HNC-8 Five-axis CNC System User Manual

V2.4 Version

Wuhan Huazhong Numerical Control Co., Ltd

Introduction

The manual may help you to quickly get familiar with the HNC-8 system, providing detailed information about commissioning, programming or application methods. Any updates or modification of the manual is not allowed without the written permission of Wuhan Huazhong Numerical Control Co., LTD (hereafter referred to as "HCNC"). Without HCNC's authorization or written permission, any units or individuals are not allowed to modify or correct the manual. HCNC will not be responsible for any losses thus incurred to customers.

In this manual we have tried as much as possible to describe all the various matters concerning of the system. However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities. Therefore, matters which are not especially described as possible in this manual should be regarded as “impossible” or “not allowed”.

Please favor me your instruction for shortages and inadequacies of the manual.

Note

- As to notes such as "Limitations" and "Usable functions", the specification provided by the machine tool manufacturer is superior to the manual. Please conduct dryrun before actual machining and confirm machining program, tool compensation volume and workpiece offset, and so on.
- Please explain matters which are not described in the manual as "Infeasible".
- The manual is prepared on the condition that all functions are configured. Please make a confirmation according to the specification provided by the machine tool manufacturer in use.
- For relevant instructions for machine tools, please refer to the specification provided by the machine tool manufacturer.
- Usable screens and functions differ with different NC systems (or versions). Please be sure to confirm specifications before use.

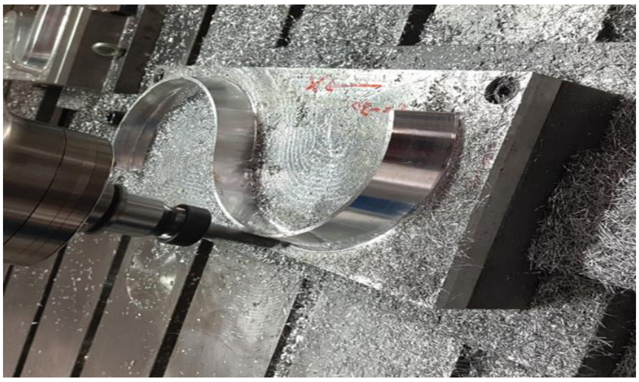
Content

1. Machining Feature of 5-Axis Machine Tool

In conventional 3-axis CNC machine tool, direction of tool axis remains unchanged during machining, and the machine tool perform interpolation motion only on 3 linear axes. For the machining of parts as shown in figure 1-1, the shorting coming of 3-axis machine tool is particularly highlighted. Compared with the 3-axis machine tool, 5-simulatenouly-axis machine tool has additional two rotational degrees of freedom, and the tool movement is more flexible, which is beneficial for the tool to maintain the best cutting state and effectively avoid machining interference. Therefore, five-simultaneous-axis CNC machining has significant advantages when processing complex free-form surfaces.

(a) Spider

(b) Impeller



(C) S-shaped part

Figure 1-1 Typical 5-axis machining parts

Compared with three-axis machining, five-axis machining has the following advantages:

(1) Reduce number of clamping and improve processing efficiency

One of the main advantages of five-axis machining is that only one clamping can complete the machining of complex-shaped parts, such as inclined hole machining, curved surface machining, etc.

Since there is no need for multiple clampings, the simultaneous-five-axis processing technology not only shortens the processing cycle, but also avoids manual or mechanical errors caused by multiple clampings, greatly improving the processing accuracy.

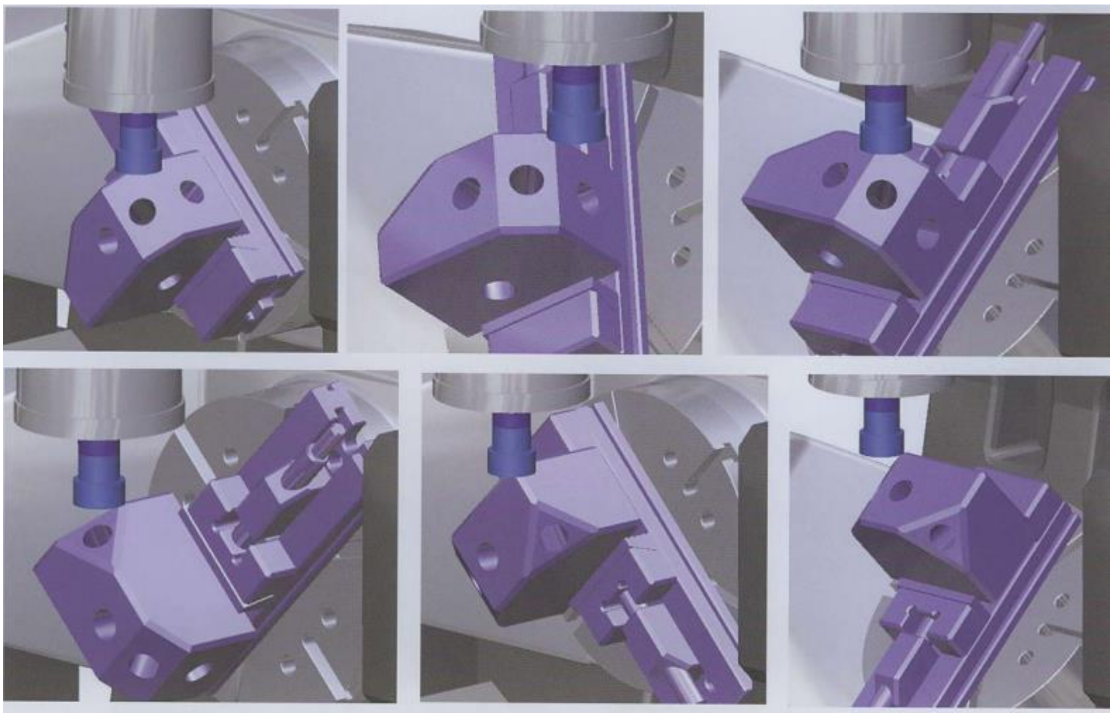


Figure 1-2 Multi-surface with one clamping

(2) Keep best cutting posture

Due to the five axes' degrees of freedom, according to the normal vector of the curved surface, the rotation of rotary axis always maintains the best cutting posture and improves the cutting efficiency.

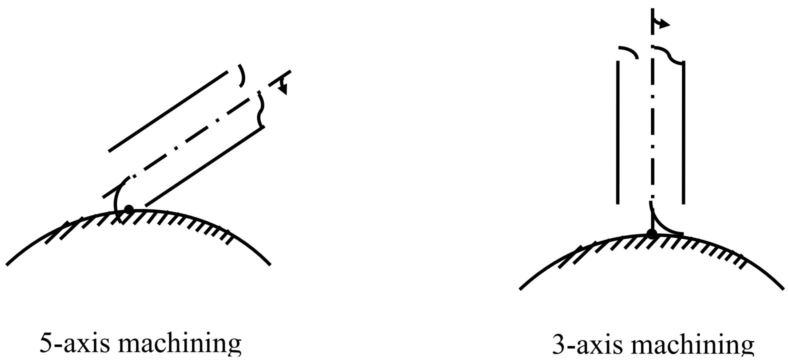


Figure 1-3 Cutting posture comparison

(3) Effectively avoid processing interference

For complex curved parts such as impellers and blades, tool interference will be caused in some machining areas due to the defects of the three-axis machine tool itself, which cannot meet the machining requirements. The five-axis machine tool solves the problem of machining interference by changing the cutting direction.

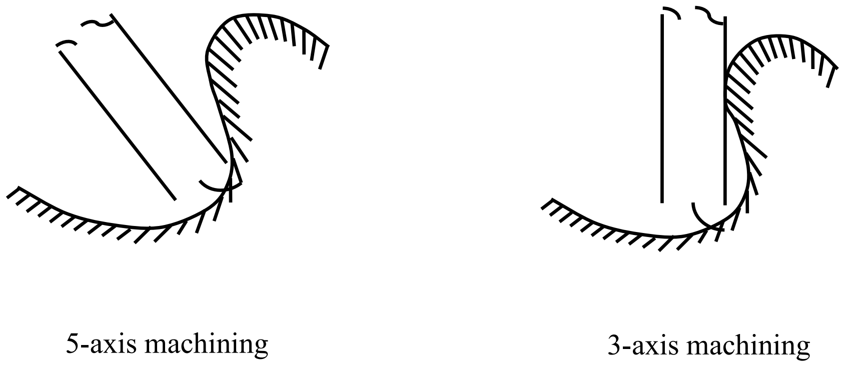


Figure 1-4 Machining interference

(4) Side milling improves machining efficiency and quality

In aerospace, there is a demand for surface sidewall contour processing. Inclining the tool to a certain angle and performing the milling with the side edge of the tool can shorten the processing time and improve the processing quality.

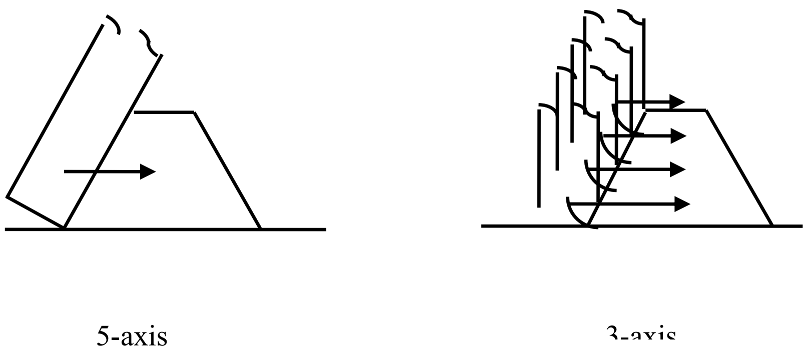


Figure 1-5 Processing efficiency

(5) Expand scope of processing

For machining of some curved surfaces such as the integral impeller, tool axis has to be rotated due to the distortion and the mutual positional restrictions between the curved surfaces. At this point, only five-simultaneous-axis CNC machine tools can be used, otherwise it is difficult or even impossible to meet the processing requirements. In addition, in mold processing, sometimes only five-simultaneous-axis CNC machine tools can avoid interference between tool and workpiece.

In short, five-axis machining plays an important role in the processing of complex curved parts. With one clamping, it completes all procedures, adjusts the tool to the best cutting posture, reasonably avoids interference positions, so as to obtain better processing quality and reduce costs.

True five-axis or false five-axis

True five-axis machine has RTCP function. It can automatically perform conversion based on swivel length of spindle and the machine coordinates of rotary table. During programming, only the coordinates of the workpiece need to be considered, not the kinematic chain structure of the five-axis machine tool. Whether it is a true five-axis motion, it does not depend on whether it can perform simultaneous 5-axis motion, the false five-axis machine can also perform simultaneous five-axis motion. The main key is that the system has the RTCP true five-axis algorithm. After applying the RTCP mode, programming 5-coordinate machining can directly plan the trajectory of the tool nose in the workpiece coordinate system, so the programming will become much simpler and more efficient. CNC systems without RTCP must rely on CAM programming and post-processing. The tool path is planned in advance, and the center positions of the fourth and fifth axes of the machine tool are input into the post-processing module. The generated CNC program is the machine coordinate point. The actual tool nose path is closer to the programmed path, and the accuracy cannot be guaranteed. For the same part, if the machine tool or the tool is changed, the program must be generated by post-processing, which is inconvenient in actual use.

2. 5-Axis CNC Machine Tool Structure

2.1. 5-Axis Machine Type

5-axis machine tool has additional two rotary axes on the basis of three of linear axes, and can control 5 axes simultaneously. There are two types of 5-axis machine tool: positioning 5-axis and simultaneous 5-axis. The positioning 5-axis machine is the 3+2 machine, and its tool axis vector can change, but the process on whole cutting path after the tool axis vector is fixed. However, for simultaneous 5-axis machine tool, tool vector can change on whole cutting path as required. According to the arrangement and combination of Cartesian coordinates X, Y, Z, A, B, C, and whether the rotary axis acts on the tool or the worktable, the five-axis machine tool structure can have hundreds of combinations, and there are as many as 21 typical five-axis machine tools on the market. For the structure of two rotary axes, five-axis machine tools can be divided into three categories:

1. Dual swivel head structure

For the simultaneous 5-axis machine tool with such type of structure, the worktable doesn't move, and two rotary axes are on spindles. Since the swivel axis is attached with a spindle, the size of the dual swivel head itself is generally larger. This type of machine generally adopts a gantry or moving-beam gantry structure, and processes larger part, such as aerospace rib parts.

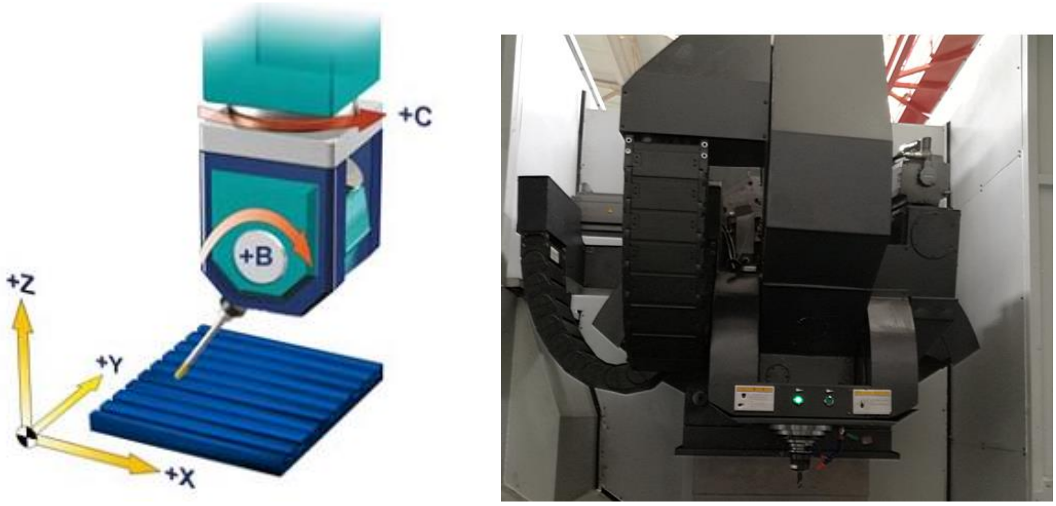


Figure 2-1 Dual swivel head machine tool

2. Dual rotary table structure

For the simultaneous 5-axis machine tool with such type of structure, tool axis direction is fixed, and two rotary axes are on worktables. During machining, workpiece rotates with worktable, and the clamping load needs to be considered. This type of machine tool can process relatively small parts, and is suitable for impellers, small precision molds, and the like.

Figure 2-2 Dual rotary table machine tool

3. Mixed structure

For the simultaneous 5-axis machine tool with such type of structure, two rotary axes are on spindle and worktable relatively. Single worktable rotating makes the workpiece size can be machine range large, and the swing axis allows the direction of the tool axis to be changed flexibly, so this type of five-axis machine tool can have a variety of different combinations to meet the processing requirements of different parts. Suitable for blade processing, etc.

Figure 2-3 Mixed structure machine tool

The system can support the structure of up to three linear axes plus four rotary axes, and support any rotary axis direction such as 45-degree tilting head.

2.2. Manual Calibration of 5-Axis Machine

The five-axis CAM process program created in advance cannot be imported into the system for processing. Before using the five-axis function, users need to calibrate the geometric dimensions of the five-axis kinematic chain and enter it into the system to achieve five-axis machining. Whether it is of the swing head structure or the rotary table structure, the key of five-axis structure parameter calibration is to determine the center and direction of rotary axis. According to the kinematics principle, the machine tool axis movement has a transmission chain relationship. In two rotary axes, the direction of an axis remains unchanged during the movement and becomes a master axis, while the direction of the other axis changes following the fixed axis and becomes a slave axis. There are a master axis and slave axis working on the tool, similarly the worktable also has a master axis and a slave axis. In traditional manual measurement, users use test arbor, dial indicator, and square gauge to measure position and direction of master and slave axes, thereby completing the five-axis calibration.

When machine tool only has a swivel head or a rotary table, only slave axis but not master axis works on tool or rotary table. Therefore, users only need to calibrate position and direction of slave axis.

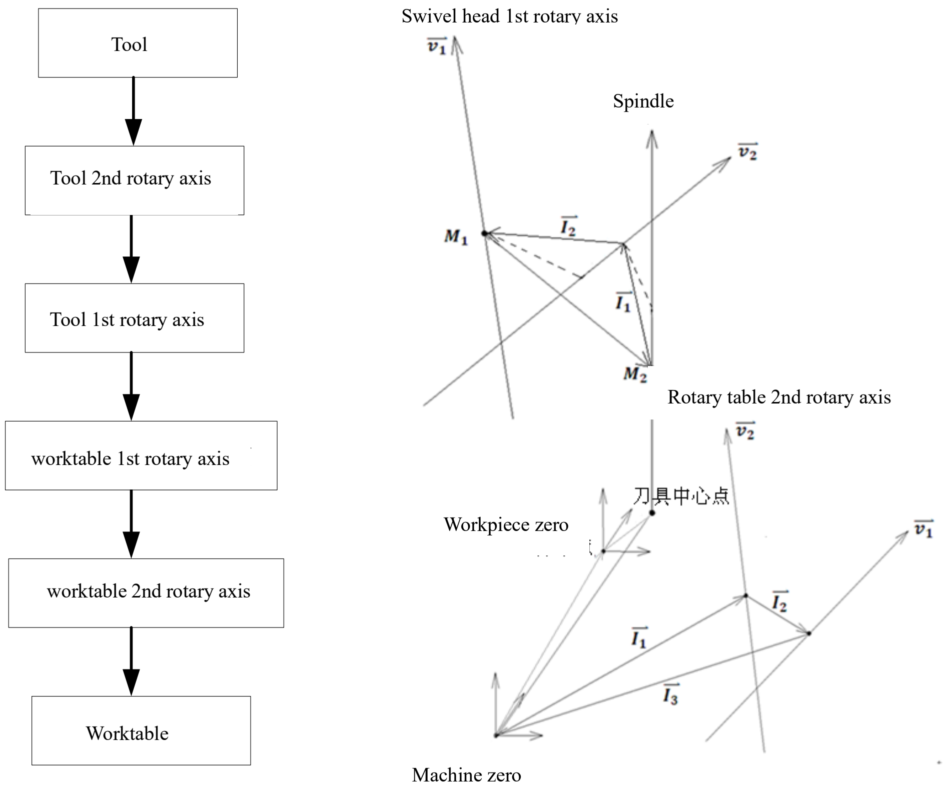


Figure 2-4 5-axis kinematic chain

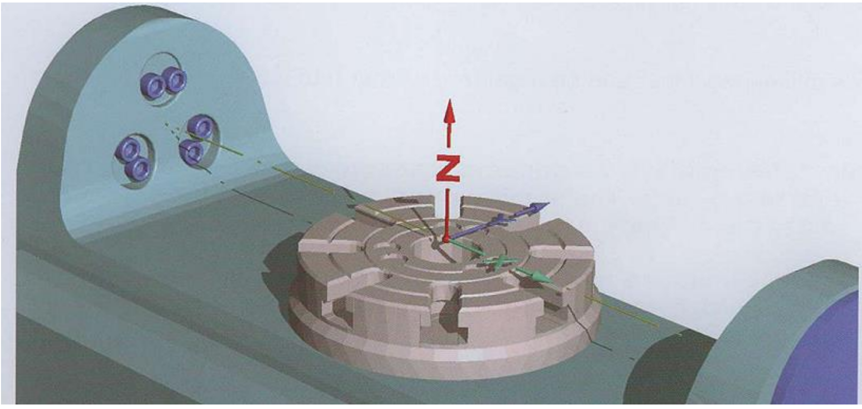


Figure 2-5 Rotation center and direction

2.3. Automatic Calibration of 5-Axis Machine

To solve the defects of manual measurement accuracy and use limitations, maximize the automation of measurement, and realize the precise control of the center point of the rotating tool, the system provides the automatic calibration function of the key geometric dimensions of the five-axis machine tool. In the calibration process, a trigger probe and a datum sphere are used. Through the macro program of measurement, the collected data points are fitted, and the five-axis machine tool structure parameters are automatically generated to improve the measurement accuracy and measurement efficiency. Automatic calibration is suitable for testing the three models A, B and C of ISO/CD 10791-6 standard. It is easy to use and does not require professional operators to complete RTCP parameter calibration.

The measuring tool: Renishaw strain probe and magnetic base of datum sphere, as shown in Figure 8.

