tip point control and restarting function, always specify as the restarting position a block in the cancellation mode of tool tip point control (G49).
   A restarting operation from within the mode of G43.4 would cause the initial approach to the starting point to be conducted improperly under tool tip point control.
6. Resetting
   The mode of tool tip point control is cancelled by resetting.
7. Corner chamfering/rounding
   Corner chamfering or rounding commands are not available in the mode of tool tip point control.
8. Mirror image function (activated by a code or external switch)
   Mirror image function is not available at all in the mode of tool tip point control.
9. Macro interruption
   Macro interruption is not available in the mode of tool tip point control.
10. Display of actual feed rate
    The feed rate displayed is the final resultant rate of feed, not the feed rate at the tool tip with respect to the workpiece.
11. Manual interruption
    As a general precaution to be taken for normal machine operation, resetting is required after any manual interruption (resumption of operation is prohibited).
12. Tool tip point control is not available if the C-axis control of the turning spindle No. 2 is concerned (on accordingly executed machines).

4 RELATED PARAMETERS

4-1 For the M640MT 5X

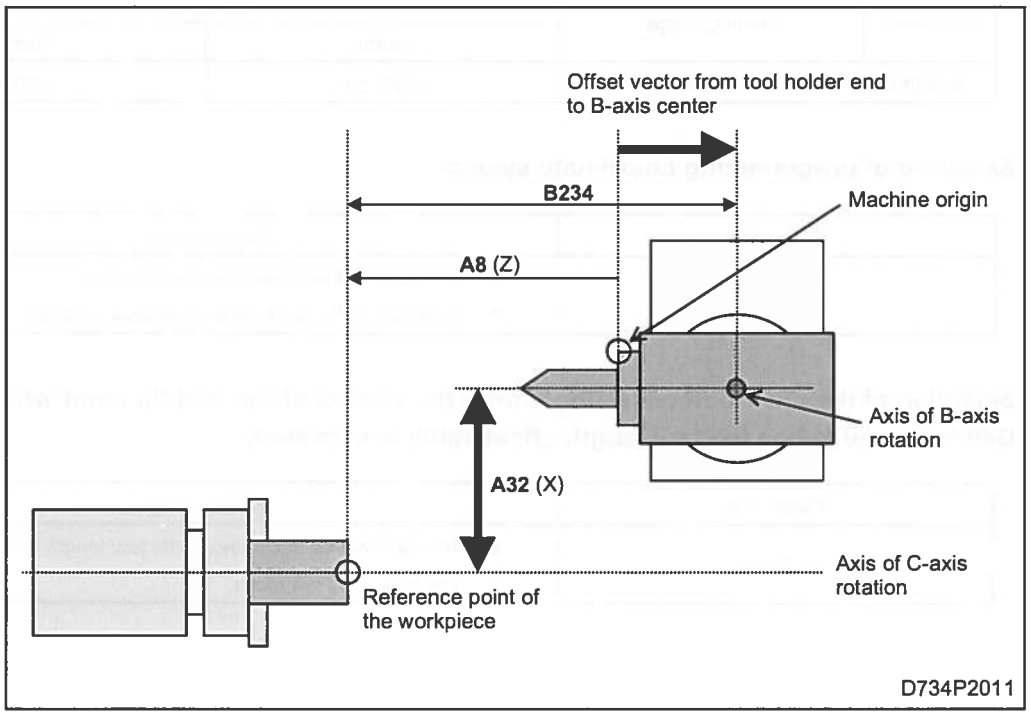


Fig. 4-1

1. Offset vector from tool holder end to B-axis center

Enter the Z-axis reference position of the workpiece in parameter **A8** and the Z-axis position of the axis of B-axis rotation in **B234**.

The value of [**A8** + **B234**] is used as the offset vector from tool holder end to B-axis center.

Note: Use the system of machine coordinates where the origin is set to the end face of the tool holder.

Parameter	Setting range	Setting unit	
		Metric	Inch
A8 (Z)	–99999999 to 99999999	0.001 mm	0.0001 in.
B234	0 to 32767	0.001 mm	0.0001 in.

2. Limit of feed rate for the mode of tool tip point control

Set the maximum admissible rate of cutting feed for the mode of tool tip point control. The cutting feed during tool tip point control will be limited by parameter **A4** (special limit of feed rate) if it is lower than the setting of parameter **A2**.

Note: Set zero (0) in **A2** to make the parameter invalid. If both parameters **A4** and **A2** are set to zero (0), then parameter **A1** (for rapid traverse) serves as the speed limit in question.

Parameter	Setting range	Axis type	Setting unit	
			Metric	Inch
A2	0 to 240000	Linear	1 mm/min	0.1 in./min
		Rotational	1 deg/min	1 deg/min

3. X-axis position of the axis of C-axis rotation (axis of the turning spindle)

Set in a diameter value the X-axis distance between the axis of C-axis rotation and that of B-axis rotation, as measured with the machine components being set at the origin of coordinates.

Parameter	Setting range	Setting unit	
		Metric	Inch
A32 (X)	0 to 99999999	0.001 mm	0.0001 in.

4. Selection of programming coordinate system

Parameter	Setting range
P12 bit 7	0: Selection of the table coordinate system 1: Selection of the workpiece coordinate system

5. Selection of the operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled)

Parameter	Setting range
P12 bit 6	0: The axis moves according to the tool length offset value. 1: The axis does not move.