Figure 2-6 Probe and datum sphere

Figure 2-7 shows the flow chart of automatic measurement, including instrument installation, parameter setting, collision collection, and RTCP parameter calculation steps. The probe is controlled via macro program to collide with the datum sphere and the machine coordinates of the collision point is latched. The least squares data processing is used to fit the directions of master axis and slave axis and spatial positions for coordinates of center of each datum sphere, and then RTCP parameters are got.

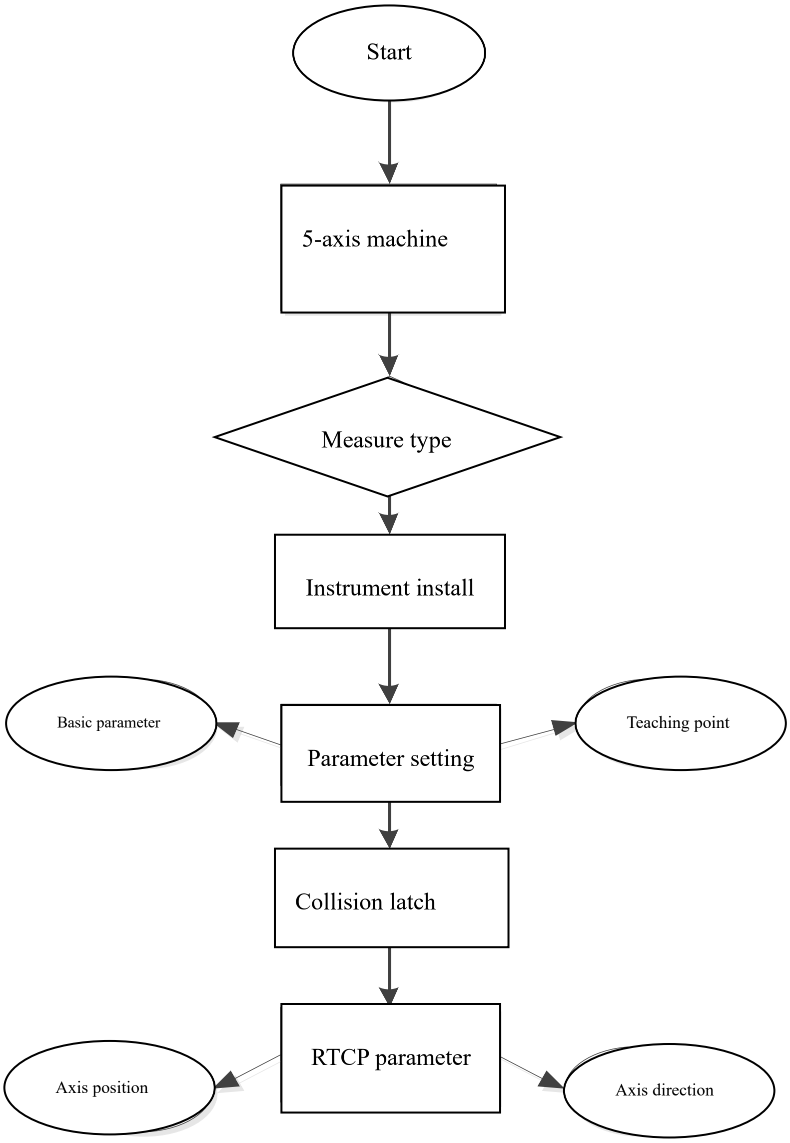


Figure 2-7 Flow chart of automatic calibration

1) Instrument installation. Install trigger probe and datum sphere based on machine tool structure type, and use the lever indicator to perform the spindle concentric calibration of the probe.

2) Parameter setting. Set measurement parameters on the automatic measurement software interface of CNC system including 10 basic parameters: measurement type, display sequence of rotary axis, rotary axis name, safe height, positioning speed, intermediate speed, trigger speed, datum sphere radius, tool length, and tool radius, and 8 teaching points of master axis and 8 teaching points of slave axis. The teaching point is a reference point used to determine the relative positions of datum sphere and probe for collision.

3) Collision collection. Based on the 10 basic parameters and the coordinates of 8 master axis teaching points and 8 slave axis teaching points determined in step 2, control the probe with macro program to collide with the datum sphere and latch coordinates X, Y, Z in the machine coordinate system during collision. Each teaching point collides 4 times; the collision process is as follows:

The probe collides with the vertex of the datum sphere in the negative direction of Z-axis, and the machine coordinate point 1 is latched; the probe collides with equator of datum sphere in the positive direction of X axis, and the machine coordinate point 2 is latched; the probe collides with equator of datum sphere in the negative direction of X axis, and the machine coordinate point 3 is latched; the probe collides with equator of datum sphere in the positive or negative direction of Y axis, and the machine coordinate point 4 is latched;

Or, the probe collides with vertex of the datum sphere in the negative direction of Z-axis, and the machine coordinate point 1 is latched; the probe collides with equator of datum sphere in the positive direction of Y axis, and the machine coordinate point 2 is latched; the probe collides with equator of datum sphere in the negative direction of Y axis, and the machine coordinate point 3 is latched; the probe collides with equator of datum sphere in the positive or negative direction of X axis, and the machine coordinate point 4 is latched;

(4) RTCP parameter calculation. Calculate the standard sphere center coordinates corresponding to each teaching point based on the latched collision point coordinates; fit the directions of master axis and slave axis and spatial positions for coordinates of center of each datum sphere to get RTCP parameters including axis direction vector of the master axis, axis direction vector of slave axis, axis offset vector of master axis, and axis offset vector of slave axis.

3. 5-Axis Command Function

3.1. Overview of 5-Axis Machining

In conventional 3-axis CNC machine tool, direction of tool axis remains unchanged during machining, and the machine tool perform interpolation motion only on 3 linear axes. For the machining of parts as shown in figure 1-1, the shorting coming of 3-axis machine tool is particularly highlighted. Compared with the 3-axis machine tool, 5-simulatenouly-axis machine tool has additional two rotational degrees of freedom, and the tool movement is more flexible, which is beneficial for the tool to maintain the best cutting state and effectively avoid machining interference. Therefore, five-simultaneous-axis CNC machining has significant advantages when processing complex free-form surfaces.

G codes of HNC-8 series 5-axis machining are shown as below:

G code	Group No.	Function
G43.4	10	RTCP rotation angle programming
G43.5		RTCP tool vector programming
[G49]		Cancel RTCP function
G53.1	00	Tool axis direction control (only for rotary axis movement)
G53.2		Tool axis direction control (tool nose moves with tool axis direction)
G53.3		Normal feed & retract
NURBSB		Hyperbolic interpolation
G68.1	05	Train coordinate system establishment with 3-point
G68.2		Train coordinate system establishment with Euler angle
[G69]		Cancel trait coordinate system
G140	25	Linear interpolation
G141		Great circle interpolation

3.2. Tool Axis Direction (G43.4 to G43.5)

According to machining mode, 5-axis machining function is divided into simultaneous 5-axis machining and 5-axis orientation machining (or inclined surface machining).

In simultaneous 5-axis machining, the tool may produce continuous or intermittent 5-coordinate motion relative to the workpiece at any time, and the tool axis vector direction is also constantly changing; for 5-axis orientation machining (or inclined surface machining), different from conventional three-axis programming, it is necessary to define a unique tool axis vector direction

Therefore, compared with ordinary three-axis programming, five-axis programming mainly focuses on the direction of tool axis. Find below table for the change of tool axis direction.

G code	Specifying mode
G43.4 H_	Position specifying mode (it also includes the tool length compensation function, which is introduced in the next section
G43.5 H_	Vector specifying mode（It also includes the tool length compensation function, which is introduced in the next section）
G49	Cancel tool axis specifying mode

3.2.1. Position Specifying Mode G43.4

Command format

G43.4 H_ （X_Y_Z_）

G49

Parameter	**Meaning**
X, Y, Z	Movement on X, Y, Z
H	Number of tool compensation amount in tool compensation

Description

1) The position specifying mode is also referred to as rotation angle programming. RTCP function is enabled with G43.4, and its subsequent command position is the position of the rotary axis (A, B, C) and the position of the linear axis (X, Y, Z). When the RTCP function is activated, these specified points are the values calculated based on the RTCP function.

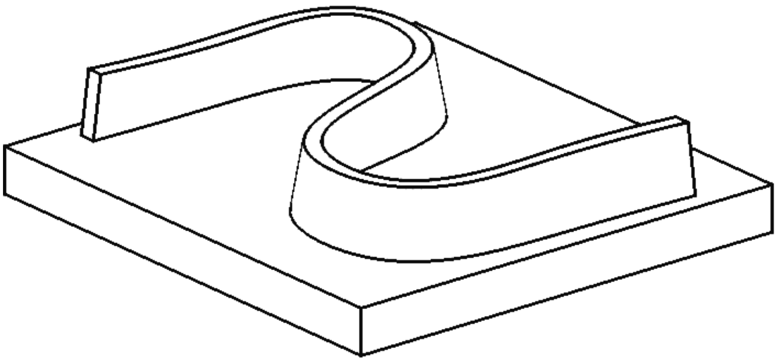
2) G43.4 H_ can be on the same line as X, Y, Z, A, C, but it is recommended that G43.4 H_ be specified on a separate line.

Application example

Example of NC program block format

G43.4 H01	Enable RTCP function, position specifying mode, Call length composition of tool 1
G01 X6.994 Y-22.71 Z116.425 A-2.984 C270.	Specify positions of linear axis and rotary axis (simultaneous 5-axis)
X9.642 Y11.694 Z37.863 A-10.326 C269.994	Specify positions of linear axis and rotary axis (simultaneous 5-axis)
……	
G49	Cancel RTCP function, cancel tool length composition

Example, program for S-shaped parts shown as below.



```
%123
G54
G43.4 H01
G01 X6.019 Y-22.492 Z116.476 A0.0 C0.0 F3000. S5000 M03
X6.019 Y-22.492 Z116.476 A-2.984 C270.
X2.031 Y-21.714 Z36.622 A-3.174 C270. F1500.
X2.579 Y-19.274 Z36.738 A-3.767 C270.
X2.937 Y-17.667 Z36.814 A-4.154 C270.
X3.464 Y-15.28 Z36.926 A-4.722 C270.
```

X3.993 Y-12.853 Z37.038 A-5.292 C270.

………………….

X8.293 Y9.452 Z37.959 A-9.922 C269.997

X8.669 Y11.837 Z38.04 A-10.326 C269.994

X9.254 Y15.816 Z38.167 A-10.955 C269.983

………………….

G49

G91 G28 Z0

M30

3.2.2. Vector specifying mode G43.5

Command format

G43.5 H_ （X_Y_Z_）

G49

Parameter	**Meaning**
X, Y, Z	Movement on X, Y, Z
H	Number of tool compensation amount in tool compensation

Description

(1) RTCP function is enabled with G43.5. The subsequent command is to replace the position of rotary axis, and specify the direction (I, J, K) of tool axis at the end of each block in the workpiece coordinate system; the position of rotary axis calculated by CNC based on the value calculated by RTCP function makes the tool move in the specified direction.

(2) G43.5 H_ can be on the same line as X, Y, Z, A, C, but it is recommended that G43.5 H_ be specified on a separate line.

Example of NC program block format

G43.5 H01	Enable RTCP function, tool vector programming, Call tool length composition of tool 1

G01 X-27.748 Y-23.632 Z15.435 I-.324 J.324 K.889	Specify positions of linear axis and direction of tool axis (simultaneous 5-axis)
Y-4.983 Z11.296 I-.341 J.068 K.938	Specify positions of linear axis and direction of tool axis (simultaneous 5-axis)
……	
G49	Cancel RTCP function

Note

(1) Both G43.4 and G54.5 are to enabled RTCP function and tool length compensation;

(2) G49 is to cancel RTCP function and tool length compensation. When there are G43.4, G43.5 commands and inclined surface machining commands, they must correspond to each other, otherwise if the coordinate value executed is incorrect, an "overtravel" alarm error will occur.

(3) For simultaneous 5-axis machining, the direction of the tool axis changes continuously in fact.

(4) When executing G43.4 or G43.5 program, if "feed hold" is selected or program is suddenly interrupted in RTCP state, users should press RESET or EMERGNECY STOP first, otherwise it is easy for the tool to interfere with the workpiece or the machine tool during manual operation

(5) When X, Y, Z, A, C, and G43.4 H_ or H43.5 H_ are executed on the same line, if the setting is incorrect, an "axis overtravel" alarm will be issued.

3.3. Length Compensation on tool axis（G43.4, G43.5/G49）

In machining of rotary axis (multi-axis), because RTCP function (tool nose following function) is usually used, the system needs to associate the control point with the tool center, so that the interpolation path is the path required. In addition, different five-axis machine tools have different tool length measurement methods, so in multi-axis machining, the tool length compensation needs to be specified in the direction of tool axis.

G43.4 and G43.5 can be used to specify both tool axis direction and tool length compensation.

Through real-time tool length compensation, the system positions the control point on the tool center and turns on the RTCP function to ensure that the tool center point moves along the specified path

Command format

G43.4 X_ Y_ Z_H_ ; Tool length compensation in tool axis direction; position specifying mode

G43.5 X_ Y_ Z_H_ ; Tool length compensation in tool axis direction; vector specifying mode

G49 ; Cancel tool length compensation

Parameter	Meaning
X, Y, Z	Movement on X, Y, Z
H	Number of tool compensation amount in tool compensation

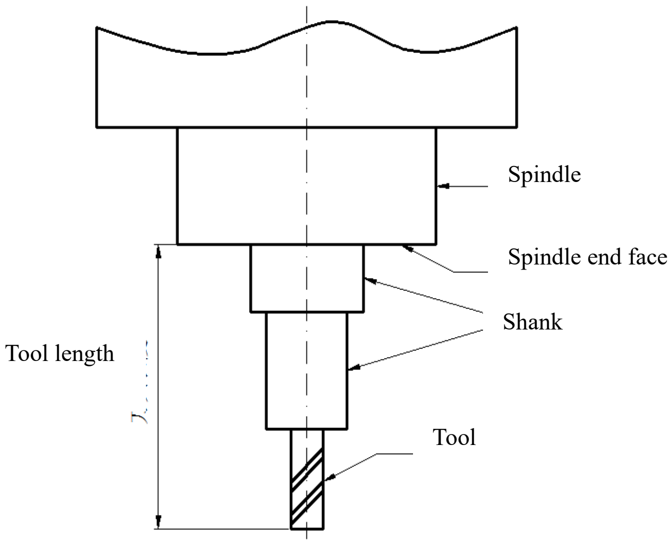
Description

(1) G43, G44 and G43.4, and G43.5 are the same group of G codes. Therefore, these compensations cannot be performed at the same time. And G49 can be used to cancel these G commands.

(2) G43.4 H_ and G43.5 H_ are generally specified on a separate line, and also can be on the same line with X, Y, Z. When G43.4 H_ and G43.5 H_ are specified on a separate line for execution, machine will not move, and will perform the movement value after tool length compensation until there is Z movement amount in program.

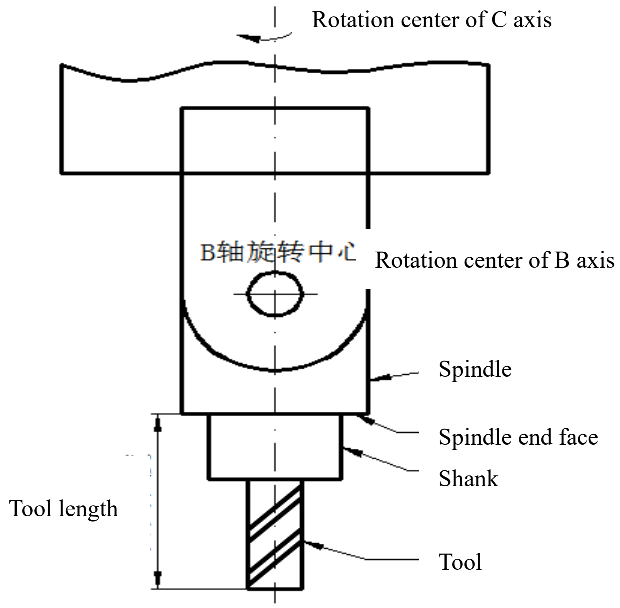
(3) The tool length compensation process for different machine tool structures is as follows:

a) For machine tool with dual swivel head AC or BC, perform tool length measurement first, and the measurement of tool length is shown as below:



Enter the tool length value shown in the figure into the geometric length compensation Z in tool compensation table, and offset current Z coordinate of workpiece coordinate system downward by the tool length value. Use G43.4 for calling in program to complete tool length compensation and RTCP function.

b) For machine tool with dual swivel head BC, perform tool length measurement first, and the measurement of tool length is shown as below:



Enter the tool length value shown in the figure into the geometric length compensation Z in tool compensation table, and offset Z coordinate of currently used workpiece coordinate system downward by the tool length value. Use G43.4 to call in program to complete tool length compensation and RTCP function.

For the measurement of tool length shown in the figure, users can install an infrared probe on machine for the automatic measurement, and also can manually use level indicator or gauge block for the measurement.

3.4. 5-Axis Inclined Surface Machining

In five-axis machining, the machining of inclined surfaces is often encountered. In order to simplify the programming and processing of inclined surfaces, this function can translate and rotate the workpiece coordinate system in the conventional plane through the setting of a special coordinate system. That is, after origin is moved on the X, Y, and Z axes of the current workpiece coordinate system and rotation angle of rotary axis is defined, a new coordinate system and coordinate plane are established. This coordinate system is called the trait coordinate system (TCS). After the special coordinate system is set, programming and processing can be performed in the usual way.

Note

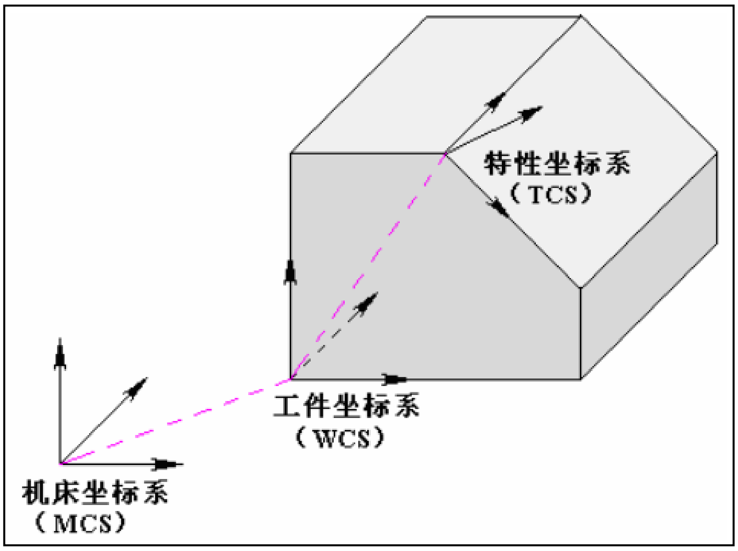
- Users should specify G43.4/G43.5 to enable trait coordinate system before use.
- After trait coordinate system is established, all programming coordinates are the coordinate values under trait coordinate system

There are two ways to specify the trait coordinate system on workpiece. one is to enter trait coordinate system data on CNC interface and use G68.1 command to select a group of data for trait coordinate system establishment, and HNC8 provides 20 trait coordinate system to use; the other is to use G68.2

command to establish trait coordinate system via Euler angle, and G69 is to cancel currently established trait coordinate system.

In addition, the tool axis direction needs to be controlled under the newly defined trait coordinate system, so that the tool axis swings to a direction parallel to the Z axis of the trait coordinate system (perpendicular to the inclined surface). If only the trait coordinate system (inclined surface) is specified in the program without tool axis direction control, the rotary axis will not generate angle rotation, or an alarm will be issued.

The following figure shows the relationship between machine coordinate system, workpiece coordinate system, and trait coordinate system.