4-2 For the M640MT Pro

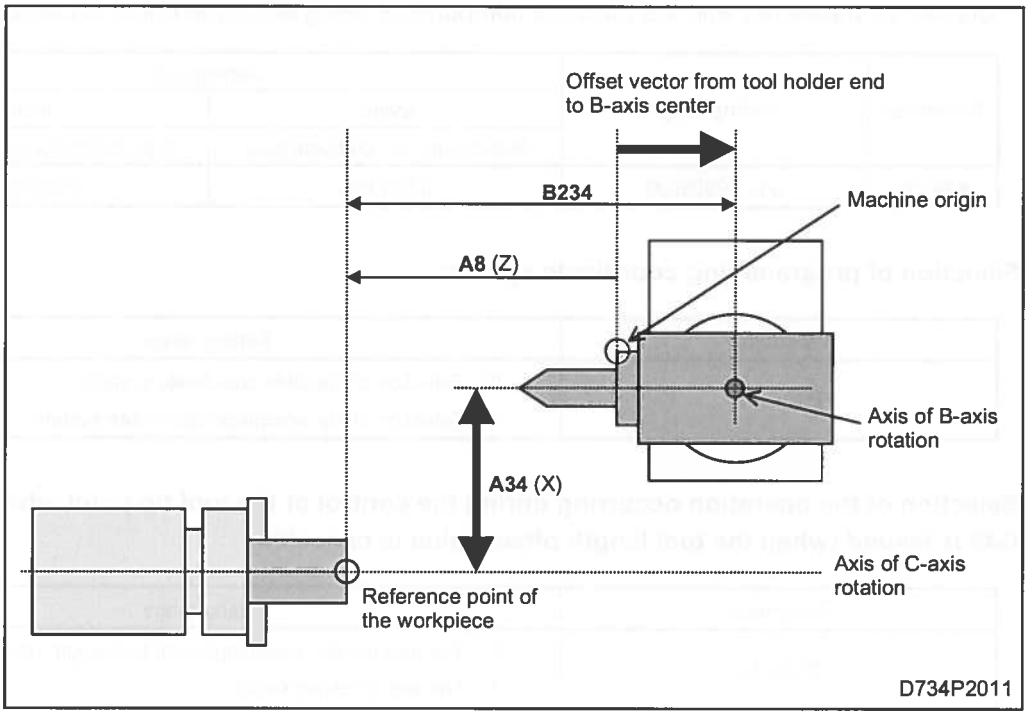


Fig. 4-2

1. Offset vector from tool holder end to B-axis center

Enter the Z-axis reference position of the workpiece in parameter **A8** and the Z-axis position of the axis of B-axis rotation in **B234**.

The value of [**A8** + **B234**] is used as the offset vector from tool holder end to B-axis center.

Note: Use the system of machine coordinates where the origin is set to the end face of the tool holder.

Parameter	Setting range	Setting unit	
		Metric	Inch
		Sub-micron for rotational axes	Sub-micron for rotational axes
A8 (Z)	–99999999 to 99999999	0.001 mm	0.0001 in.
B234	0 to 32767	0.001 mm	0.0001 in.

2. Limit of feed rate for the mode of tool tip point control

Set the maximum admissible rate of cutting feed for the mode of tool tip point control. The cutting feed during tool tip point control will be limited by parameter **A4** (special limit of feed rate) if it is lower than the setting of parameter **A2**.

Note: Set zero (0) in **A2** to make the parameter invalid. If both parameters **A4** and **A2** are set to zero (0), then parameter **A1** (for rapid traverse) serves as the speed limit in question.

Parameter	Setting range	Axis type	Setting unit	
			Metric	Inch
			Sub-micron for rotational axes	Sub-micron for rotational axes
A2	0 to 240000	Linear	1 mm/min	0.1 in./min
		Rotational	1 deg/min	1 deg/min

3. X-axis position of the axis of C-axis rotation (axis of the turning spindle)

Set in a diameter value the X-axis distance between the axis of C-axis rotation and that of B-axis rotation, as measured with the machine components being set at the origin of coordinates.

Parameter	Setting range	Setting unit	
		Metric	Inch
		Sub-micron for rotational axes	Sub-micron for rotational axes
A34 (X)	0 to 99999999	0.001 mm	0.0001 in.

4. Selection of programming coordinate system

Parameter	Setting range
P12 bit 7	0: Selection of the table coordinate system 1: Selection of the workpiece coordinate system

5. Selection of the operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled)

Parameter	Setting range
P12 bit 6	0: The axis moves according to the tool length offset value. 1: The axis does not move.

4-3 For the M640M Pro

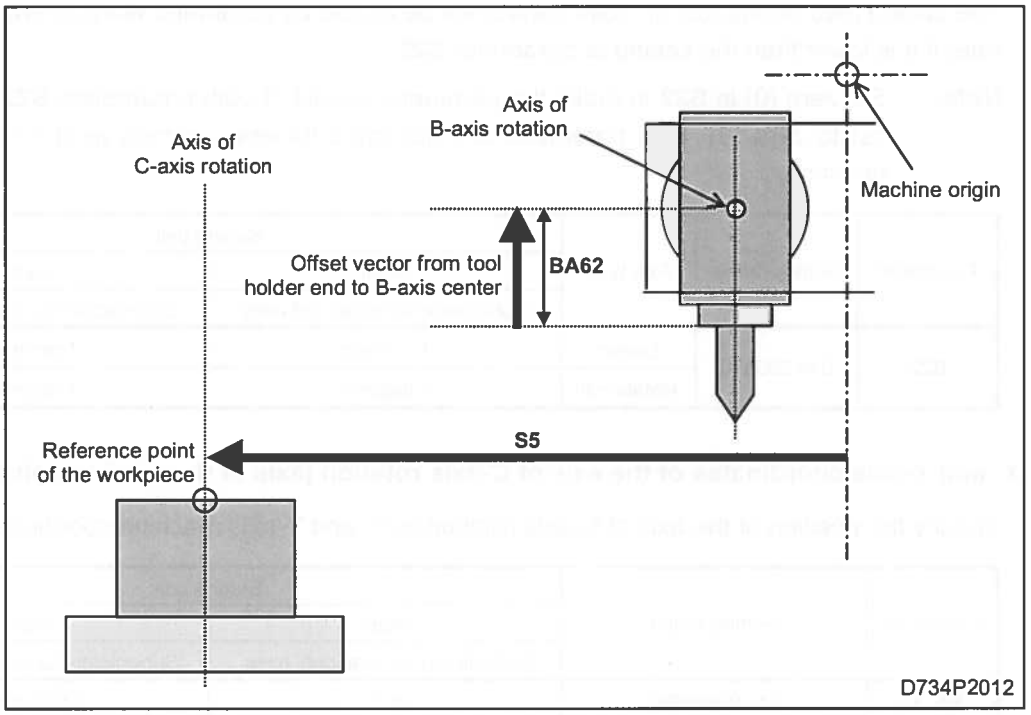


Fig. 4-3 Type V

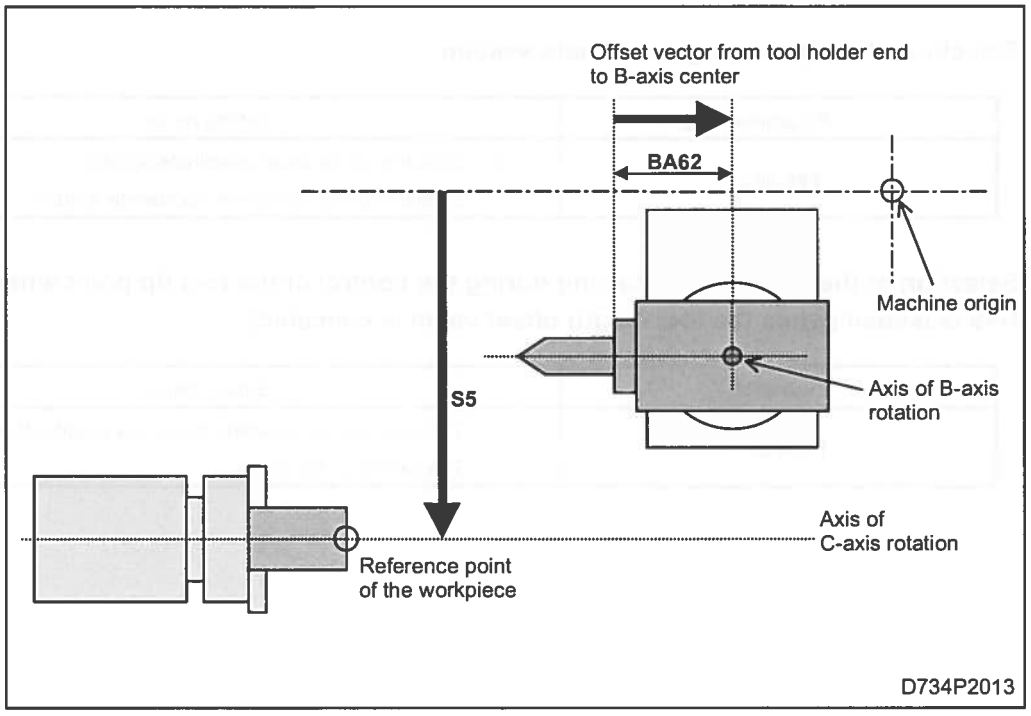


Fig. 4-4 Type H

1. Limit of feed rate for the mode of tool tip point control

Set the maximum admissible rate of cutting feed for the mode of tool tip point control. The cutting feed during tool tip point control will be limited by parameter **M3** (general limit of feed rate) if it is lower than the setting of parameter **S22**.

Note: Set zero (0) in **S22** to make the parameter invalid. If both parameters **S22** and **M3** are set to zero (0), then parameter **M1** (for rapid traverse) serves as the speed limit in question.

Parameter	Setting range	Axis type	Setting unit	
			Metric	Inch
			Sub-micron for rotational axes	Sub-micron for rotational axes
S22	0 to 200000	Linear	1 mm/min	1 mm/min
		Rotational	1 deg/min	1 deg/min

2. X- and Y-axis coordinates of the axis of C-axis rotation (axis of the turning spindle)

Specify the position of the axis of C-axis rotation in X- and Y-axis machine coordinates.

Parameter	Setting range	Setting unit	
		Metric	Inch
		Sub-micron for rotational axes	Sub-micron for rotational axes
S5 (X)	0 to 99999999	0.0001 mm	0.00001 in.
S5 (Y)	0 to 99999999	0.0001 mm	0.00001 in.

3. Selection of programming coordinate system

Parameter	Setting range
F85 bit 2	0: Selection of the table coordinate system 1: Selection of the workpiece coordinate system

4. Selection of the operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled)

Parameter	Setting range
F114 bit 1	0: The axis moves according to the tool length offset value. 1: The axis does not move.

4-4 For the MATRIX

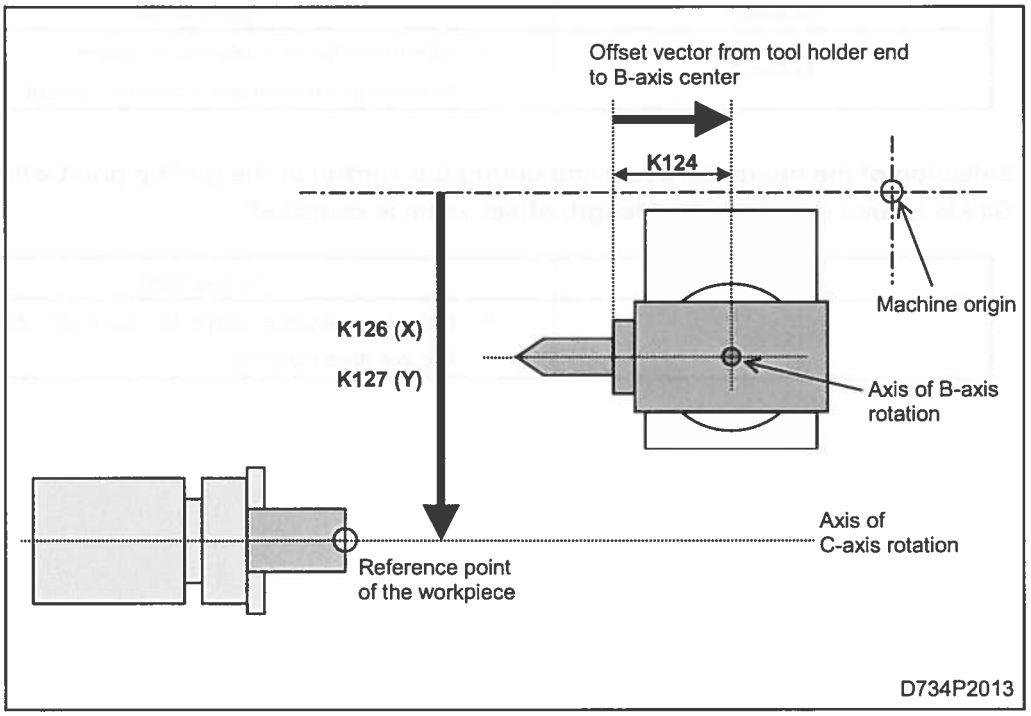


Fig. 4-5 Parameter

1. Limit of feed rate for the mode of tool tip point control

Set the maximum admissible rate of cutting feed for the mode of tool tip point control. The cutting feed during tool tip point control will be limited by parameter **M3** (general limit of feed rate) if it is lower than the setting of parameter **S22**.