MCS: Machine coordinate system
WCS: Workpiece coordinate system
TCS: Trait coordinate system

3.4.1. Trait coordinate system setting （G68.1 to G68.2）

1. Set trait coordinate system via three-point in the plane

Command format

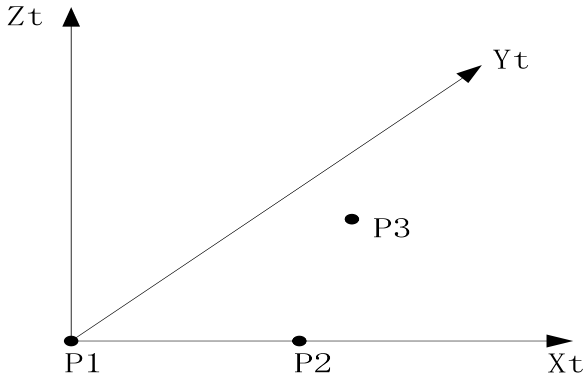
G68.1 Q_ ; Select a group of trait coordinate system

G69 ; Cancel trait coordinate system

Parameter	Meaning
Q	Select a group of trait coordinate system, ranging from 1 to 20

Description

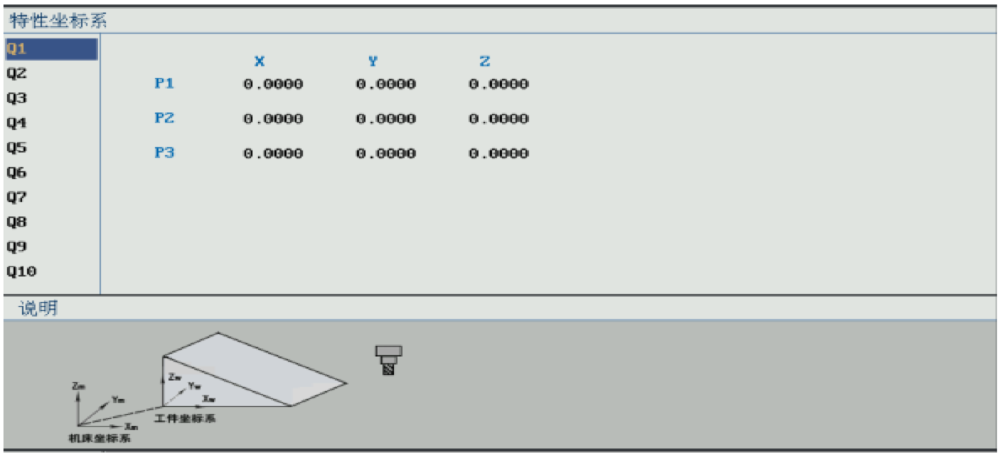
The establishment of trait coordinate system data is established by specifying the following three points. These three points establish an inclined surface, that is, the trait coordinate system, as shown in the figure:



P1: Zero of trait coordinate system;

P2: Any point in X positive direction of trait coordinate system (Z axis is the X axis of tool axis)

P3: Any point in quadrants 1 or 2 of the XY plane in trait coordinate system.



(1) Points P1, P2, and P3 are coordinate value in workpiece coordinate system.

(2) Point P1 is the zero of trait coordinate system, and the direction from P1 to P2 is the X positive direction.

(3) P1, P2, and P3 are preset inputs in the CNC system. The above figure is the display interface of trait coordinate system of CNC system. The three points are input into any Q_, and the system supports 20 groups of trait coordinate systems.

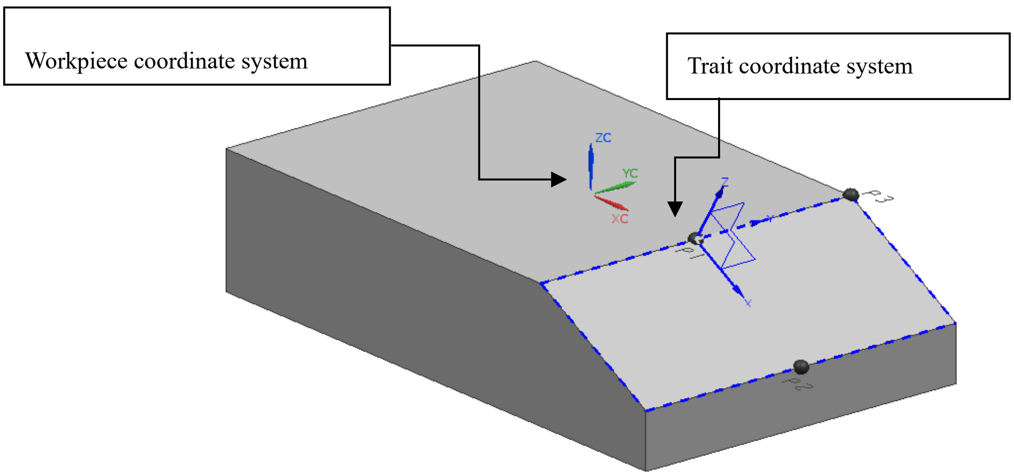
(4) A group of Q trait coordinate system is selected with G68.1.

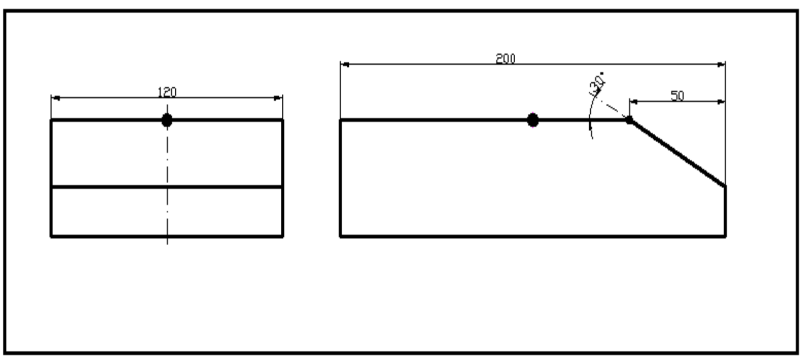
Application example

Example of point Q setting

For the designation of P1, P2, and P3 of a group of Q, X, Y, and Z values of P1, P2, and P3 is the values in the workpiece coordinate system shown as below, that is, the set trait coordinate system.

Q	X	Y	Z
P1	50	0	0
P2	100	0	-28.8675($50\times\frac{\sqrt{3}}{3}$)
P3	50	60	0





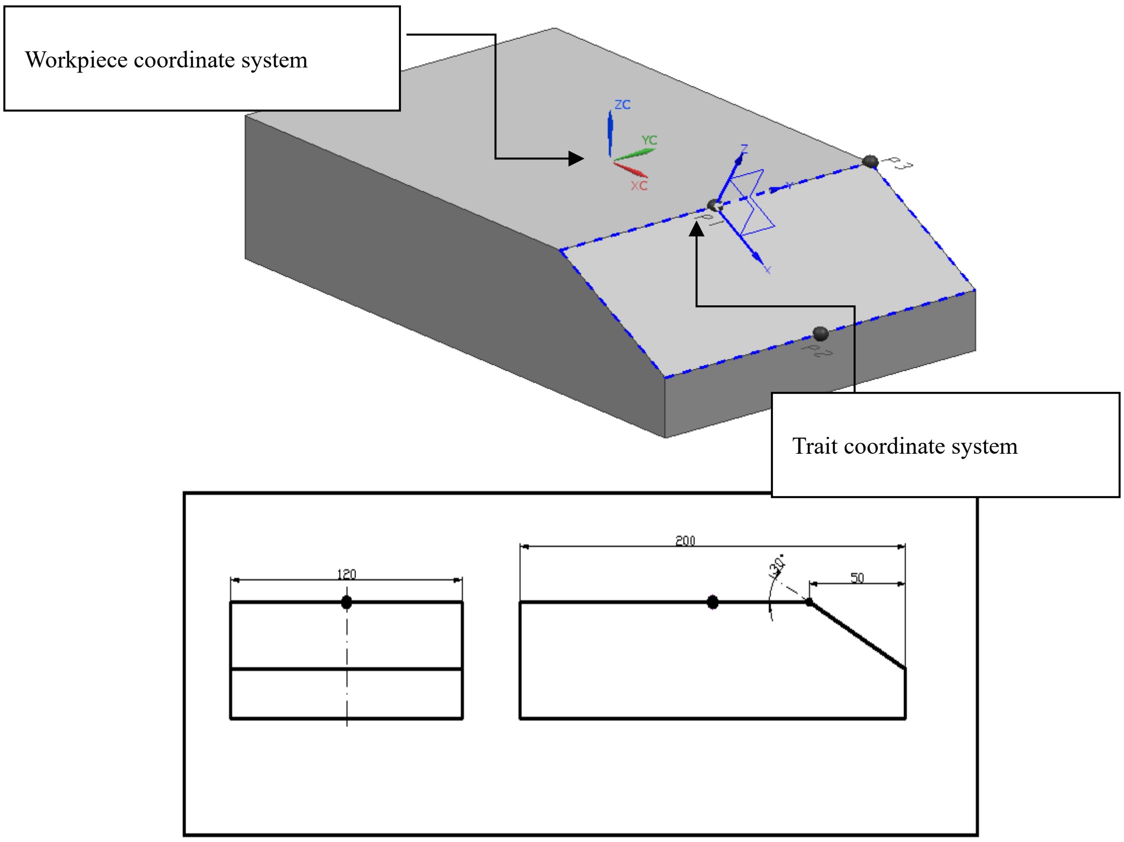
If the machine is with AC dual rotary table, then the running result is: C axis rotates 90°, A axis rotates 30°; or C axis rotates 270°, and A axis rotates -30°.

Example of NC program block format

G43.4 H1	Enable tool center control
G68.1 Q1	Enable trait coordinate system Q1
G53.1（G53.2）	Tool axis swings to the direction parallel to the Z axis of the trait coordinate system
G01 X-20 Y-5 F500	In trait coordinate system, X axis moves to -20mm, and Y axis moves to -5mm.

Programming example

In 5-axis machine tool with AC dual rotary table, tool center path (milling) is along green contour line as shown in the figure, and the depth is -2mm.



Select a certain Q in the CNC system to set its three points P1, P2, P3, respectively

P1：50，0，0

P2：100，0，-28.8675

P3：50，60，0

As shown in the right figure, the green dashed outline is the tool center path. The trait coordinate system is in the position as shown in the figure

%123

G54

G43.4 H1

G68.1 Q1

G53.1 （G53.2）

G01 X-20 Y-60 Z30 F2000 M03 S3000

Z3

Z-2 F300

X57.735 F1000

Y60

X0

Y-80

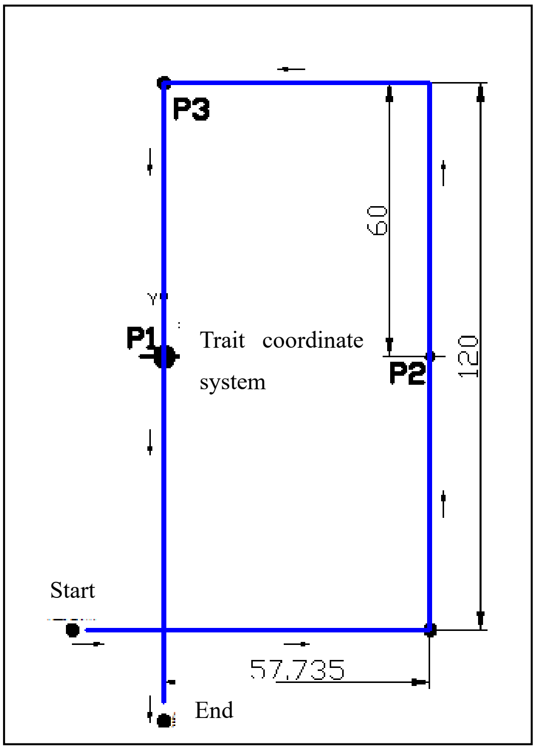
Z3

Z30 F2000

G69

G49

M30



2. Establish trait coordinate system via Euler angle

Command format

G68.2 X_Y_Z_I_J_K_ ; Establish trait coordinate system

G69 ; Cancel trait coordinate system

Parameter	Meaning
X	The X value in workpiece coordinate system is the X-axis zero of trait coordinate system
Y	The Y value in workpiece coordinate system is the Y-axis zero of trait coordinate system
Z	The Z value in workpiece coordinate system is the Z-axis zero of trait

	coordinate system
I	Yaw angle, rotation angle of axis C around axis Z in workpiece coordinate system
J	Pitch angle, rotation angle of axis A aroud X axis changed by yaw angle in workpiece coordinate system
K	Roll angle, rotation angle of axis C around axis Z changed by pitch angle in workpiece coordinate system

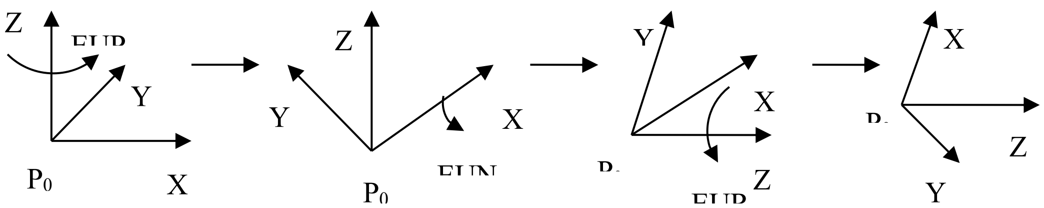
Description

G68.2 is a trait coordinate system established by Euler angle. Euler angle is the angle of rotation around coordinate axis in the coordinate system. The rotation combination mode of the CNC system is "ZXZ", which is defined as follows:

Yaw angle (EULPR): Rotation angle around Z axis

Pitch angle (EULNU): Angle of rotation around the X axis changed by yaw angle

Roll angle (RULROT): Angle of rotation around the Z axis changed by pitch angle



Based on the command G68.2 Xx Yy Zz Ia Jb Kc, The CNC executes the characteristic coordinate system as follows,

Xx Yy Zz specified with G68.2 command is the origin of trait coordinate system.

In the offset trait coordinate system, rotate an angle around Z axis

In the rotated coordinate system, rotate b degrees around X axis

In the rotated coordinate system, rotate c degrees around Z axis

The coordinate system at this time becomes the coordinate system established on the inclined plane, that is, the trait coordinate system.

For the above rotation angle, the counterclockwise direction of the rotation center axis is the positive direction

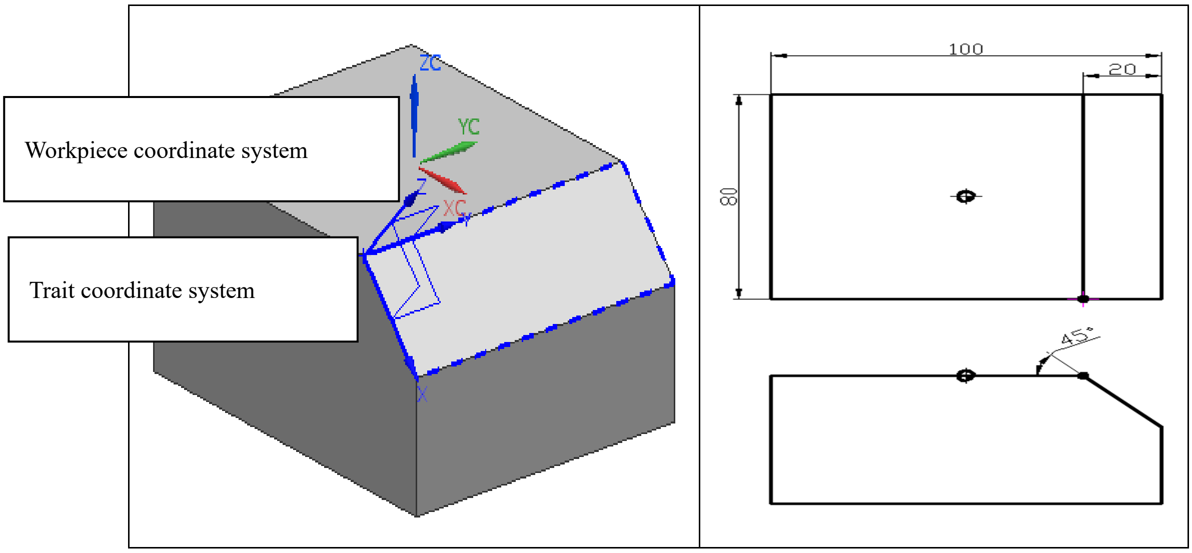
Application example

Example of NC program block format

G43.4 H1	Enable tool center control
G68.2 X30. Y-40. Z0.0 I90. J45. K-90.	Establish trait coordinate system with Euler angle
G53.2 (G53.1)	Tool axis swings to the direction parallel with Z axis of trait coordinate system
G01 X0 Y-15 F2000	In trait coordinate system, X axis moves to 0mm, and Y axis moves to -15mm

Programming example

In 5-axis machine tool with AC dual rotary table, tool center path (milling) is along blue contour line as shown in the figure, and the depth is -2mm.

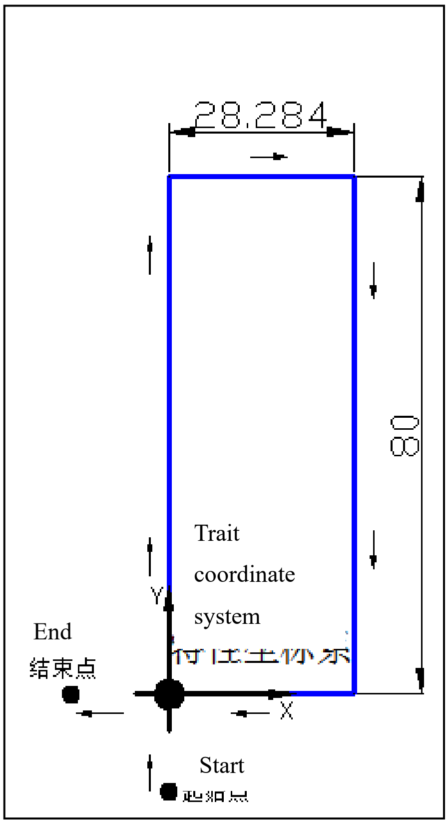


G68.2 block format: G68.2 X30. Y-40. Z0.0 I90. J45. K-90.

Take the dual rotary table (AC) as an example: firstly rotate around Z axis C90°, then rotate around X axis A45°, and finally rotate around Z axis C-90°.

Program: (as shown in right figure, the blue contour line is the path of tool center. The trait coordinate system is at the position as shown in the figure)

```
%0010
G54
G91 G28 Z0
G90 G00 Y-70. S3000 M03
G43.4 H1
G68.2 X30. Y-40. Z0.0 I90. J45. K-90.
G53.1 （G53.2）
G01 X0 Y-15. F2000.
Z50
Z3.
Z-2.0 F300.
Y80. F1000.
X28.284
Y0.0
X-15.
Z50. F2000.
G69
G49
G91 G28 Z0
G90 G00 A0 C0
M30
```



3.4.2. Tool axis direction control

When processing an inclined surface, users firstly establish a trait coordinate system with G68.1/G68.2 command, and then use G43.4 (or G43.5) to define the direction of tool axis. There are two control modes for direction of the tool axis, which are set by G53.1 or G53.2 command. After special coordinate system setting, tool axis direction setting, and tool axis control mode setting, the tool axis can swing to the direction parallel to Z axis of trait coordinate system (that is, perpendicular to the XY plane of trait coordinate system).

Command format

G53.1/G53.2

G53.1: Tool axis direction control (type 1). Only rotary axis moves;

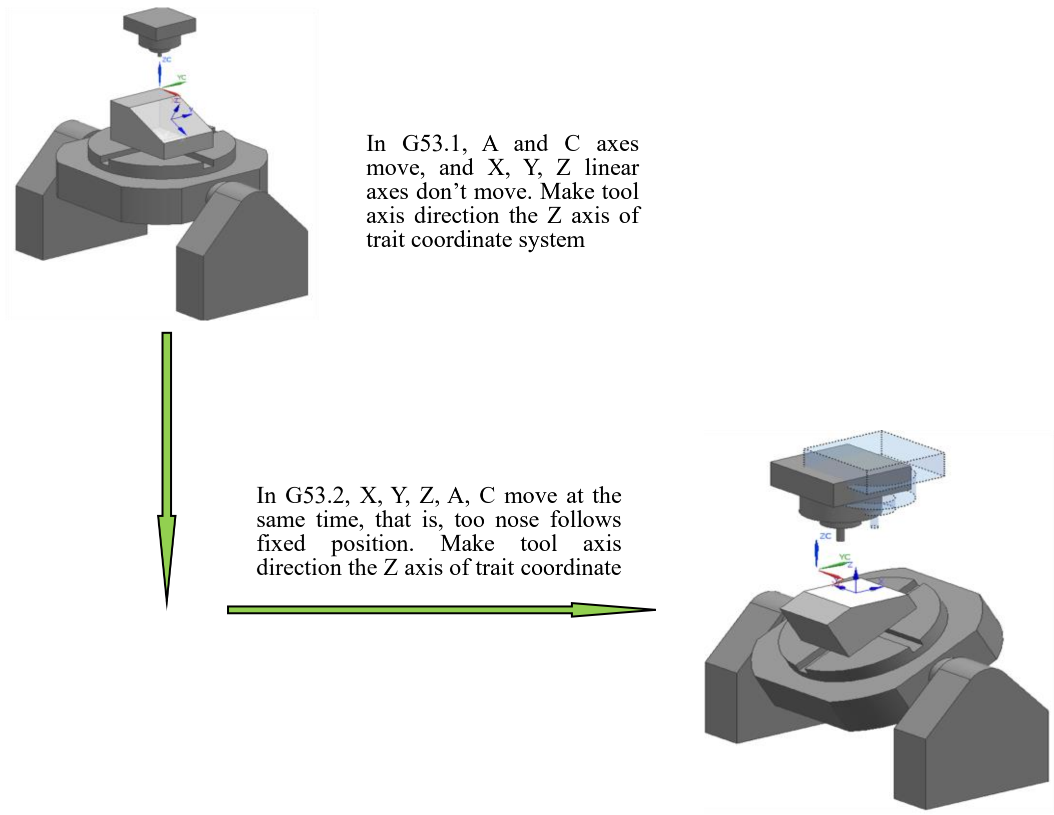
G53.2: Tool axis direction control (type 1). Linear axis and rotary axis moves simultaneously. Keep the relative position of tool nose and workpiece unchanged

While G53.1 is used, it only rotates according to the angle of the specified trait coordinate system, and the two rotary axes moves simultaneously, but the linear axis does not move.