Note: Set zero (0) in **S22** to make the parameter invalid. If both parameters **S22** and **M3** are set to zero (0), then parameter **M1** (for rapid traverse) serves as the speed limit in question.

Parameter	Setting range	Axis type	Setting unit	
			Metric	Inch
			Sub-micron for rotational axes	Sub-micron for rotational axes
S22	0 to 200000	Linear	1 mm/min	1 mm/min
		Rotational	1 deg/min	1 deg/min

2. X- and Y-axis coordinates of the axis of C-axis rotation (axis of the turning spindle)

Specify the position of the axis of C-axis rotation in X- and Y-axis machine coordinates.

Parameter	Setting range	Setting unit	
		Metric	Inch
		Sub-micron for rotational axes	Sub-micron for rotational axes
K126	0 to 99999999	0.0001 mm	0.00001 in.
K127	0 to 99999999	0.0001 mm	0.00001 in.

3. Selection of programming coordinate system

Parameter	Setting range
F85 bit 2	0: Selection of the table coordinate system 1: Selection of the workpiece coordinate system

4. Selection of the operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled)

Parameter	Setting range
F114 bit 1	0: The axis moves according to the tool length offset value. 1: The axis does not move.

PROGRAMMING MANUAL

for

Tool Radius Compensation for Five-Axis Machining

MAZATROL FUSION 640M Pro

MAZATROL MATRIX

MANUAL No. : H736PB0031E

Serial No. :

Before using this machine and equipment, fully understand the contents of this manual to ensure proper operation. Should any questions arise, please ask the nearest Technical Center or Technology Center.

IMPORTANT NOTICE

1. **Be sure to observe the safety precautions described in this manual and the contents of the safety plates on the machine and equipment. Failure may cause serious personal injury or material damage. Please replace any missing safety plates as soon as possible.**
2. **No modifications are to be performed that will affect operation safety. If such modifications are required, please contact the nearest Technical Center or Technology Center.**
3. **For the purpose of explaining the operation of the machine and equipment, some illustrations may not include safety features such as covers, doors, etc. Before operation, make sure all such items are in place.**
4. **This manual was considered complete and accurate at the time of publication, however, due to our desire to constantly improve the quality and specification of all our products, it is subject to change or modification. If you have any questions, please contact the nearest Technical Center or Technology Center.**
5. **Always keep this manual near the machinery for immediate use.**
6. **If a new manual is required, please order from the nearest Technical Center or Technology Center with the manual No. or the machine name, serial No. and manual name.**

Issued by *Manual Publication Section, Yamazaki Mazak Corporation*, Japan

06. 2006

CONTENTS

- NOTE -

1 FUNCTION OUTLINE

The function described in this manual refers to three-dimensional tool radius compensation on a five-axis control machine (with the B- and C-axis for rotating the tool axis and the table, respectively) by computing the offset vector in a plane perpendicular to the tool axis.

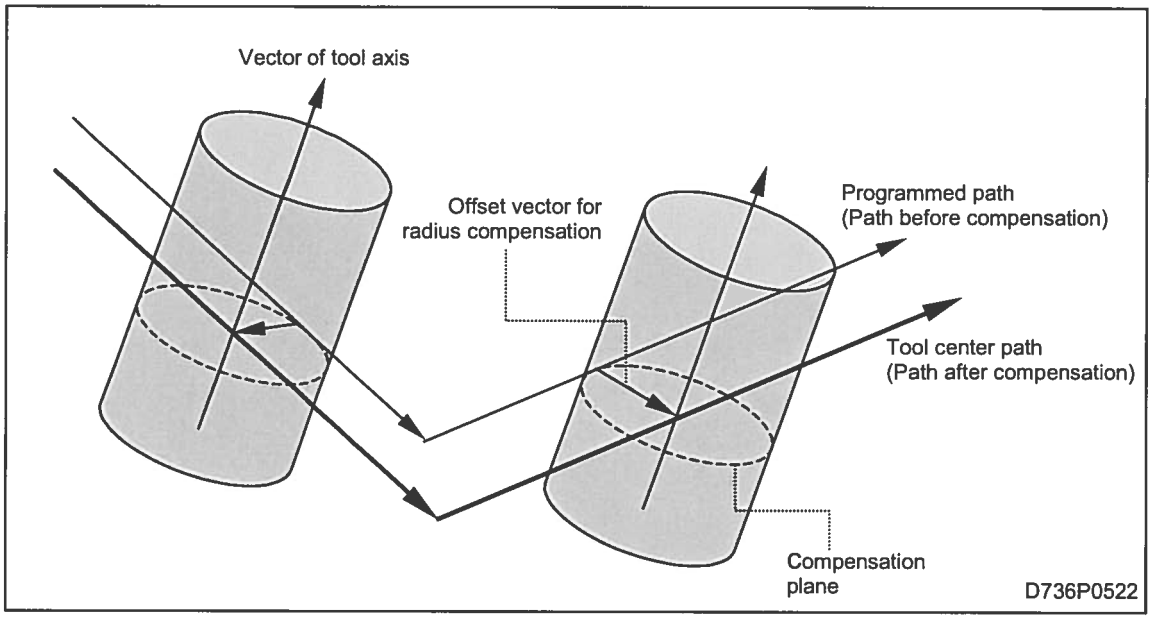


Fig. 1-1 Tool radius compensation for five-axis machining

- NOTE -

2 DETAILED DESCRIPTION

The offsetting function in question conducts tool-path control for tool radius compensation in a plane (called 'compensation plane') perpendicular to the tool axis whose direction is determined by the motion command of the rotational axis concerned. This chapter gives operational particulars proper to this special function, with the compensation plane explained in Section 2-3.

Refer to Section 12-4 of the EIA/ISO Programming Manual for a detailed description of the general tool radius compensation.