While G53.12 is used, it rotates according to the angle of the specified trait coordinate system, and three linear axes joint the simultaneous motion, forming simultaneous 5-axis motion.

Description

For the 5-axis machine tool with AC dual rotary table, difference between G53.1 and G53.2



Application example

Example of NC block format

G43.4 H1	Enable tool center control
G68.2 X30. Y-40. Z0.0 I90. J45. K-90.	Establish trait coordinate system with G68.2

（G68.1 Q1）	(Establish trait coordinate system with G68.1)
G53.1 （G53.2）	Tool axis swings to the direction parallel to Z axis of trait coordinate system, and A and C axes move simultaneously (Tool axis swings to the direction parallel to Z axis of trait coordinate system, and X, Y, Z, A, and C axes move simultaneously)
G01 X0 Y-15 F2000	

Note

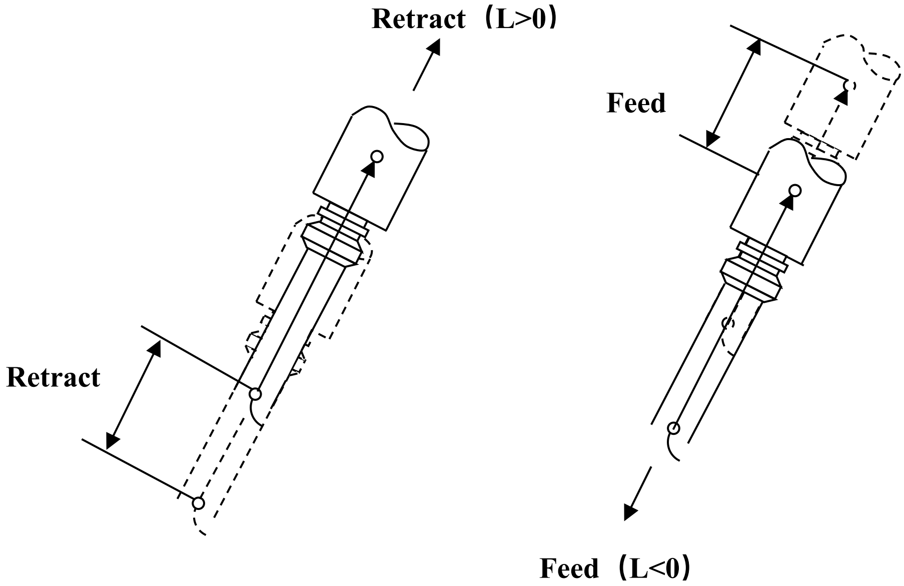
(1) Tool axis direction control must be used with trait coordinate system, and placed on the line behind trait coordinate system, that is: command G68.1/G68.2 is to establish the trait coordinate system, and the following line must be behind G53.1 or G53.2, so that the tool axis is perpendicular to the inclined plane. It should be on a separated line.

(2) For the machining of inclined surface, when the program executes the command line G68.1/G68.2, no movement occurs, but trait coordinate system is established. When the program executes G53.1/G53.2, the movement starts, and the moving speed inherits the F value. If there is only G00 in first program blocks, the execute parameter 040030 "channel default feedrate".

(3) When executing G53.2, If machine tool position is not suitable, it is possible to alarm “axis overtravel” when executing the line of simultaneous motion, or it is possible for the tool to interfere with workpiece and worktable

3.4.3. Normal retract G53.3

Tool feeds or retracts along tool axis direction (normal direction of inclined surface). As shown below



Command format

G53.3 L_

Parameter	Meaning
L	Tool retract. L > 0, tool moves away from tool nose. Tool feed. L < 0, tool moves to tool nose.

Description

Generally used for machine tools with swivel head structure. When the tool is near or in the workpiece, and this function is not enabled, if only moving Z axis upward, the tool may interfere with the workpiece. That can be avoided using the normal feed and retract.

Structure of machine must be set correctly in the parameter; otherwise, the normal feed and retract command cannot be executed properly.

G43.3H1 (RTCP function) must be specified in program; otherwise. the normal feed and retract command cannot be executed properly. Please find below,

```
%123
G54
G43.4 H1
```

G53.3 L-50 ; Feed 50mm

P2000

G53.3 L50 ; Retract 50mm

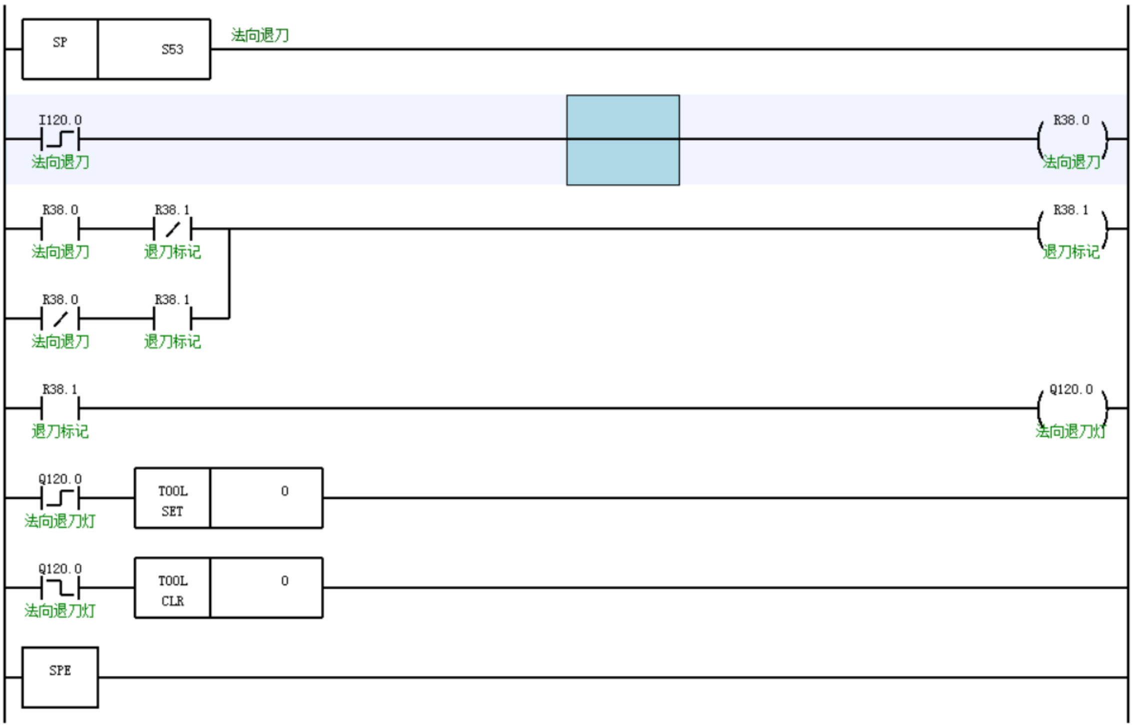
P2000

G49

M30

Manual feed and retract function is also supported. Manual feed and retract function is enabled in PLC via TOOLSET module, and disabled via TOOLCLR module.

When using the manual normal feed and retract function, first edit the system PLC and use the spare keys on the panel to achieve the purpose. The PLC is as follows:



Select a spare button on the panel as the button for turning on and off the feed and retract function, and tell whether the normal feed and retract function is turned on or off by the button light.

Note

1) Structure of machine must be set correctly in the parameter; otherwise, the normal feed and retract command cannot be executed properly.

2) Manual feed and retract function is also supported. Manual feed and retract function is enabled

3.5. Application of Other Functions in Inclined Surface Machining

In inclined surface machining, in addition to general basic commands that can be used, there are also some functional commands that can be used.

3.5.1. Application of tool radius compensation function

Left tool radius compensation and right tool radius compensation can also be sued in inclined surface machining, and the NC block format is as follows.

G43.4 H1	Enable tool center control, and add tool length compensation
G68.2 X30. Y-40. Z0.0 I90. J45. K-90. （G68.1 Q1）	Establish trait coordinate system
G53.1	Tool axis swings to the direction parallel to Z axis of trait coordinate system
G01 X-20 Y-20 F2000	In trait coordinate system, X axis moves to -20mm, and Y axis moves to -20mm
G41 X-15 Y-15 D01	Add tool radius compensation
…………	
G40 Y-80	Cancel tool radius compensation
…………	

3.5.2. Application of hole canned cycle

In inclined surface machining, hole canned cycle also can be used, which includes G73, G76, G80, G81, G82, G83, G85, G86, G87, and G89. The NC block format is as follows,

G43.4 H1	Enable tool center control, add tool length compensation
G68.2 X30. Y0. Z0.0 I90. J45. K-90. （G68.1 Q1）	Establish trait coordinate system
G53.2	Tool axis swings to the direction parallel to Z axis of trait coordinate system

G01 X10 Y20 F2000	In trait coordinate system, X axis moves to 10mm, and Y axis moves to 20mm (position of hole center)
G01 Z3	Position 3mm above it in the normal direction of the inclined surface
G90 G99 G73 X15 Y20 Z-25 R4 Q-3 K2 P2000 F300	Drilling on inclined surface
X30	Drilling
…………	

Programming example

%1234

G55 G90 ; Establish workpiece coordinate system

G43.4 H1 ; Enable RTCP function with G43.4; specify tool length compensation number as H1

G1 X0 Y0 Z200

M3 S1000

G90 G01 Z200

A-20

G68.2 X10 Y10 Z20 i0 j90 k0 ; Workpiece position (10,10,20), establish trait coordinate system with Euler angle

G53.2 ; Tool direction control

G01 X20 Y15

G98 G81 R20 Z-3 F200 ; Center drilling

G69 ; Cancel trait coordinate system

G1 Z200

A0

G49 ; Cancel RTCP function with G49

M30

Note

When using hole canned cycle for the inclined surface, users must set the machine user parameter 01008 *Drilling tapping canned cycle type* to 0

3.5.3. Application of tapping canned cycle

For the conventional 3-axis synchronous tapping, C axis is generally set as the synchronous axis; for 5-axis machine (AC or BC structure), C axis is the programming axis, and address of rotary axis C is occupied, users need to use programming addresses of other axes as the tapping axis, so for the purpose of multi-axis application in 5-axis orientation tapping, programming axes A, B, C, U, V, or W can be used for rigid tapping. The tapping axis type is defined by the parameter J.

Programming format

G84 X_ Y_ Z_ R_ Q_ P_ F_ L_H_J_;

Parameter	Value	Meaning
J	1	A-axis tapping
	2	B-axis tapping
	3	C-axis tapping
	4	U-axis tapping
	5	V-axis tapping
	6	W-axis tapping

Take B axis tapping as an example

Set 040005 *B coordinate axis number* to -2

Example of NC block format

G43.4 H1	Enable tool center control
G68.2 X30. Y0. Z0.0 I90. J45. K-90. （G68.1 Q1）	Establish trait coordinate system
G53.1	Tool axis swings to the direction parallel to Z axis of trait coordinate system
G01 X15 Y20 F2000	In trait coordinate system, X axis moves to 15mm, Y axis moves to 20mm, that is, the position of tapping hole center
G01 Z3	Position 3mm above it in the normal direction of the inclined surface
G90 G99 G84 X15 Y20 Z-15 R4 P2000 F1.5J2 S500	Tapping on inclined surface
X30	Tapping
………	

Programming example

%1234

G55 G90 ; Establish workpiece coordinate system

G43.4 H1 ; Enable RTCP function with G43.4; specify tool length compensation number as H1

G1 X0 Y0 Z200

M3 S1000

M05

G90 G01 Z200

A-20

G108 B0 ; Switch to B axis for position control

G68.2 X10 Y10 Z20 I0 J90 K0 ; Workpiece position (10, 10, 20), establish trait coordinate system with Euler angle

G53.2 ; Tool direction control

G01 X20 Y15

G98 G84 Z-100 R0 P500 F1 J2 ; B axis tapping

G69 ; Cancel trait coordinate system

G1 Z200

A0

G109 B0 ; Switch to spindle for speed control

G49 ; Cancel RTCP function with G49

M30

Note

(1) Sometimes the tapping axis is occupied by the programming address of rotary axis, and users need to use programming addresses of other axes as tapping axes. If C axis is a rotary axis, and the programming address of the tapping axis is occupied, B axis needs to be set as the tapping axis and set to -2.

(2) When the spindle is switched to position mode with G108, if B axis is the tapping axis, then mode of command calling is G108 B0; similarly, when switching to spindle mode, mode of command calling is G109 B0

3.5.4. Supplement of hole and tapping canned cycle in inclined surface machining

When it is necessary to drill and tap on the G18 and G19 planes, the drilling cycle and tapping cycle can also be completed by the principle of inclined surface machining.

When using inclined tapping, it is necessary to debug the parameters of rigid tapping, and the debugging is different from that of three-axis. In the case of three-axis, synchronization error of Z and C axes need be ensured, while in the case of five-axis, it is necessary to ensure synchronization error of axial feed axis and C-axis. For example, in the G18 plane, synchronization error of Y axis and C axis needs to be ensured; in any inclined surface, the synchronization error of X, Y, Z axis and C axis needs to be ensured.

Cimatron CAM software has integrated the post-processing module of the HNC8 inclined surface machining. After the inclined surface machining technology is defined in the CAM software, the inclined surface machining program that can be executed by system is directly generated.

3.6. Tool Orientation Interpolation Mode

In 5-axis tool orientation interpolation function, there is three interpolation modes including linear interpolation, great circle interpolation, and NURBS curve interpolation. In actual use, users select different interpolation mode based on requirements of machining technology.

3.6.1. Linear interpolation

In 5-axis linear interpolation, the rotary axis interpolation is to establish the same proportional mapping between the linear axis displacement increment and the rotation angle increment. In this interpolation mode, during the rotation, only the position of the tool center can be controlled, but the direction of the tool axis cannot be controlled.

Command format

G140 Enable linear interpolation

In position specifying mode (rotary axis programming mode), the linear interpolation mode is the default.

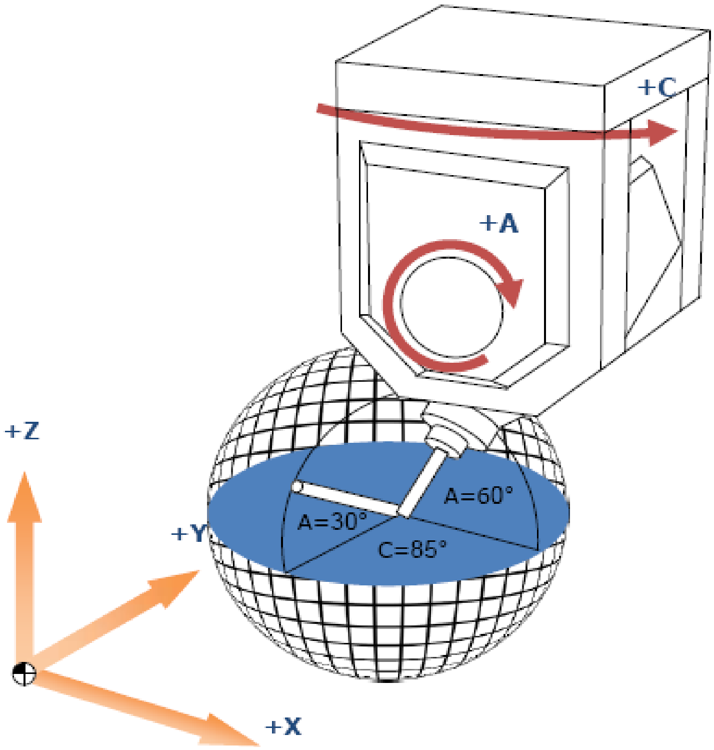
3.6.2. Great circle interpolation

Great circle interpolation is an interpolation mode developed based on the rotation of tool axis. In this mode, the tool axis trajectory interpolated between two programming points is always on the same planar arc. The swing of the tool axis is guaranteed to be on the same planar arc, and the trajectory of tool axis on the spatial sphere swings on the great circle arc formed by the two tool axes, so it is called great circle interpolation.

Command format

G141 Enable great circle interpolation

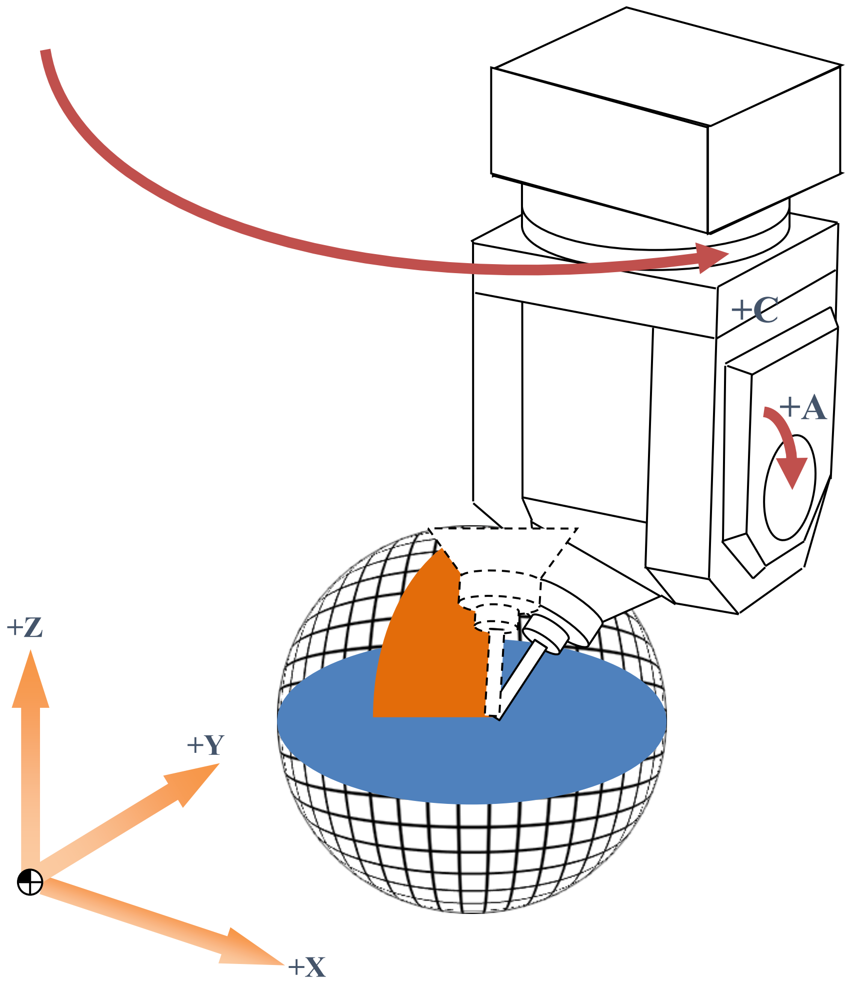
In tool vector specification mode, the great circle interpolation is the default.