2-1 Programming Format

1. Tool radius compensation for five-axis machining ON

```
G41.5 (X_ Y_ Z_ B_ C_ D_);
G42.5 (X_ Y_ Z_ B_ C_ D_);
```

G41.5 : Tool radius compensation (to the left) [group 07]
G42.5 : Tool radius compensation (to the right) [group 07]
XYZBC : Axis motion commands
D : Tool offset data No. for radius compensation

2. Tool radius compensation for five-axis machining OFF (cancellation)

```
G40 (X_ Y_ Z_ B_ C_);
```

G40 : Cancellation of tool radius compensation [group 07]

2-2 Offset Data Items Used for Tool Radius Compensation

The data settings of the **TOOL DATA** display (prepared for the execution of MAZATROL programs) can also be used in tool radius compensation for five-axis machining. The table below indicates those usage patterns of the externally stored tool offset data items which are applied to the tool radius compensation according to the settings of the relevant parameters (**F92** bit 7 and **F94** bit 7).

Parameter		Data in the **TOOL DATA** display		Data in the **TOOL OFFSET** display
F92 bit 7	**F94** bit 7	ACT-ϕ	ACT-ϕ CO./No.	
0	0	×	×	○
0	1	×	○	×
1	0	○	×	○
1	1	○	○	×

○: Used for tool radius compensation.
×: Not used.

2-3 Operation of Tool Radius Compensation for Five-Axis Machining

2-3-1 Startup of the tool radius compensation

The G41.5 or G42.5 code given in the cancellation mode turns on the mode of tool radius compensation for five-axis machining, and describes such an initial offset path to the G41.5 or G42.5 block's ending point as includes compensation in the plane perpendicular to the tool axis in that position. The startup operation in the compensation plane is the same as for the general mode of tool radius compensation.

Take care to give the startup code, G41.5 or G42.5, under the appropriate conditions of G-codes (see the table concerned in Section 3-1); otherwise an alarm will be caused (**962 CAN NOT USE G41.5, G42.5**).

2-3-2 Operation in the mode of tool radius compensation

In the mode of G41.5 or G42.5 the tool radius compensation for five-axis machining only applies to commands of positioning (G00) and linear interpolation (G01). Take care not to use G-codes unavailable in the mode (see the table concerned in Section 3-1); otherwise an alarm will be caused (**961 G41.5, G42.5 MODE IS ACTIVE**).

As for motion blocks automatically interpolated for turning a corner, the direction of the tool axis at the ending point of the first one of the two blocks concerned (as specified in the last B-axis command) is kept intact, along with the rate of feed and other modal information items, up to the stop point for the single-block operation.

2-3-3 Cancellation of the tool radius compensation

The mode of tool radius compensation for five-axis machining is cancelled when one of the following conditions is satisfied:

1. The cancellation command concerned (G40) is executed,
2. Zero is specified as the number of offset data for radius compensation (D00), or
3. The NC is reset.

2-4 Method of Computing the Offset Vector

This section describes how the compensation with respect to the diameter of the tool is performed three-dimensionally by taking account of axis motion commands for rotating the axis of the tool and the workpiece on the table. Let us now take as an example the machining of the side faces of the workpiece by simultaneously rotating the workpiece and the tool axis on the C- and the B-axis.

2-4-1 Conversion into the table coordinates

As the workpiece fixed on the table rotates with the C-axis rotation, the path of compensating the tool radius is to be computed with respect to the table coordinate system, which is fixed on the table and therefore rotates with the C-axis rotation (see Fig. 2-1).

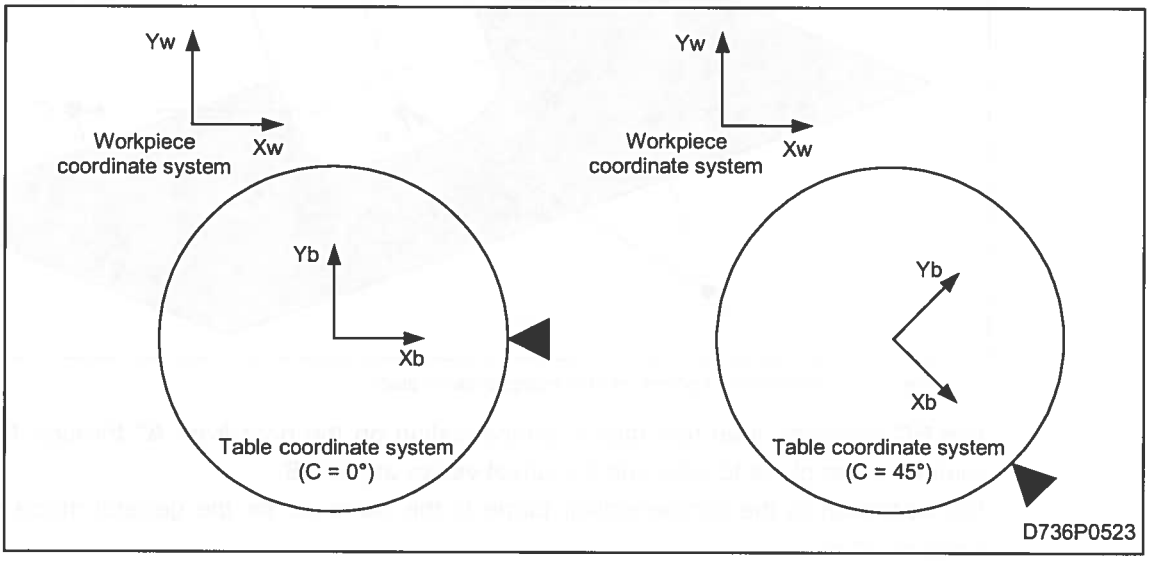


Fig. 2-1 Table coordinate system

Provided that the side faces of the workpiece should be machined by feeding the tool on the path from point [1] through point [2] to point [3], as shown below in Fig. 2-2 (on the left for C = 0°). Since the workpiece simultaneously rotates on the C-axis, however, the real locus of the tool motion in the machine coordinate system draws the path from **A** through **B** to **C**. The NC calculates, for example, the offset vector at point **B** (in the table position of C = 45°) for the path from **A'** through **B** to **C'**, where **A'** and **C'** denote respectively those points in the table coordinate system for C = 45° into which the points **A** and **C** are converted.