Description

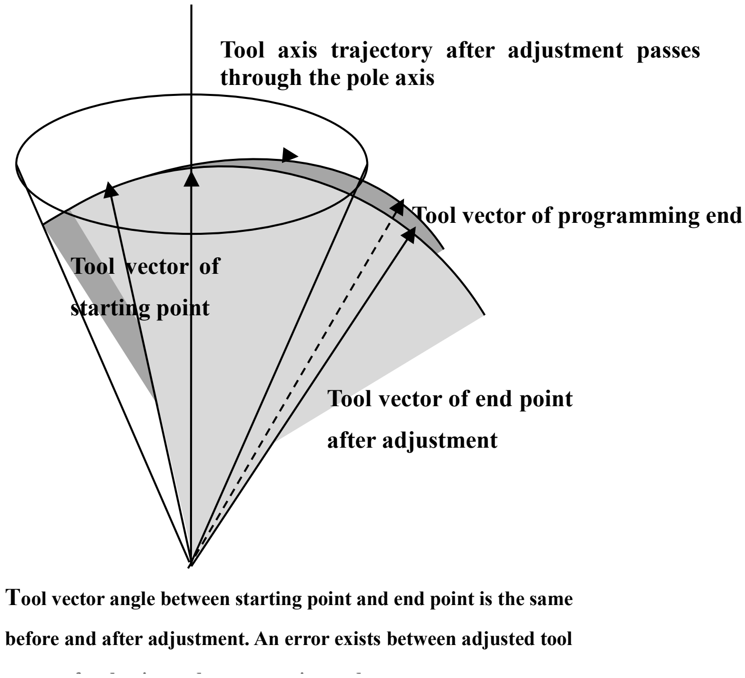
Pole processing during great circle interpolation

When the tool passes near the pole, due to the uncertainty of the rotary axis direction, if there is no corresponding treatment, it will cause the rotary axis to overspeed. For example: The initial direction (A45, C0) and end direction (A-45, C0) of the tool are parallel to the YZ plane. When tool axis approaches A0 position, since the position of the C axis is uncertain, only one directional interpolation will have C axis immediately turn to the 180-degree position. At this moment, it will inevitably cause the C-axis to exceed the speed, so it needs to be handled accordingly. Only in the process of great circle interpolation, will the pole be processed.



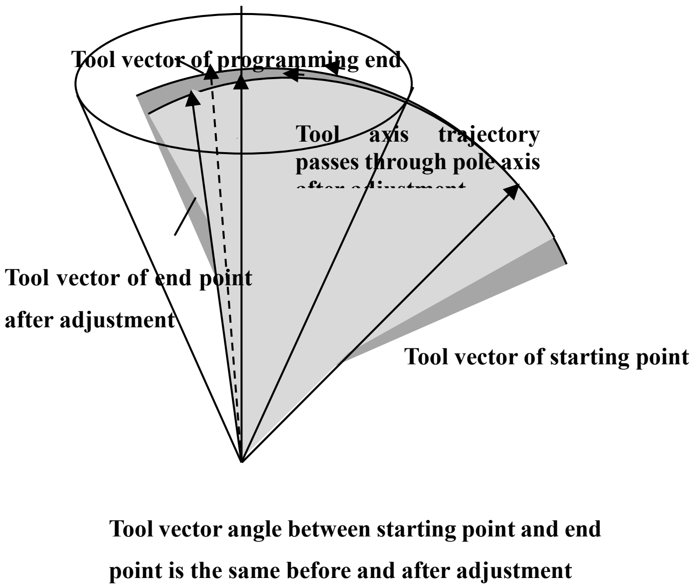
In 5-axis machine tool, the first rotation position is the pole. The channel parameter 040407 defines the range of pole angle, and the pole area is defined by the angle. A conical area with the pole axis as the axis and the angle as the taper angle is within the range of pole.

（1）The starting point is in the pole area, and the end point is out of the pole area.



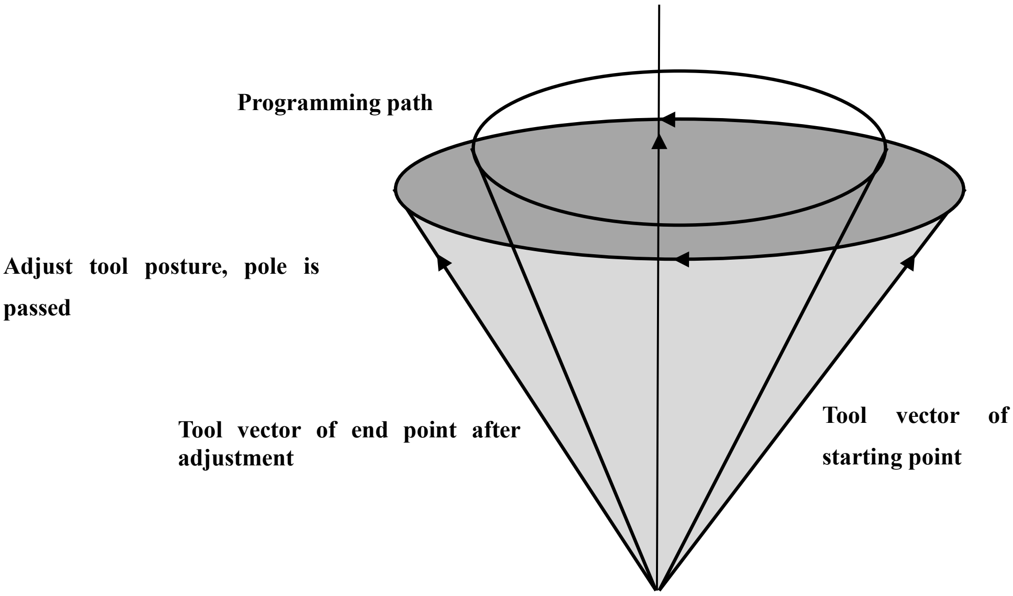
In this case, the part in the pole area is automatically converted to linear interpolation, and the area outside the area keeps in great circle interpolation

（2）The starting point is out of the pole area, and the end point is in the pole area.



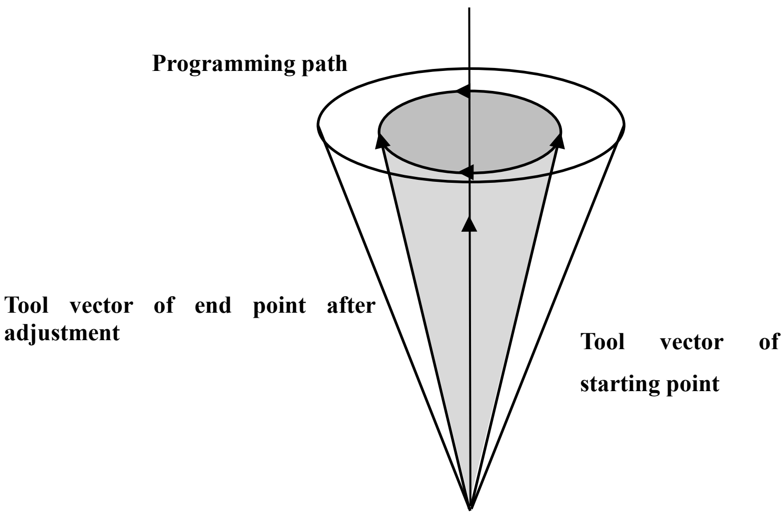
In this case, the tool posture is automatically adjusted so that it can pass through the pole.

（3）The starting point and the end point are out of the pole area, but the pole area is passed



In this case, the tool posture is automatically adjusted so that it can pass through the pole.

（4）Starting point and end point are in the pole area



In this case, the entire trajectory is automatically converted to linear interpolation.

3.6.3. NURBS hyperbolic interpolation

5-axis small line segment includes tool center data and tool axis data. The linear interpolation only control tool center path, but not the tool axis direction. The direction of tool axis is discontinuous during interpolation, which may cause sudden jumps. Fit the tool center and the tool axis to NURBS curves, and perform synchronous interpolation on the two curves, which can improve the processing speed and surface quality.

Command format

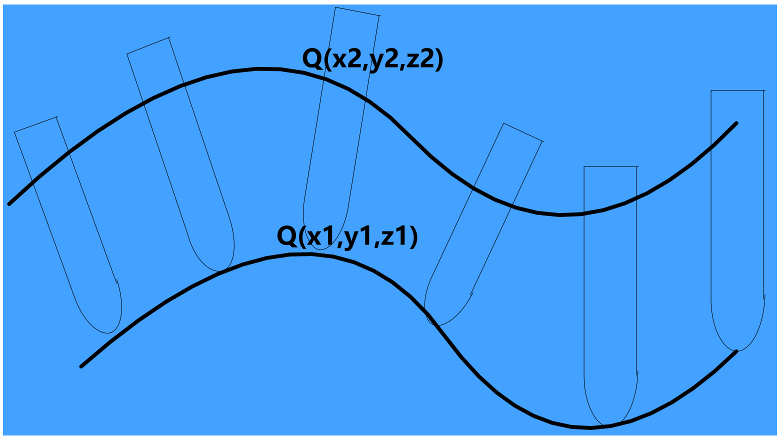
NURBSB P_K_Q_W_

P: Orders of NURBS curve

K: Node

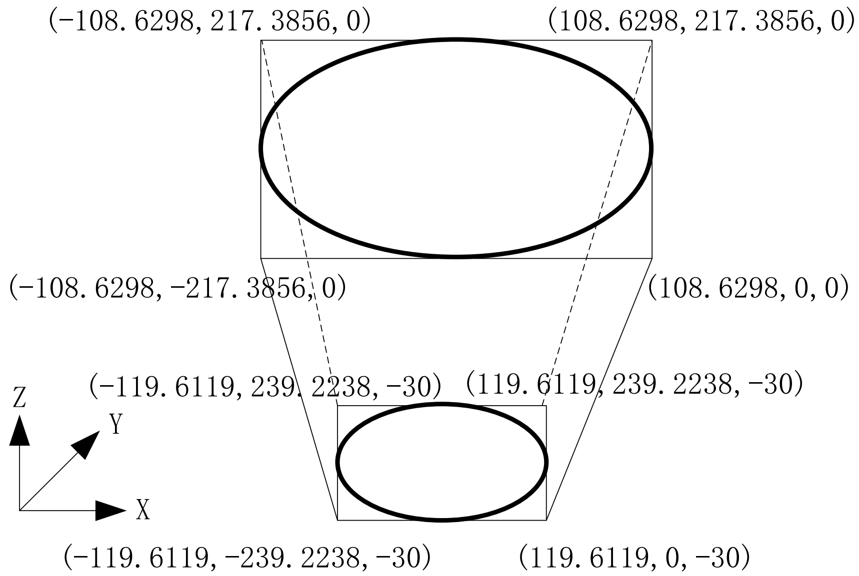
Q: Control point (X1,Y1,Z1,X2,Y2,Z2)

W: Weight



Programming example

Use the dual NURBS spline hyperbolic interpolation function to process the truncated cone below, with the bottom circle radius R=119



.

```
%0001
  G54
  F1000
  G43.4H1
  X150 Y0 Z30
  G01 A20.1061 C270
  X119.6119 Y0 Z-30
  NURBSB P4 K{0,0,0,0,0.5}   Q{119.6119,0,-30,108.6298,0,0} W1F600
  K0.5          Q{119.6119,239.2238,-30,108.6298,217.3856,0} W0.333
  K0.5          Q{-119.6119,239.2238,-30,-108.6298,217.3856,0} W0.333
  K0.5          Q{-119.6119,0,-30,-108.6298,0,0} W1
  K1      Q{-119.6119,-239.2238,-30,-108.6298,-217.3856,0}   W0.333
  K1      Q{119.6119,-239.2238,-30,108.6298,-217.3856,0}     W0.333
  K1          Q{119.6119,0,-30,108.6298,0,0}       W1
  G01 Z30
```

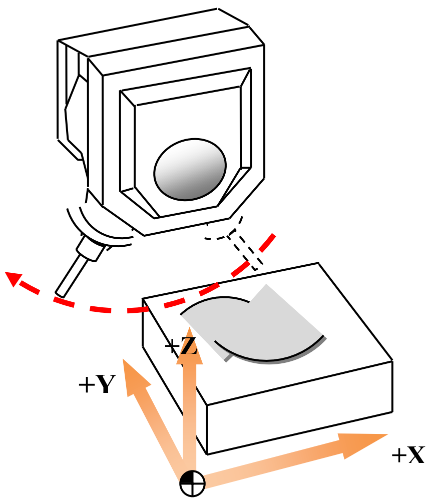
G49

G01 A0 c0

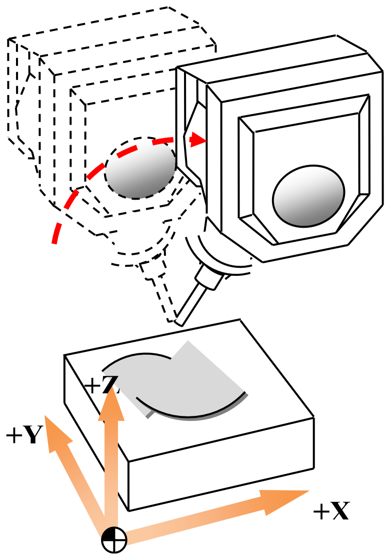
M30

3.7. 5-Axis RTCP Control Function

The RTCP function is also the tool center control. It is a characteristic function of five-axis machining. The characteristic of this function is that when there is a rotary axis movement (one or two rotary axes), the real-time synchronous compensation of three linear axes is used to make a certain point of the tool or workpiece always remain stationary in space, that is, the tool or workpiece rotates around this specific point; this specific point is generally the point where the tool nose contacts the workpiece for cutting (cutting point). This function is also referred to as tool following function. The figure below shows the positional relationship between tool and workpiece with or without the RTCP function.



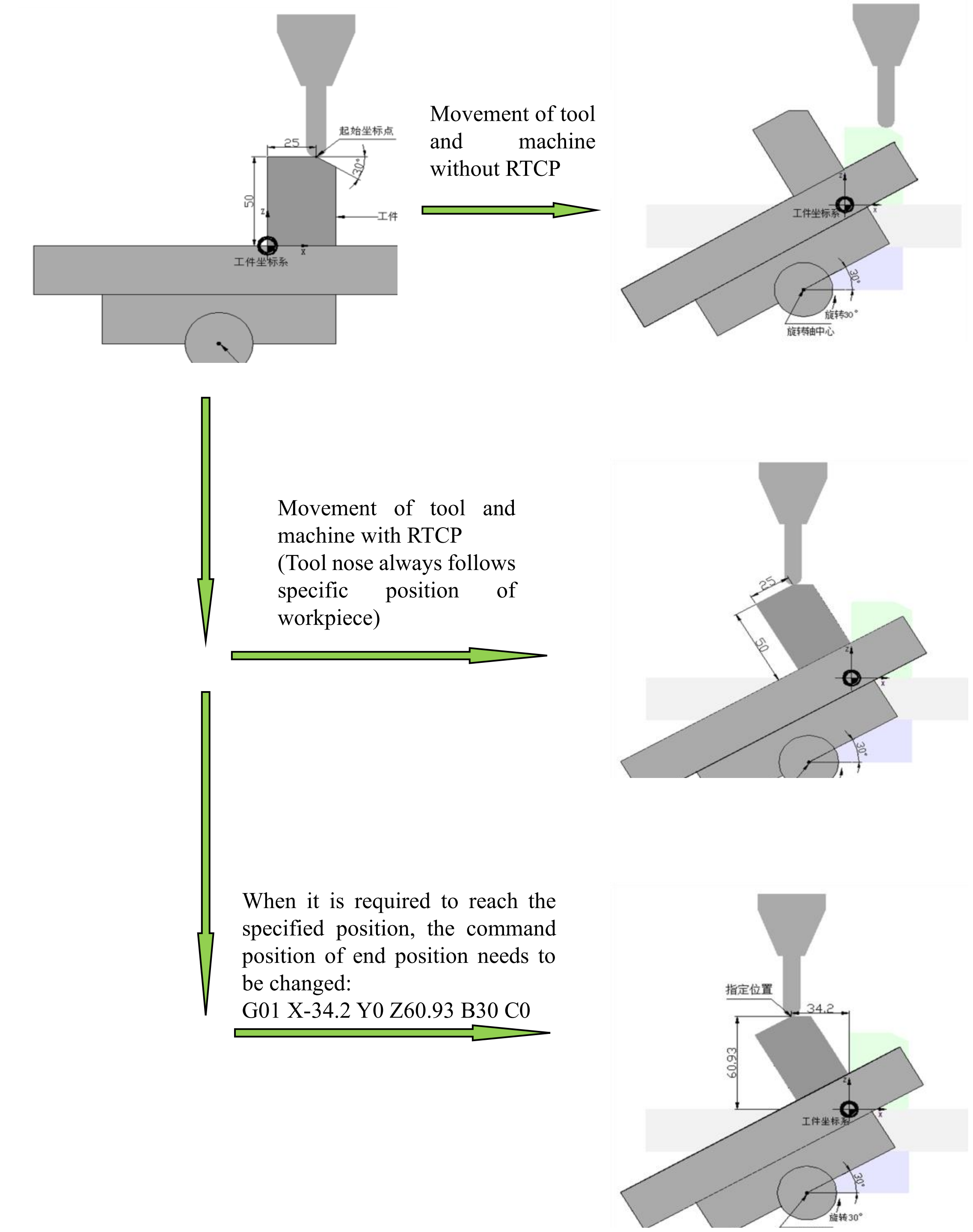
Without RTCP, tool rotates around rotary axis center, and tool noses moves out of fixed point



With RTCP, tool nose stays at the fixed point, and system will automatically perform linear axis compensation when rotary axis moves

In five-axis machining, the trajectory of tool center is changed due to the addition of rotary axis and the error of machine tool structure. When RTCP mode is turned on, the system sets the control point at the tool center, and implements real-time tool length compensation to make sure that the tool center point moves along the specified path. Users only need to perform five-axis programming in the workpiece coordinate system, and does not need to consider the error of machine tool structure, which greatly simplifies the CAM programming and improves the machining accuracy.

The below figure shows that in 5-axis machine with BC dual rotary table when a rotary axis moves, the difference between the movement with RTCP and without RTCP when only one rotary axis moves



Start coordinate: G01 X25 Y0 Z50 B0 C0   End coordinate: G01 X25 Y0 Z50 B30 C0

To realize the RTCP function of five-axis machining, it is necessary to know the motion structure of machine tool, the positional relationship between rotary axes, the eccentricity of rotary axis, and the distance from tool nose to center of rotation. Enter these values into the CNC parameters. In the G code program, the RTCP mode is turned on through the corresponding command. The system offsets the control point to the tool center, and ensures that the tool center moves along the specified path by real-time tool length compensation.）。

In addition, in five-axis machining, due to the addition of rotary axis, the moving speed of the tool center may be inconsistent with the actual moving speed of machine tool, and sometimes the speed of the sub-axis may exceed the set maximum speed limit. In this case, CNC system will reduce the processing speed to ensure that the sub-axis speed is within the set range.

Command format

G43.4（G43.5） H_ ; Enable RTCP function

G43.4: Angle programming of rotary axis

G43.5: Tool vector programming

G49 ; Cancel RTCP function

G43.4 (G43.5) enables the RTCP function; H specifies the tool length compensation number to offset the tool center toward the control point by one tool length compensation along tool axis; G49 cancels the RTCP function.

Description

1）Speed control

In the program, F specifies the moving speed of the tool center in the workpiece coordinate system. In five-axis machining, due to the addition of rotary axis, the moving speed of the tool center may be inconsistent with the actual moving speed of machine tool, and sometimes the speed of the sub-axis may exceed the set maximum speed limit. In this case, CNC system will reduce the processing speed to ensure that the sub-axis speed is within the set range.

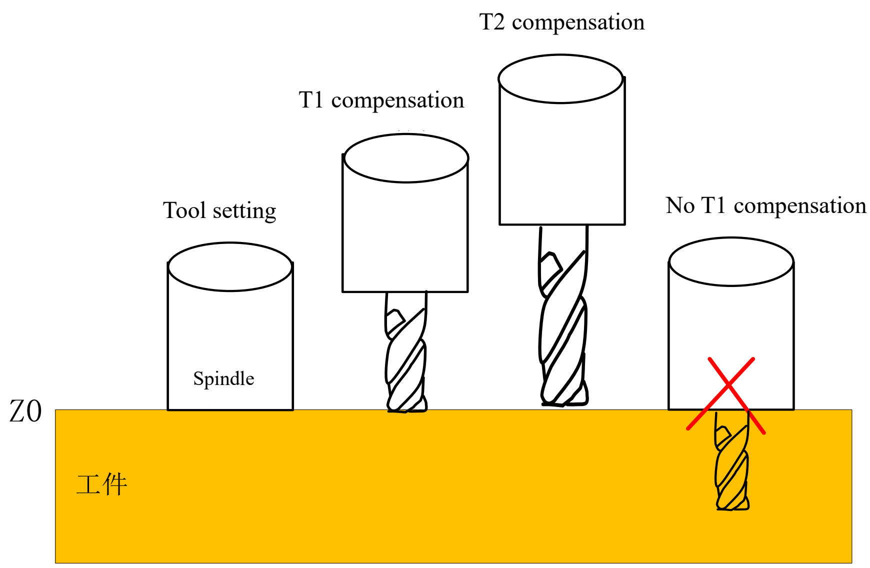
2）Tool setting mode

System provides two tool setting modes for users: tool setting of spindle end face and tool setting of tool nose. The former is the default.