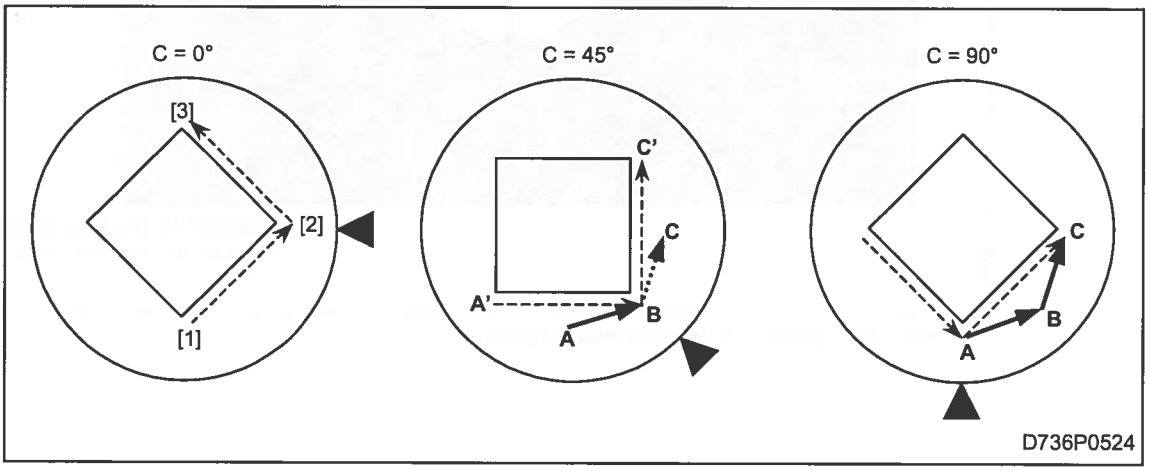


Fig. 2-2 Conversion in the table coordinate system

2-4-2 Conversion of points into the compensation plane

As the tool axis rotates with the B-axis rotation, the path of compensating the tool radius is to be computed within a plane perpendicular to the tool axis. To calculate the offset vector at point **B** (in Fig. 2-3), for example, the NC first converts both the preceding block's ending point **A'** and the next block's ending point **C'** into points **A"** and **C"** by orthogonal projection onto the compensation plane for point **B**.

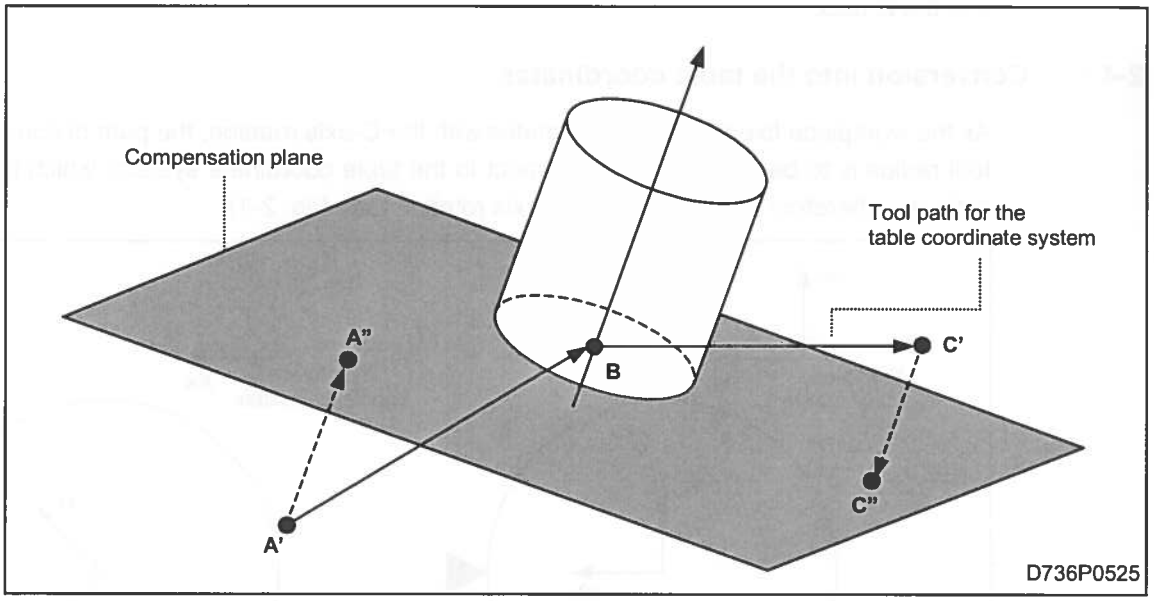


Fig. 2-3 Conversion of points into the compensation plane

The NC conducts then tool radius compensation on the path from **A"** through **B** to **C"** in the compensation plane to calculate the offset vector at point **B**.
The operation in the compensation plane is the same as for the general mode of tool radius compensation.

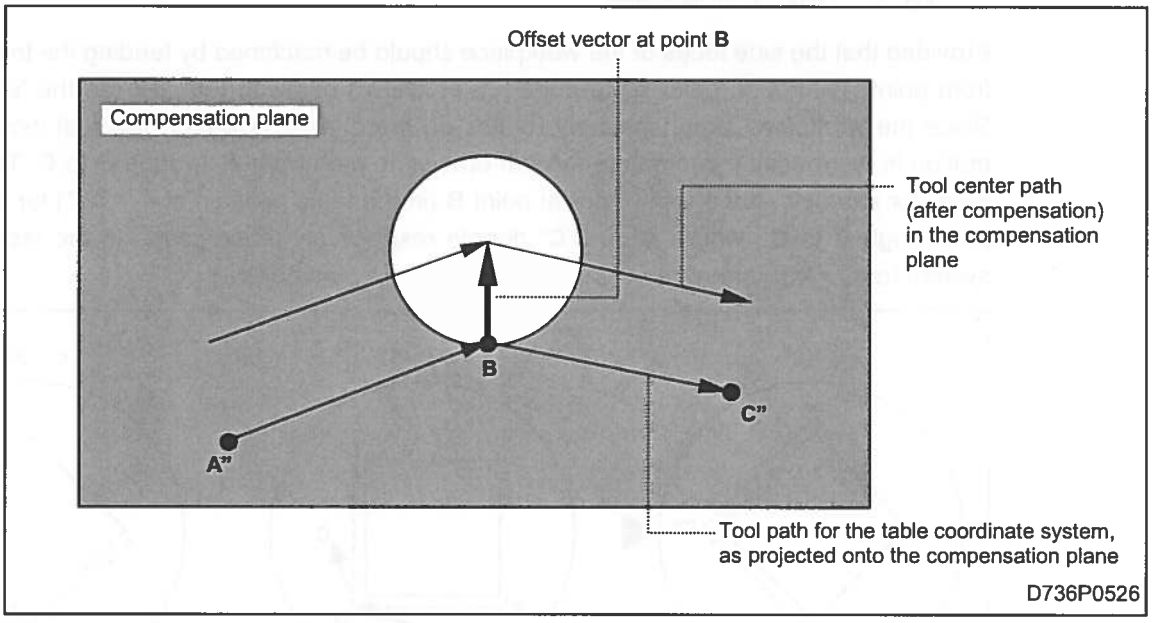


Fig. 2-4 Operation in the compensation plane

3 RELATIONSHIP TO OTHER FUNCTIONS

3-1 Relationship to Other Preparatory Functions

The table below indicates the compatibility of each G-code with the tool radius compensation for five-axis machining in the two columns on the right.

[1] Is the G-code available in the mode of tool radius compensation for five-axis machining?
O: Yes, ×: No [Alarm **961 G41.5, G42.5 MODE IS ACTIVE**].

[2] Is the mode of tool radius compensation for five-axis machining selectable under the condition of the G-code?
O: Yes, ×: No [Alarm **962 CAN NOT USE G41.5, G42.5**].
(– for non-modal codes)

G-code	Group	Function	[1]	[2]
G00	01	Positioning	O	O
G01	01	Linear interpolation	O	O
G01.1	01	Threading with C-axis interpolation	×	×
G02	01	Circular interpolation CW	×	×
G02.1	01	Spiral interpolation CW	×	×
G03	01	Circular interpolation CCW	×	×
G03.1	01	Spiral interpolation CCW	×	×
G04	00	Dwell	O	–
G05	00	High-speed machining mode	×	×
G06.1	01	Spline interpolation	×	×
G06.2	01	NURBS interpolation	×	×
G07	00	Virtual-axis interpolation	×	–
G07.1	00	Cylindrical interpolaiton	×	×
G09	00	Exact-stop check	O	–
G10	00	Programmed parameter input ON	O	–
G10.9	00	Selection between diameter/radius data input	×	–
G11	00	Programmed parameter input OFF	O	–
G12.1	26	Polar coordinate interpolation ON	×	×
G13.1	26	Polar coordinate interpolation OFF	×	O
G17	02	Plane selection X-Y	O	O
G18	02	Plane selection Z-X	O	O
G19	02	Plane selection Y-Z	O	O
G20	06	Inch command	×	O
G21	06	Metric command	×	O
G22	04	Pre-move stroke check ON	O	O
G23	04	Pre-move stroke check OFF	O	O
G27	00	Reference point check	×	–
G28	00	Return to reference point	×	–
G29	00	Return to starting point	×	–
G30	00	Return to reference point No. 2 to No. 4	×	–
G31	00	Skip	×	–
G31.1	00	Multi-step skip 1	×	–
G31.2	00	Multi-step skip 2	×	–
G31.3	00	Multi-step skip 3	×	–
G32	01	Threading	×	×

G-code	Group	Function	[1]	[2]
G33	01	Threading	×	×
G34	01	Variable-lead threading	×	×
G37	00	Automatic tool length measurement	×	–
G38	00	Vector setting for tool radius compensation	×	–
G39	00	Interpolation of corner arc for tool radius compensation	×	–
G40	07	Nose R/Tool radius compensation OFF	O	O
G41	07	Nose R/Tool radius compensation to the left	×	×
G41.5	07	Tool radius compensation (for five-axis machining) to the left	O	O
G42	07	Nose R/Tool radius compensation to the right	×	×
G42.5	07	Tool radius compensation (for five-axis machining) to the right	O	O
G43	08	Tool length offset (+)	O	O
G43.4	08	Tool tip point control, type I	×	O
G43.5	08	Tool tip point control, type II	×	O
G44	08	Tool length offset (–)	O	O
G45	00	Tool position offset, extension	×	–
G46	00	Tool position offset, reduction	×	–
G47	00	Tool position offset, double extension	×	–
G48	00	Tool position offset, double reduction	×	–
G49	08	Tool length offset OFF	O	O
G50	11	Scaling OFF	O	O
G50.1	19	Mirror image by G-code OFF	×	O
G50.2	23	Polygonal machining OFF	×	O
G51	11	Scaling ON	O	O
G51.1	19	Mirror image by G-code ON	×	×
G51.2	23	Polygonal machining ON	×	×
G52	00	Local coordinate system setting	×	–
G53	00	Selection of machine coordinate system	×	–
G54	12	Selection of workpiece coordinate system 1	×	O
G54.1	12	Selection of additional workpiece coordinate systems	×	O
G55	12	Selection of workpiece coordinate system 2	×	O
G56	12	Selection of workpiece coordinate system 3	×	O
G57	12	Selection of workpiece coordinate system 4	×	O
G58	12	Selection of workpiece coordinate system 5	×	O
G59	12	Selection of workpiece coordinate system 6	×	O
G60	00	One-way positioning	×	–
G61	13	Exact-stop check mode	O	O
G61.1	13	Shape correction mode	O	O
G62	13	Automatic corner override	×	×
G63	13	Tapping mode	×	×
G64	13	Cutting mode	O	O
G65	00	User macro simple call	×	–
G66	14	User macro modal call A	×	×
G66.1	14	User macro modal call B	×	×
G67	14	User macro modal call OFF	×	O
G68	16	Programmed coordinates rotation ON	×	×
G69	16	Programmed coordinates rotation OFF	×	O
G71.1	09	Fixed cycle (chamfering cutter 1)	×	×
G72.1	09	Fixed cycle (chamfering cutter 2)	×	×