Tool setting of spindle end face

When the channel parameter 040403 is set to 0, tool setting of spindle end face is enabled. After tool setting is completed on Z, it is necessary to offset a tool length in the negative direction, and set the spindle end face as the workpiece coordinate origin. In this tool setting mode, there is no need to perform tool setting again after tool change, and the previously set coordinate system can continue to be used.

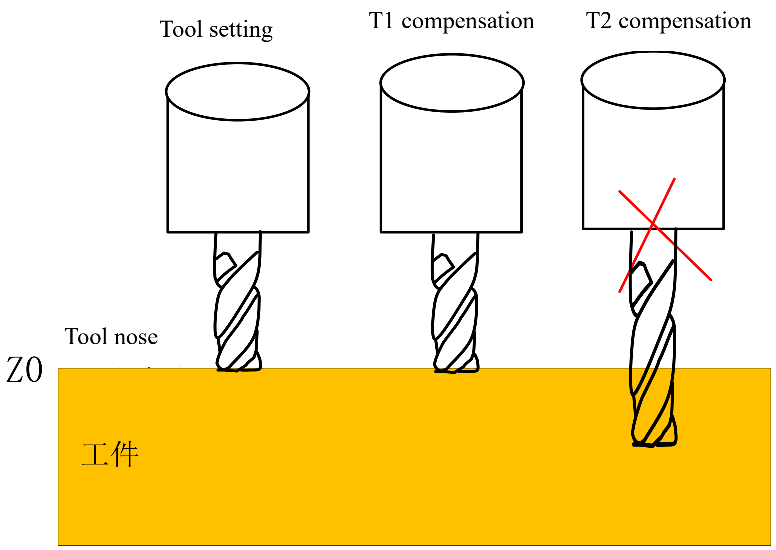
Note: During tool setting of spindle end face, if RTCP mode is not enabled, when Z axis moves to the area near workpiece origin, the interference between tool and workpiece may occur.



Tool setting spindle end face

Tool setting of tool nose

When the channel parameter 040403 is set to 1, tool setting of tool nose is enabled. In this mode, the tool nose is set as the workpiece coordinate origin after tool setting on Z is completed. During tool setting of tool nose, tool setting must be performed again after tool change, and new workpiece origin is set.



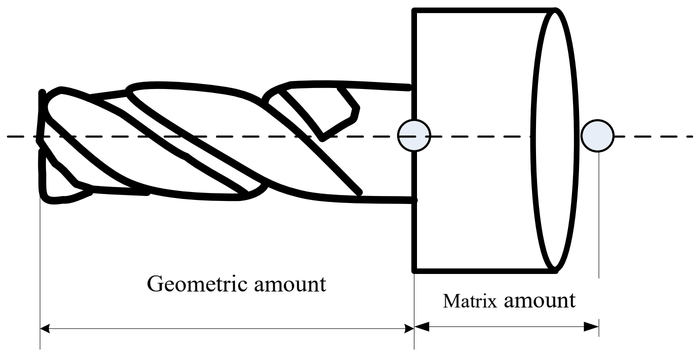
Tool setting of tool nose

3）Tool length compensation amount

Tool length compensation includes geometric amount, wear amount and matrix amount: the geometric amount is the actual measured length of the tool, the wear amount is the length change of the tool after a long time cutting, and the matrix amount is the retract amount along the compensation direction. After the RTCP function is activated, its compensation is the sum of the three components.

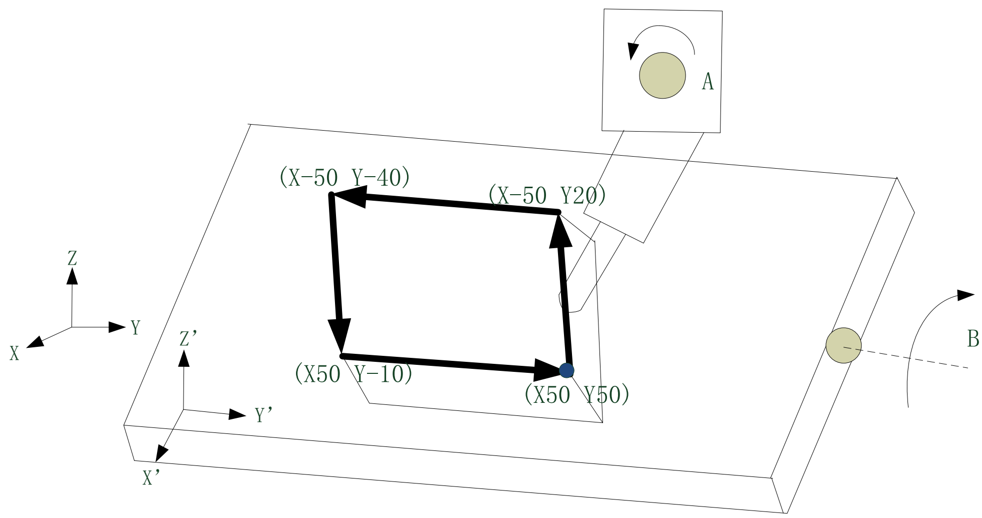
Generally, users only need to set the geometric amount of the tool length compensation on Z. When using the bidirectional tool length compensation of 90°head, the tool length compensation on X or Y is required. Refer to Section 3.7 for this function.

Tool length compensation = Geometric amount + wear + matrix



Programming example

Machining a quadrilateral with 30 degrees of A axis and -45 degrees of B axis



%0001

G54

G90 G01 X-50 Y-50 Z-4 A0 B0 ; **Move to initial position**

G43.4 H1 ; **Enable tool center control, and use No. 1 of tool compensation**

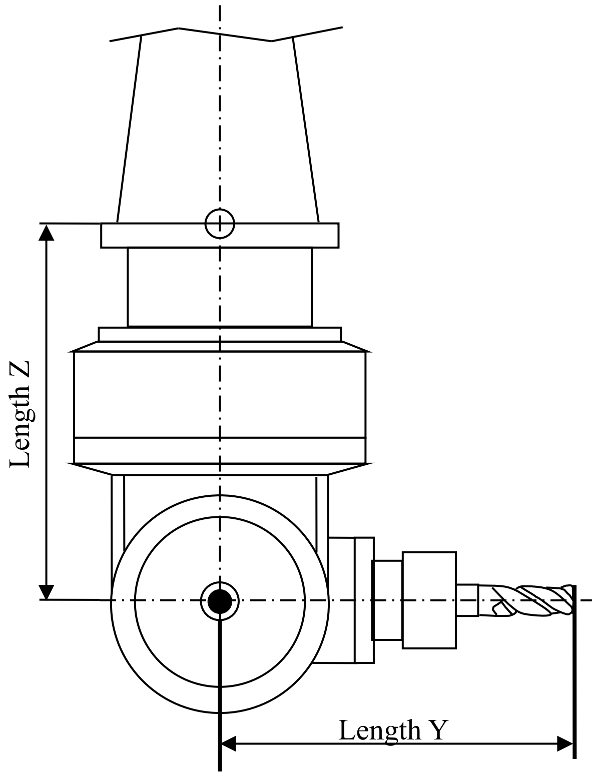
```
A30 B-45                      ; Change tool direction
X50 Y50                       ; Start processing quadrilateral
X-50 Y20
X-50 Y-40
X50 Y-10
X50 Y50
A0 B0                         ; Rotary axis returns to the initial position
Z50                           ; Tool lifting
G49                           ; Cancel tool center control
M30
```

4. Auxiliary Function of 5-Axis Program

4.1. Bidirectional Tool Compensation of 90° head

The 90°angle milling head is an important spindle accessory. It has two right-angled sides forming L-shaped structure. The 5-axis machine's processing technology is enriched and it processing range is expanded by working with the 90°angle head, and the conversion between vertical and horizontal processing is easily achieved. The tool length of the ordinary milling head is only along the spindle direction, and the tool length compensation only needs to be performed on Z axis; while the 90° angle milling head contains two directions for tool length, and bidirectional tool length compensation is required for the 90°angle head.

When the channel parameter 040409 is set to 1, bidirectional tool length compensation is enabled. At this point, the length compensation X and Y values are effective, and system will perform the compensation based on the set value.



Note

(1) The initial tool direction must be set to (0,0,1) for this function. Channel parameters 040400 to 040402 indicate the initial tool direction.

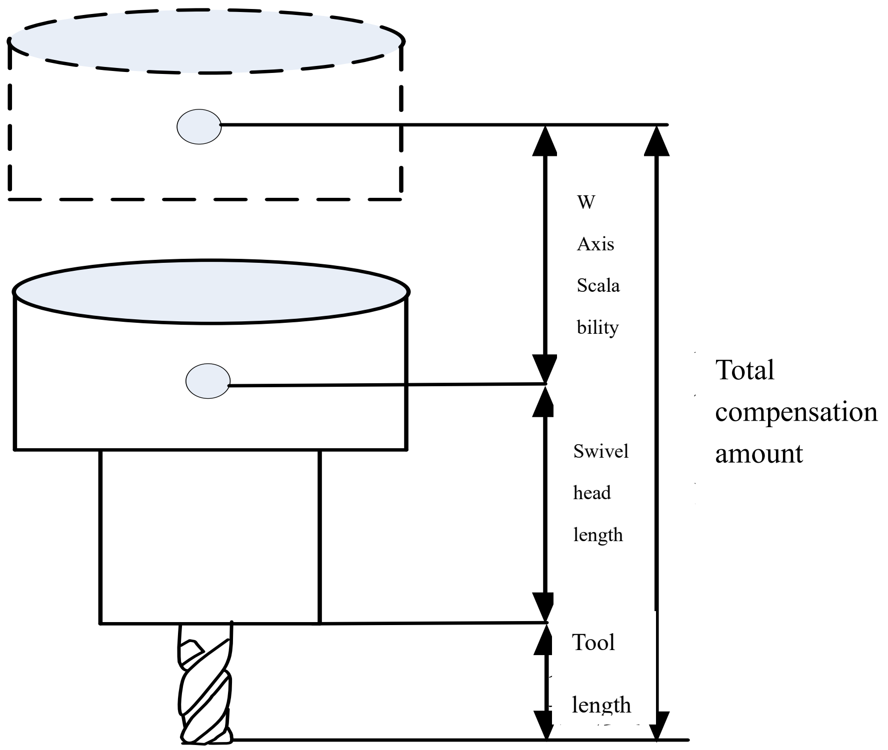
(2) In tool compensation table, any of X, Y, and Z can set geometric amount, wear amount, and matrix amount.

(3) Generally, the length on two directions of L-shaped 90°angle head need to be set, and the signs are determined based on the direction that tool nose points to spindle center.

4.2. Automatic Compensation of W Axis Scalability

Install W axis on Z axis of 5-axis machine, and use scalability of W axis to compensate the travel on Z and enlarge the machining range. After movement on W axis occurs, the tool center also changes. System provides real-time compensation of W axis to ensure tool moves based on workpiece position.

When the parameter 040404 is set to 1, W axis compensation function is turned on. Turning off is the default.



Note

(1) In order to effectively use W axis compensation function, it is necessary to ensure that the W axis is at the machine origin during calibration of the structural parameter.

(2) When setting workpiece coordinate origin, W axis is at the machine origin.

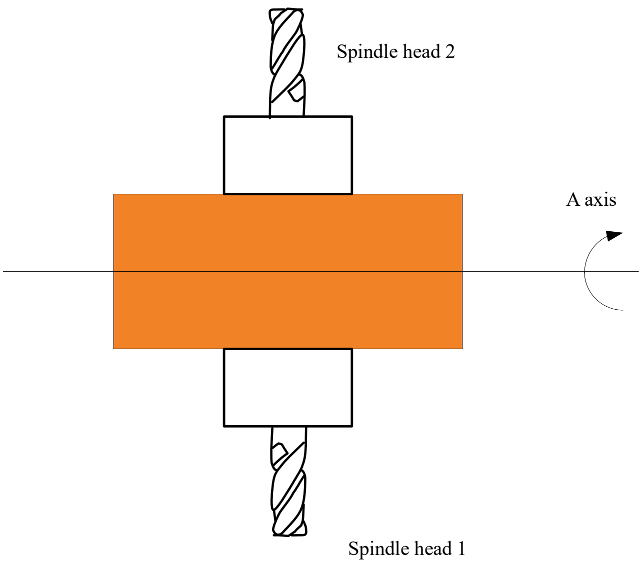
4.3. Swivel Head Indexing

After swivel head indexing function is turned on, the RTCP compensation position has nothing to do with the workpiece origin of rotary axis. During multi-spindle head RTCP swivel machining, when the spindle head needs to be switched, there is no need to consider the current angular position of rotary axis of the spindle head, and just create the tool path program directly in the workpiece coordinate system

The channel parameter 040408 Swivel head indexing:

（1） When 0 is set, swivel head indexing is turned off (default). During RTCP compensation, Calculation angle of rotary axis = Workpiece coordinate of rotary axis + Workpiece origin of rotary axis

（2） When 1 is set, swivel head indexing is turned on. During RTCP compensation, Calculation angle of rotary axis = Workpiece coordinate of rotary axis



Example

%0001

G43.4H1

G54 Spindle head 1, A axis coordinate system 0 degree

M98P1002

G49

G55 Spindle head 2, A axis coordinate system 180 degrees

G43.4H1

M98P1002

G49

M30

%1002

G01 X30Y30Z30A30

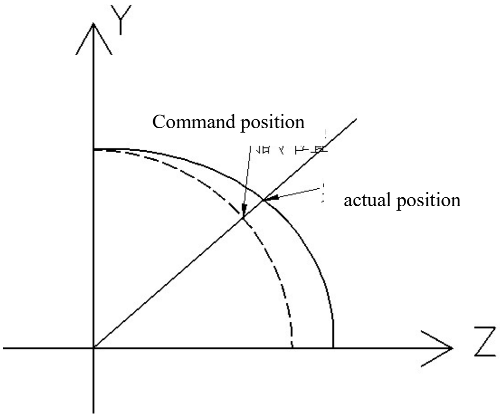
G01 X0Y0Z0A0

M99

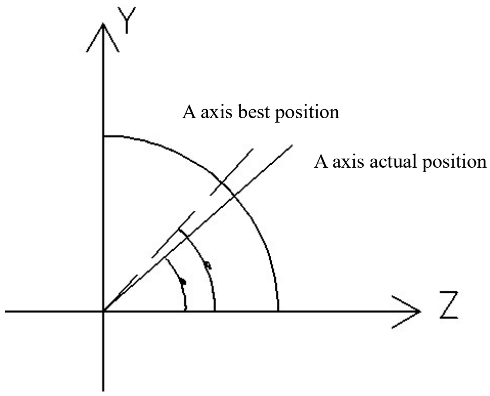
4.4. Dynamic Accuracy Test

For 3-coordinate machine tool, matching of dynamic features between linear axes is tested by the roundness test. For 5-coordinate machine tool, due to the addition of rotary axes, and the linear axis deviation caused by the movement of the rotary axis must be nonlinearly compensated at each interpolation point, servo matching of dynamic features between rotary axes and linear axes is also needed. Control unit of rotary axis is different from that of linear axis, and roundness cannot be used for rotary axis. System provides dynamic accuracy to test and sort dynamic features for 5 axes, which becomes the basis of parameter tuning.

According to the principle of RTCP, the movement of a rotary axis is associated with the movement of two linear axes, and the actual movement trajectory of the two linear axes is a circular arc, then the dynamic accuracy test includes: mismatch between two linear axes and mismatch between and linear axis and rotary axis.



Mismatch of linear axis



Mismatch of rotary axis

4.5. Digital MPG

Based on the standard system, this function can receive data requests sent by the MPG from the serial port and send the corresponding axis name, type, current coordinates and other information to the MPG, so as to realize the function of remotely viewing part of the machine tool information by MPG.

Specific operations:

1. Ensure correct connection of MPG control line and video display line between MPG and MCP panel of CNC controller

2. Set the NC parameter 000051 (serial port hardware type) to 5, save it, and restart the controller.

Digital MPG parameter

4.6. Tool Setting of Inclined Surface

A submenu for tool setting of inclined surface is added on the workpiece tool setting interface. With this function, vertical tool setting position of is calculated based on the current actual coordinate of axis, and is automatically input to X, Y, Z workpiece coordinates.

Operations

1. set the NC parameter 00353 5-axis function application to 1

2. Press Set -> Coordinate axis, find the button of inclined surface tool setting

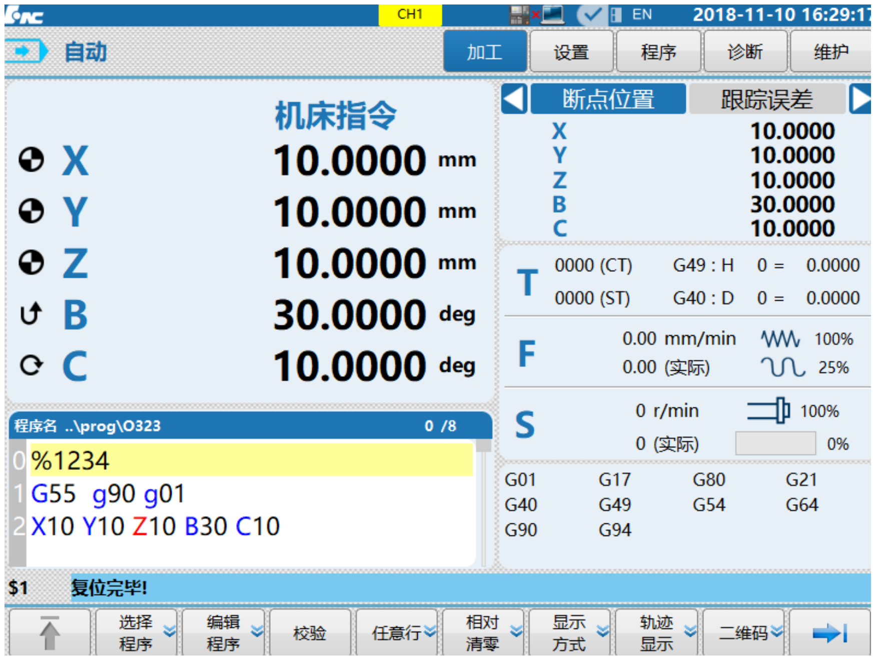
3. After the tool setting of inclined surface is completed (The tool is tilted at this time), current focus is on G54-G59 coordinate system, and click inclined surface tool setting.

4. Enter current tool number and click Confirm, then the vertical tool setting position is automatically input into the current corresponding coordinate system.

Example

Machine Structure: B swivel head C rotary table 5-axis machine

Description: After tool setting, according to the machine structure and the current machine coordinate system position, the swivel head is changed from 30° to 0°, and the translation axis is compensated accordingly, and the coordinates are set as the workpiece coordinate system. The specific process is shown in the figure below.



Tool setting test interface of incline surface

1. Turn on 5-axis function

Parameter setting

2. Click Set -> coordinate system, and enter current tool number after focus moves to the coordinate system to be set.

Tool setting of inclined surface

3. Afterward, G55 is updated.

4.7. Zero Position Function in Full-closed Loop

In full-closed loop, incremental grating ruler or distance-coded grating ruler is generally used for rotary axis. Reference point return is needed after starting up. During reference point return, Occasionally, reading head of grating ruler will cause the zero point offset due to pollution or signal interference.

With the new function, system automatically remembers machine actual position when power failure occurs, and determines whether deviation of reference point return will happen by comparing reference point return position after power up.

5. Related Functions of Lathe Mill Combo

5.1. Related functions of lathe mill combo

For the lathe mill function, the related system parameter needs to be modified, and the specific setting is as following,

Parameter type	Parameter		Value	Description
NC parameter	000065	Lathe tool diameter display Enable	1	X axis diameter display
Machine user parameter	010001	Channel 0 cutting type	2	Lathe-mill combo system
Channel parameter (channel 0)	040032	Diameter programming Enable	1	X axis diameter programming mode is enabled
Channel parameter (channel 0)	040094	Machine type of power failure	0	After applying power to machine, the working mode before power failure is automatically selected
Channel parameter (channel 0)	040101	Number of tools on spindle	2	--

5.1.1. Lathe Milling Programming Switching Command

LATHE: Lathe mode

MILL: Milling mode

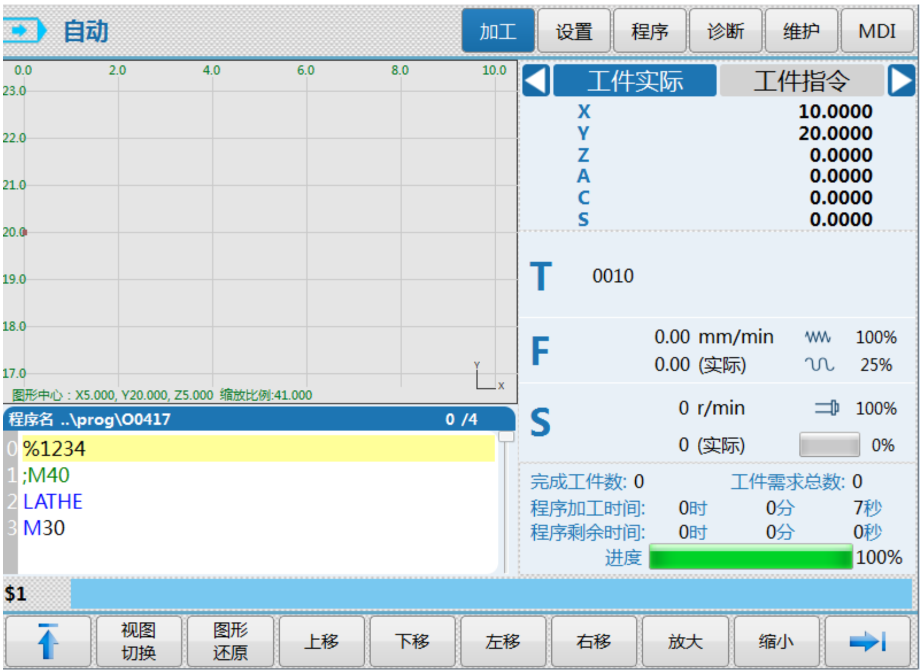
5.1.2. Visual interface of lathe milling switching

The upper bar shows that the current channel is in (turning/milling).

Channel status display

5.1.3. Graphics simulation for lathe milling combo

In Machining menu, click Path display to enter the graphics simulation interface.



Path display

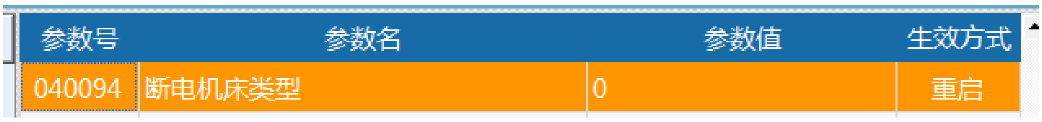
5.1.4. Working mode selection of machine

The channel parameter 040094 Machine type

0: Automatically selected as the working mode (lathe or milling) before power failure after machine is powered on.

1: Current working mode is milling after machine is powered on.

2: Current working mode is lathe after machine is powered on.



参数号	参数名	参数值	生效方式
040094	断电机床类型	0	重启

Working mode setting

5.1.5. Synthesis Tool Table Management

Tool table management

Tool type is divided into,

No tool (invalid tool), milling tool (valid tool), lathe tool (valid tool)

1. It is not allowed to set other values except for the tool type for the type of no tool

2. When the tool type is set to no tool from valid tool, all information of the tool (length X-tool nose direction column) is cleared, and the tool wear of the tool is also cleared.

3. The menu of milling tool is: current position (corresponding wear is cleared), incremental input, relative actual (corresponding wear is cleared). The invalid menu is not displayed.

4. The menu of general tool is: current position (corresponding wear os cleared), incremental input, relative actual (corresponding wear is cleared). The invalid menu is not displayed.

5. The menu of lathe tool is: precutting diameter (corresponding wear value is cleared), Y axis precutting (corresponding wear value is cleared), precutting length (corresponding wear value is cleared). The invalid menu is not displayed.

6. Clear all: clear all tool information including wear

5.1.6. Number of spindle tools

1. When the channel parameter 040101 *Number of tools on spindle* is set to 1 (if the number is not 2, then it is treated as 1), FST display is same as the lathe, as shown below:

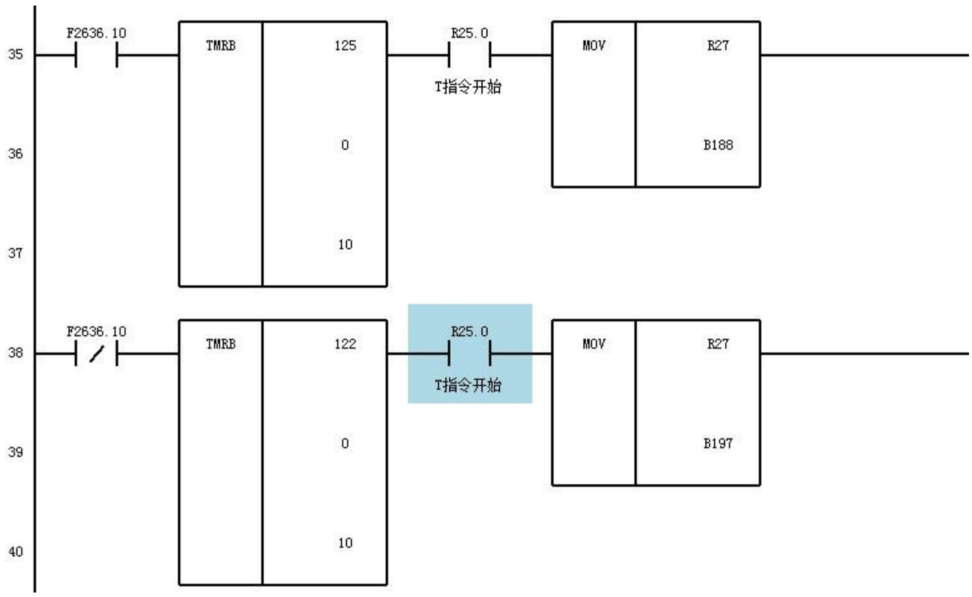
Spindle tool number display

2. When the channel parameter 040101 *Number of tools on spindle* is set to 2, T area display is changed to simultaneous display of current lathe tool and milling tool, as shown below

Number of tools on spindle display

The tool number on spindle is obtained via reading register, and corresponding modification in PLC is needed.

Register	Meaning	
F2636.10	F2636.10=0 channel 0 is in milling mode	F2636.10=1 channel 0 is in lathe mode
B188	Current tool number of spindle in lathe mode	
B197	Current tool number of spindle in milling mode	



PLC modification of spindle tool number

5.1.7. Magazine table display for lathe mill combo

1. When the channel parameter 040101 *Number of tools on spindle* is set to 1 (if the number is not 2, then it is treated as 1), the display is as following:

Tool number display interface for lathe mill combo

Where, tool number and machining mode can be set. When tool number is 0 or tool type is no tool, machining mode is not allowed to be set.

2. When the channel parameter 40101 *Number of tools on spindle* is set to 2, the display is as following:

Tool number display interface for lathe mill combo

Note

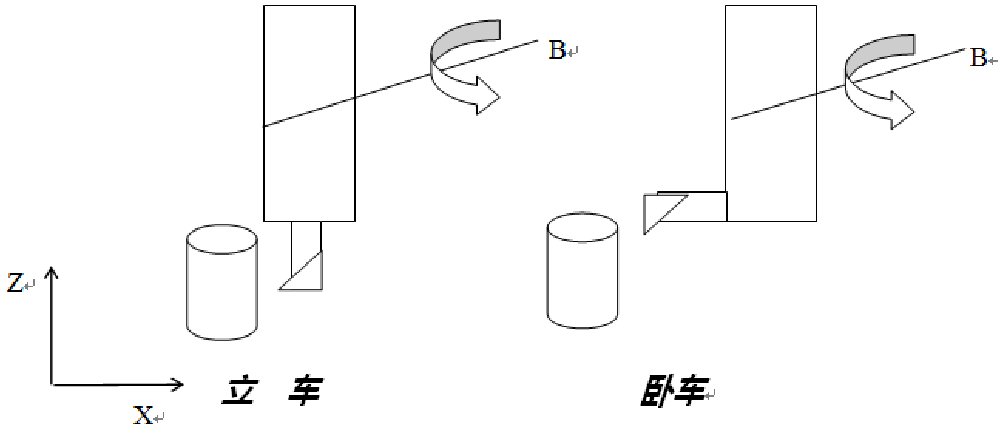
1. The first two lines in magazine table is: current lathe tool information of magazine and current milling tool information of magazine.

2. Tool number and machining mode can be set in the table. When tool number is 0 or tool type is no tool, machining mode is not allowed to be set.

5.1.8. Vertical/Horizontal turning conversion

When the machine is in lathe mode, there are two states: vertical turning and horizontal turning.

For example, 5-axis lathe mill combo with B swivel head and C rotary table, and spindle is installed on B swivel head. When B is 0 degree, it is used as vertical lathe, and when B is 90 degrees, it is used as horizontal lathe.



Vertical/horizontal turning conversion, left: horizontal lath; right: vertical lathe

After vertical/horizontal turning conversion of lathe, the lathe has following features:

1. Programming is still performed in lathe mode, with axial Z axis and radial X axis.

2. Axis exchange is performed with GETD command. X axis movement is actually performed for Z axis programming, and Z axis movement is actually performed for Z axis programming.

3. The movement in MPG and JOG is performed in the converted mode. For example, if X axis is selected, Z axis is the actual moving axis.

4. After vertical/horizontal turning conversion, tool setting, workpiece coordinate system establishment, and tool offset table establishment are performed normally according to lathe usage.

5. After vertical turning is converted to horizontal turning, ensure that the tool nose of the turning tool can coincide. Users only needs to perform tool setting once, and can complete the machining in both vertical and horizontal turning modes.

6. Interface display: coordinate axis of machine coordinate system will be interchanged, and workpiece coordinate system keeps unchanged.

7. If the display is in two columns, and one of them is the machine coordinate system, is is based on the machine coordinate system.

Interface after vertical/horizontal turning conversion

1. Programming is still performed in lathe mode after vertical/horizontal turning conversion

After vertical/horizontal turning conversion, the programming is still performed in lathe mode. Axis exchange is performed with GETD command. X axis movement is actually performed for Z axis programming, and Z axis movement is actually performed for Z axis programming.

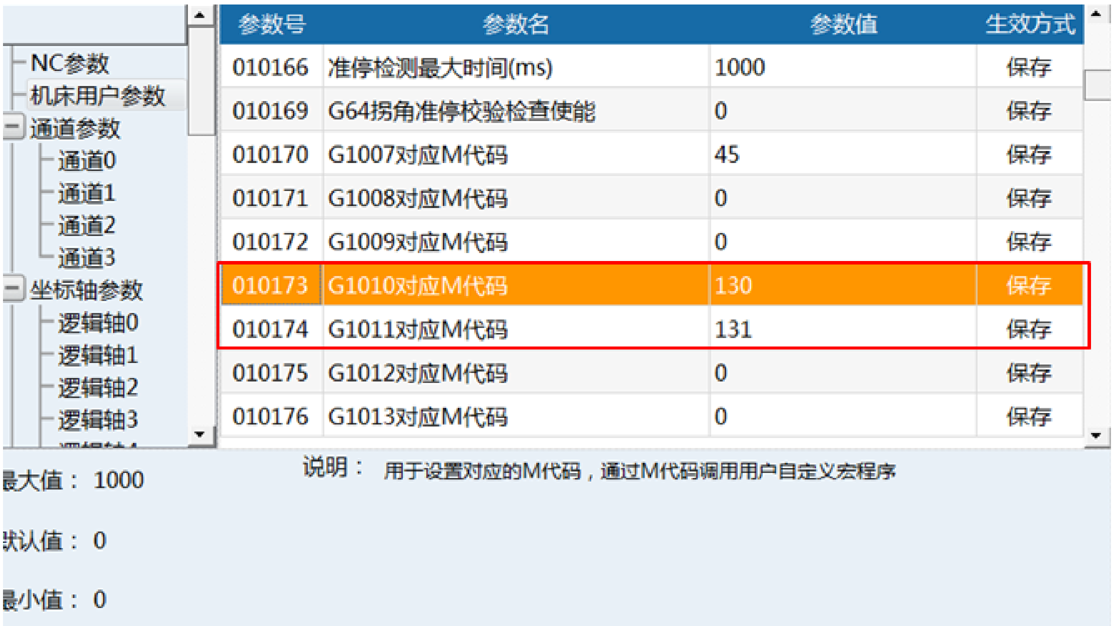
Function configuration

（1）Parameter modification

The channel parameter 040102 Dynamically switching axis mask is set based on the axis to be switched. For example, for switching of XZ axes, 0x1 is set; 0x2 is set for switching of YZ axes.

Parameter setting

（2） When calling a canned cycle with M code, pay attention that the M code is not occupied



参数号	参数名	参数值	生效方式
010166	准停检测最大时间(ms)	1000	保存
010169	G64拐角准停校验检查使能	0	保存
010170	G1007对应M代码	45	保存
010171	G1008对应M代码	0	保存
010172	G1009对应M代码	0	保存
010173	G1010对应M代码	130	保存
010174	G1011对应M代码	131	保存
010175	G1012对应M代码	0	保存
010176	G1013对应M代码	0	保存

Setting of calling canned cycle with M code

（3）Canned cycle file

a) Add G1010 and G1011 canned cycle in the canned cycle file USERDEF.CYC

```
%1010
;车床立卧转换（X轴和Z轴进行互换）
FREE X-1 Z-1
GETD X2 Z0
M49 ;配合PLC, 实现手动和手摇方下, 按转换后的方式运动
M99

%1011
;车床解除立卧转换
FERR X-1 Z-1
GETD X0 Z2
M48 ; 配合PLC, 实现手动和手摇方下, 按转换前的方式运动
M99
```

Canned cycle of vertical/horizontal processing conversion

b) Description

FREE X-1 Z-1

This command is to release axes X and Z

GETD X2　Z0

This command maps logical axis 2 to X axis and logical axis 0 to Z axis

c) Example

%1234

G54

M130；　Vertical/horizontal processing conversion, exchange X axis and Z axis

M131；　Vertical/horizontal processing conversion is disabled

M30

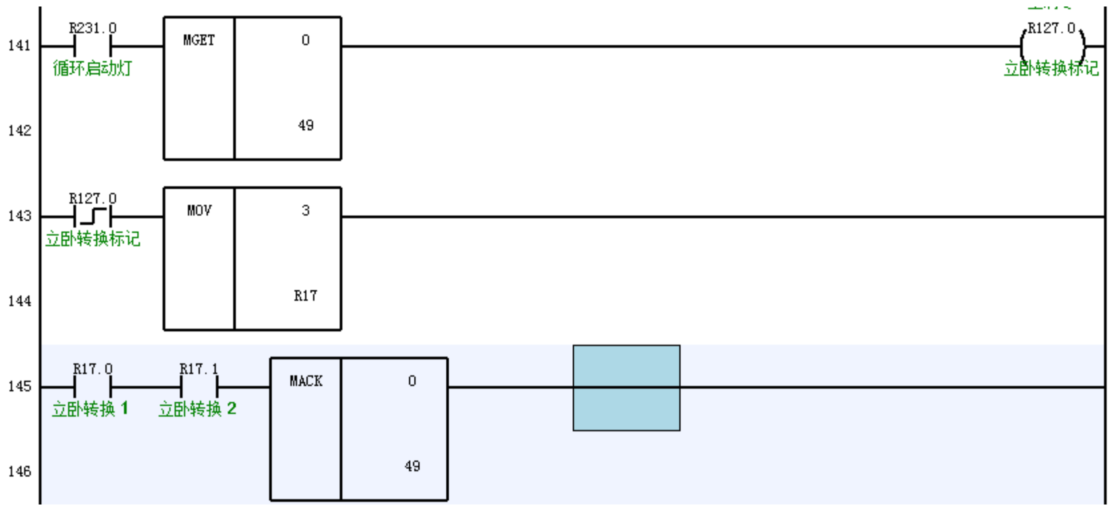
2. Movement in MPG and JOG is performed in the converted mode after vertical/horizontal processing conversion

After vertical/horizontal processing conversion, the movement in MPG and JOG is performed in the converted mode. For example, if X axis is selected, the actual moving axis is Z axis. Tool setting, workpiece coordinate system establishing, and tool offset table establishment are performed according to lathe usage.

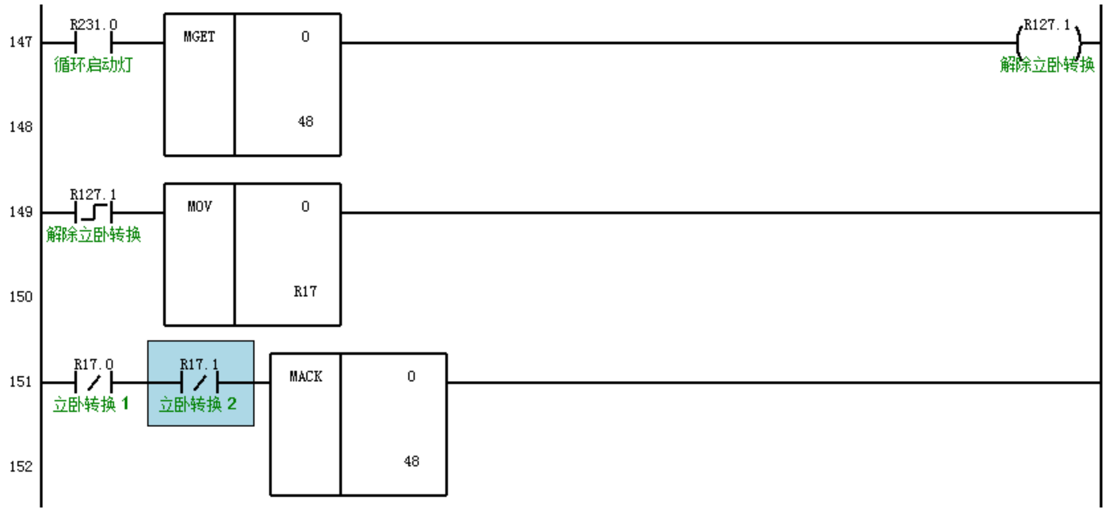
Example

Take the 818D panel as an example.

（1）Add M48 in PLC to disable vertical/horizontal processing conversion, and M99 to perform vertical/horizontal processing conversion.