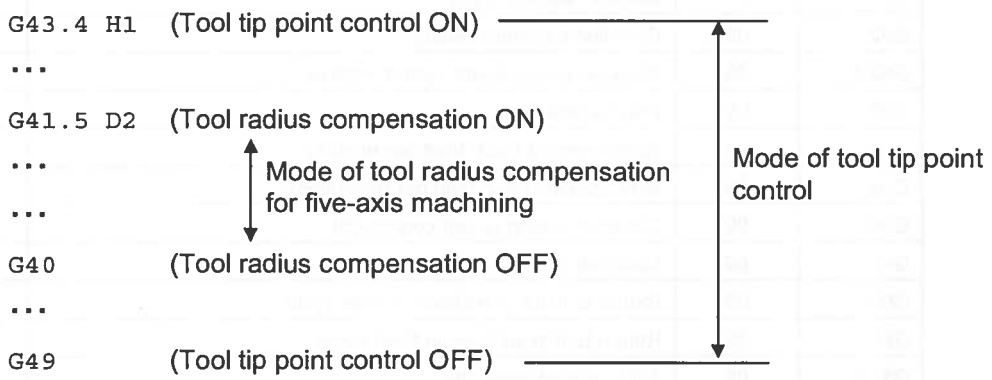
G-code	Group	Function	[1]	[2]
G73	09	Fixed cycle (high-speed deep-hole drilling)	×	×
G74	09	Fixed cycle (reverse tapping)	×	×
G75	09	Fixed cycle (boring)	×	×
G76	09	Fixed cycle (boring)	×	×
G77	09	Fixed cycle (back spot facing)	×	×
G78	09	Fixed cycle (boring)	×	×
G79	09	Fixed cycle (boring)	×	×
G80	09	Fixed cycle OFF	×	O
G81	09	Fixed cycle (spot drilling)	×	×
G82	09	Fixed cycle (counter boring)	×	×
G83	09	Fixed cycle (deep-hole drilling)	×	×
G84	09	Fixed cycle (tapping)	×	×
G84.2	09	Fixed cycle (synchr. tapping)	×	×
G84.3	09	Fixed cycle (synchr. reverse tapping)	×	×
G85	09	Fixed cycle (reaming)	×	×
G86	09	Fixed cycle (boring)	×	×
G87	09	Fixed cycle (back boring)	×	×
G88	09	Fixed cycle (boring)	×	×
G89	09	Fixed cycle (boring)	×	×
G90	09	Absolute data input	O	O
G91	09	Incremental data input	O	O
G92	09	Coordinate system setting	×	–
G92.5	09	Workpiece coordinate system rotation	×	–
G93	09	Inverse time feed	O	O
G94	09	Asynchronous feed (feed per minute)	O	O
G95	09	Synchronous feed (feed per revolution)	O	O
G96	09	Constant cutting speed control ON	×	O
G97	09	Constant cutting speed control OFF	×	O
G98	09	Return to initial point level in fixed cycle	×	O
G99	09	Return to R-point level in fixed cycle	×	O
G110	09	Axis name change ON	×	×
G111	09	Axis name change OFF	×	O
G113	09	Hob milling OFF	×	O
G114.3	09	Hob milling ON	×	×
G270	09	Finishing cycle	×	×
G271	09	Longitudinal roughing cycle	×	×
G272	09	Transverse roughing cycle	×	×
G273	09	Copying cycle	×	×
G274	09	Transverse cut-off cycle	×	×
G275	09	Longitudinal cut-off cycle	×	×
G276	09	Compound threading cycle	×	×
G290	09	Fixed cycle A (Outside/Inside turning)	×	×
G292	09	Threading cycle	×	×
G294	09	Fixed cycle B (Transverse turning)	×	×

3-2 Restrictions

- The calculated path of tool radius compensation cannot be checked for interference, irrespective of the setting of the parameter concerned (**F92** bit 5: Checking to avoid interference ON/OFF).
- The radius compensation codes G38 (to set an offset vector) and G39 (to interpolate a circular arc at a corner) are not available in the mode of G41.5 or G42.5.
- Corner chamfering or rounding commands are not available in the mode of G41.5 or G42.5.
- The tool change command, if required, must always be given after cancelling the mode of G41.5 or G42.5.
- Manual interruption in general, MDI interruption, and interruption by the manual pulse handle in particular, cannot be used in the mode of G41.5 or G42.5.
- Tool radius compensation for five-axis machining is not available if the C-axis control of the turning spindle No. 2 is concerned (on accordingly executed machines).
- The function in question cannot be used at all in the mode of operation for turning.
- Take the following precautions for compound use with the tool tip point control:

1. The tool radius compensation for five-axis machining must be turned on and off within the mode of tool tip point control.

<Programming example>



2. The tool tip point control must have the workpiece coordinate system selected (with **F85** bit 2 = 1) for describing the tool path in the program. Otherwise (i.e. when the parameter **F85** bit 2 is set to 0 to use the table coordinate system in programming) an alarm will be caused (**962 CAN NOT USE G41.5, G42.5**) by the selection of the mode of tool radius compensation for five-axis machining even with the above condition being satisfied.

4 RELATED ALARMS

Alarm No.	Alarm message	Description
936	OPTION NOT FOUND	The system of machining is not equipped with the optional function of tool radius compensation for five-axis machining.
961	G41.5, G42.5 MODE IS ACTIVE	The command given is not compatible with the mode of tool radius compensation for five-axis machining.
962	CAN NOT USE G41.5, G42.5	Tool radius compensation for five-axis machining is not selectable under the current modal conditions.

- NOTE -

Mazak

YAMAZAKI MAZAK CORPORATION
1-131 Takeda, Oguchi-cho, Niwa-gun,
Aichi-pref., Japan
Phone: 0587-95-1131
Facsimile: 0587-95-2717

MAZAK CORPORATION
8025 Production Drive,
Florence, KY41042
U.S.A
Phone: 859-342-1700
Facsimile: 859-342-1865

YAMAZAKI MAZAK EUROPE, N.V.
Research Park, Grauwmeer 7
3001 Leuven,
BELGIUM
Phone: 16-39-1611
Facsimile: 16-40-0196

YAMAZAKI MAZAK DEUTSHLAND GmbH
Esslinger Strasse 4-6
D-73037 Goeppingen
GERMANY
Phone: 7161-6750
Facsimile: 7161-675274

YAMAZAKI MAZAK U.K. LTD.
Badgeworth Drive Worcester
WR49NF
UNITED KINGDOM
Phone: 1905-755755
Facsimile: 1905-755001

YAMAZAKI MAZAK SINGAPORE PTE., LTD.
21, Joo Koon Clrcle,
Jurong, Singapore 629053
Phone: 6862-1131
Facsimile: 6861-9284

URL: www.mazak.com

Printed in Japan