PLC modification example

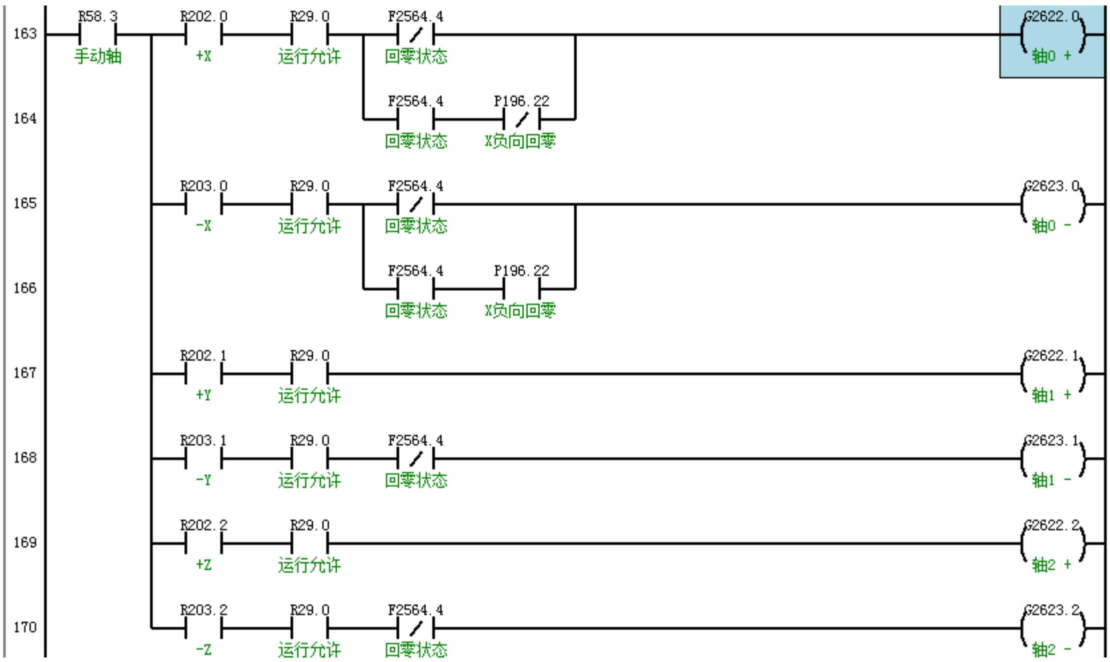


PLC modification example

（2）Modification of situation in JOG

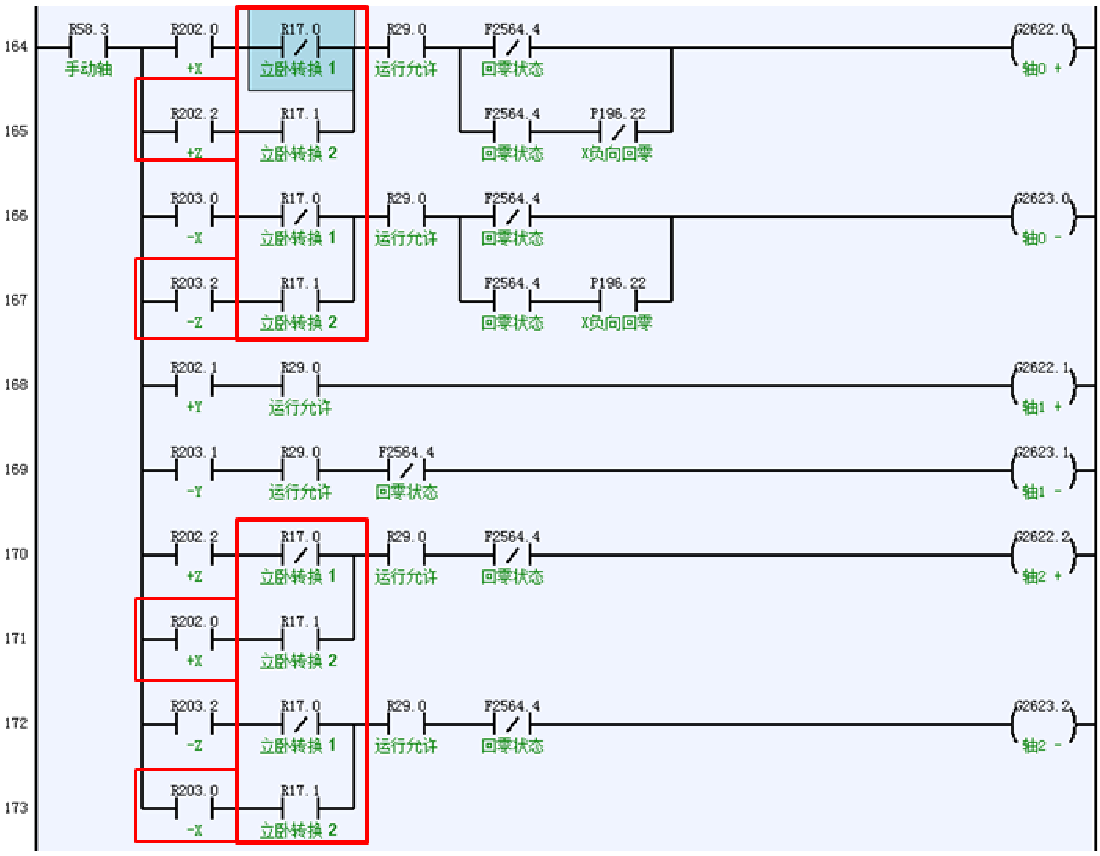
Look for G2622.0, and change conversion conditions

The PLC before modification is as follows



PLC modification example

The PLC after modification is as follows

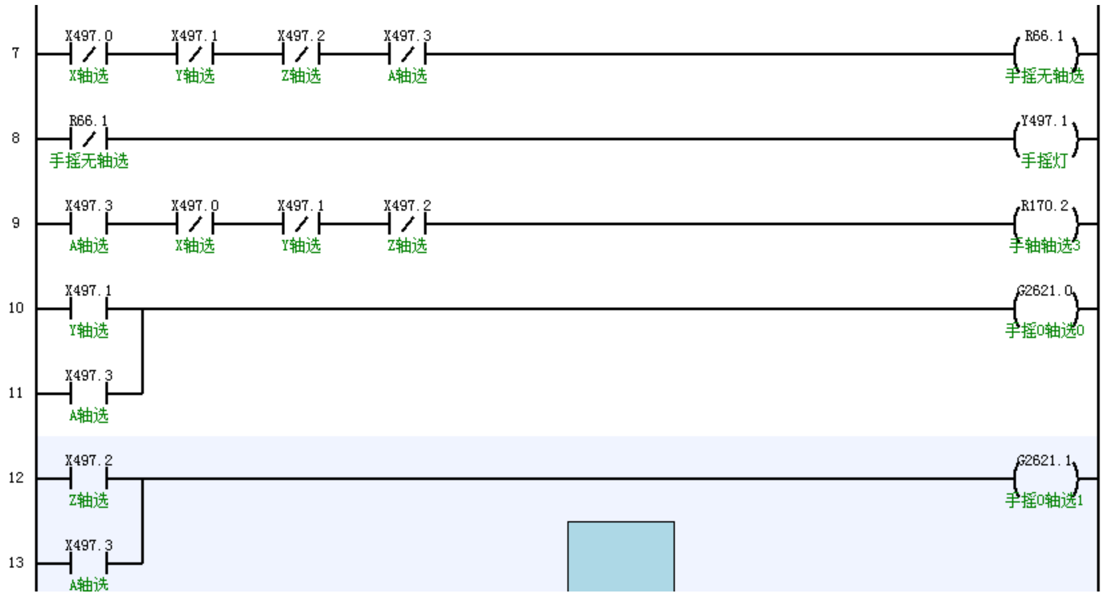


PLC modification example

（3）Modification of situation in MPG

Look for G2621.0

The PLC before modification is as follows



PLC modification example

The PLC after modification is as follows

PLC modification example

3. Tool nose coincides after vertical/horizontal processing conversion

After vertical turning is converted to horizontal turning, ensure that the tool nose of the turning tool can coincide. Users only needs to perform tool setting once, and can complete the machining in both vertical and horizontal turning modes.

Function configuration

（1）Parameter modification. Use M code to call the canned cycle to ensure that the M code is not occupied

M code setting for vertical/horizontal turning conversion

（2）USERDEF.CYC Canned cycle file USERDEF.CYC

Add G1012 and G1013 user canned cycle. If the canned cycle number is occupied, other ones can be used.

G1012 canned cycle is as follows

```
%1012
IF [AR[#7] EQ 0]    ; H is not defined
G110
ENDIF
IF #7 LT 0     ; H is smaller than 0
G110
ENDIF
G10 L43 P[#7] X0Y0Z0A0C0 R100
G10 L43 P[#7] X0Y0Z0A90C0 R105
#110 = #105-#100; X offset
#111 = #106-#101; Y offset
#112 = #107-#102; Z offset
G52 X[#110] Y[#111] Z[#112]
G80
M99
```

G1013 canned cycle is as follows

%1013

G52 X0Y0Z0

G80

M99

Description

G10 L43 P_ X_ Y_ Z_ A_ B_C_ R_

The command calculates machine coordinates in RTCP by the given workpiece coordinates, and writes it into system variable.

Example

%1234

G54

M132 H1; In the horizontal processing state, the current tool number is 1, call G1001 to complete the vertical/horizontal processing conversion and tool nose coincidence

M133; Revert to the coordinate system in vertical processing state

M30

Note

(1) With this function used in canned cycle, machine coordinates of tool nose in vertical processing and horizontal processing are calculated respectively by the command G10L43, and the tool nose offset is obtained and put into G53 local coordinate system. When executing canned cycle without performing RTCP simultaneous motion, if the follow-up command executes displacement movement, the system will compensate the tool nose offset to ensure that turning can be performed without re-calibrating the tool.

(2) Tool compensation setting of lathe tool. Channel parameter 040409 Bidirectional tool compensation of 90°head is set as 1. Fill the tool compensation value of lathe tool into the tool compensation table: length X and length Z

5.1.9. Dual channel lathe mill combo

1. Parameter configuration

Parameter type of channel 0 and channel 1 are set as the lathe mill combo type. Channel 0 is switched to milling mode, and channel 1 is switched to lathe mode. The specific parameters are set as follows.

Parameter type	Parameter		Value	description
NC parameter	000065	Lathe tool diameter display enable	1	X axis diameter display
Machine user parameter	010001	Channel 0 cutting type	2	Lathe mill combo system
Machine user parameter	010002	Channel 1 cutting type	2	Lathe mill combo system
Channel parameter (channel 0)	040032	Diameter programming enable	1	X axis diameter programming mode is enabled
Channel parameter (channel 0)	040094	Power failure machine type	1	After machine is powered on, current working mode is milling
Channel parameter (channel 0)	040101	Number of tools on spindle	1	--
Channel parameter (channel 1)	041032	Diameter programming enable	1	X axis diameter programming mode is enabled
Channel parameter (channel 1)	041094	Power failure machine type	2	After machine is powered on, current working mode is lathe
Channel parameter (channel 1)	041101	Number of tools on spindle	1	--

2. Multi-magazine management

（1） Channel 0 is in milling mode, and the type of all tools in magazine is milling tool; channel 1 is in lathe mode, and the type of all tools in magazine is lathe tool;

(2) Current tool number of channel 0 is obtained by the register B188; current tool number of channel 1 is obtained by the register B204

Parameter type	**Parameter**		**Value**	**Description**
Channel parameter (channel 0)	040125	Initial magazine number	1	--
	040126	Number of magazines	1	--
	040127	Initial tool number	1	--
	040128	Number of tools	24	The parameter is set based on the actual situation
Channel parameter (channel 1)	041125	Initial magazine number	2	--
	041126	Number of magazines	1	--
	041127	Initial tool number	40	The parameter is set based on actual situation, and must be greater than the last tool number in channel 0
	041128	Number of tools	24	The parameter is set based on actual situation

(3) Magazine data in system parameter should be modified accordingly

After the setting is successful, the tool compensation interface of channel 0 and channel 1 is displayed as follows

Milling channel (channel 0) Lathe channel (channel 1)

Tool compensation interface display

6. 5-Axis Parameters

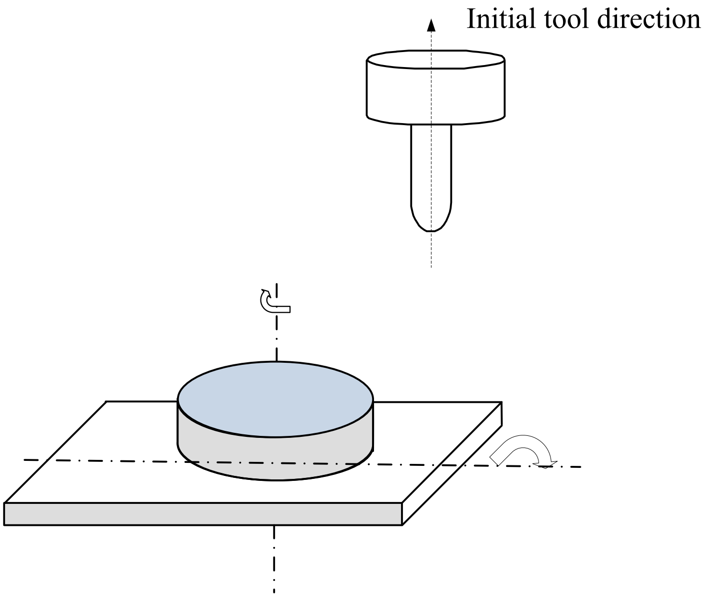
6.1. Initial Tool direction

Parameter No.	040400 to 040402
Name	Initial tool direction
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the initial direction of tool. Generally, the initial tool direction is parallel to Z axis

Illustration



Example

Generally, the initial tool direction is set as shown below,

- Parameter 040400 *Initial tool direction (X)* is set to 0.0
- Parameter 040401 *Initial tool direction (Y)* is set to 0.0
- Parameter 040402 *Initial tool direction (Z)* is set to 1.0

6.2. RTCP Tool Setting Mode

Parameter No.	040403
Name	RTCP tool setting mode
Data unit	
Data type	INT
Value	0 to 1
Default	0
Access level	Machine manufacturer

Activation	Activation_Reset

Description

System provides two tool setting modes for users: tool setting of spindle end face and tool setting of tool nose. The first mode is the default.

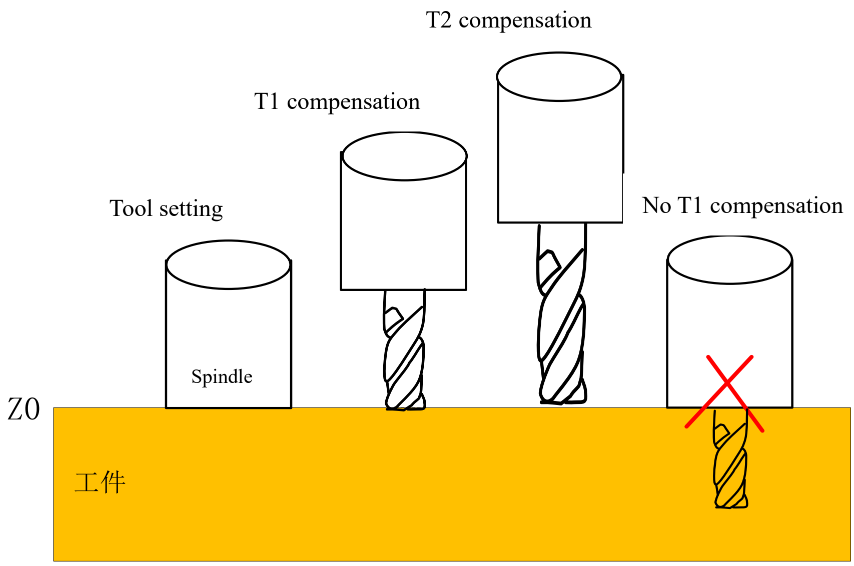
➢ Tool setting of spindle end face

The setting of 0 indicates the tool setting of spindle end face. After tool setting is completed on Z, it is necessary to offset a tool length in the negative direction, and set the spindle end face as the workpiece coordinate origin. In this tool setting mode, there is no need to perform tool setting again after tool change, and the previously set coordinate system can continue to be used.

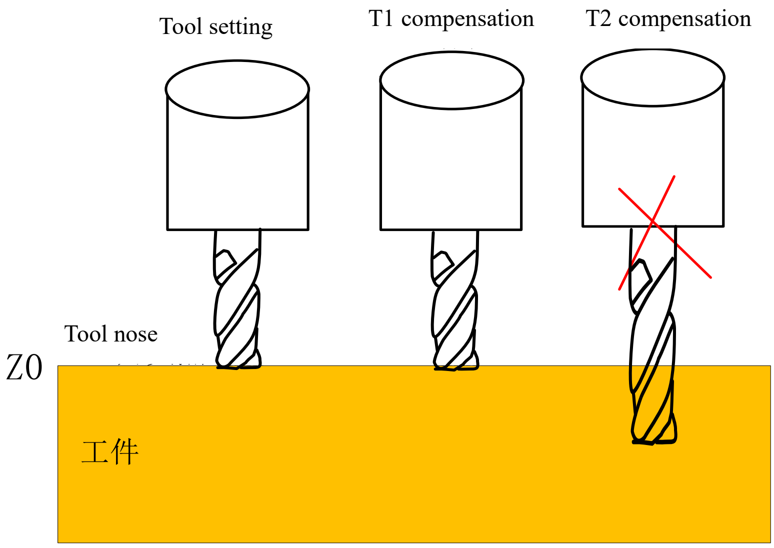
➢ Tool setting of tool nose

The setting of 1 indicates the tool setting of tool nose. In this mode, the tool nose is set as the workpiece coordinate origin after tool setting on Z is completed.

Illustration



(a) Tool setting spindle end face



(b)　Tool setting of tool nose

Note

- During tool setting of spindle end face, if RTCP mode is not enabled, when Z axis moves to the area near workpiece origin, the interference between tool and workpiece may occur.
- During tool setting of tool nose, tool setting must be performed again after tool change, and new workpiece origin is set.

6.3. W Axis Compensation

Parameter No.	040404
Name	W axis compensation
Data unit	
Data type	INT
Value	0~1
Default	0
Access level	Machine manufacturer

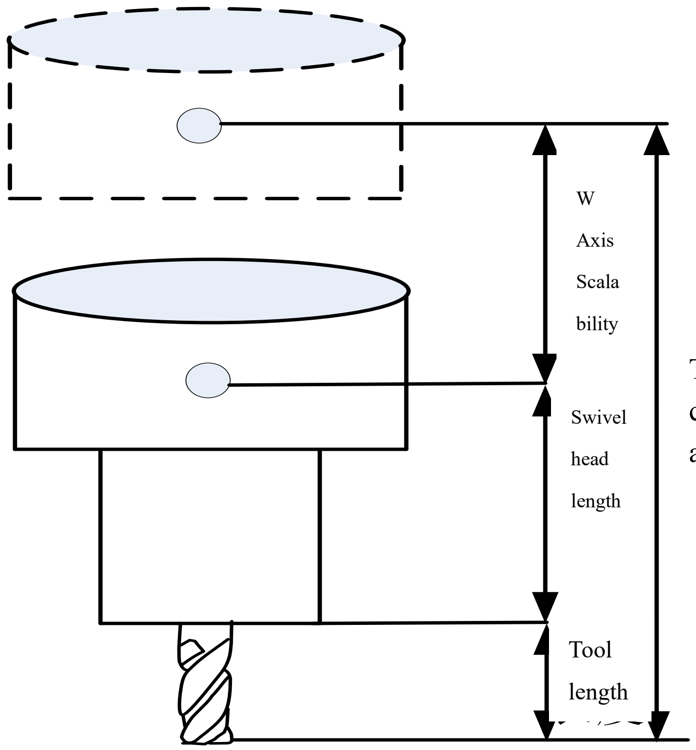
Activation	Activation_Reset

Description

Install W axis on Z axis of 5-axis machine, and use scalability of W axis to compensate the travel on Z and enlarge the machining range. After movement on W axis occurs, the tool center also changes. System provides real-time compensation of W axis to ensure tool moves based on workpiece position.

When 1 is set, W axis compensation function is turned on. Turning off is the default.

Illustration



Note

- In order to effectively use W axis compensation function, it is necessary to ensure that the W axis is at the machine origin during calibration of the structural parameter.
- When setting workpiece coordinate origin, W axis is at the machine origin.

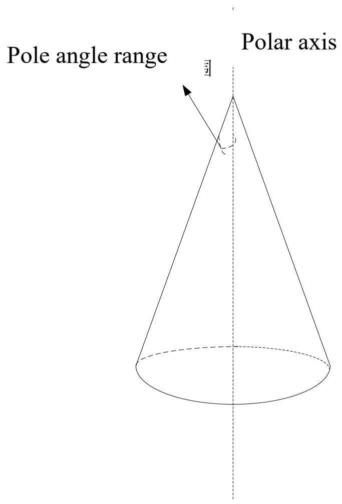
6.4. Range of Pole Angle

Parameter No.	040407
Name	Range of pole angle
Data unit	
Data type	REAL
Value	0.0 to 360.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The pole area is defined by the angle, that is, a conical area with the pole axis as the axis line and the angle as the cone angle. The pole range is with in this area. When the tool passes near the pole, due to the uncertainty of the direction of rotary axis, if there is no corresponding handling, it will cause the rotary axis to overspeed.

Illustration



6.5. Swivel Head Indexing

Parameter No.	040408
Name	Swivel head indexing
Data unit	
Data type	INT4
Value	0 to 1
Default	0
Access level	Machine manufacturer
Activation	Activation_Reset

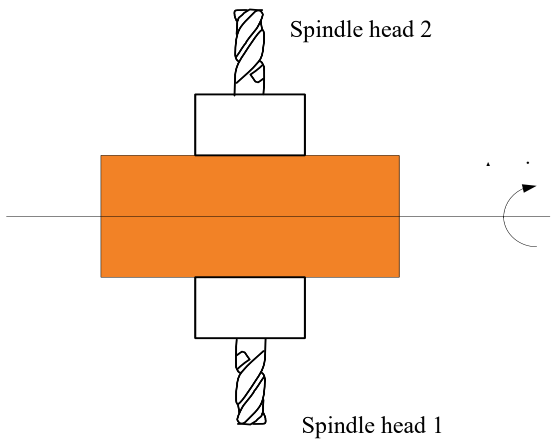
Description

After swivel head indexing function is turned on, the RTCP compensation position has nothing to do with the workpiece origin of rotary axis. During multi-spindle head RTCP swivel machining, when the spindle head needs to be switched, there is no need to consider the current angular position of rotary axis of the spindle head, and just create the tool path program directly in the workpiece coordinate system

Swivel head indexing:

- When 0 is set, swivel head indexing is turned off (default). During RTCP compensation, Calculation angle of rotary axis = Workpiece coordinate of rotary axis + Workpiece origin of rotary axis
- When 1 is set, swivel head indexing is turned on. During RTCP compensation, Calculation angle of rotary axis = Workpiece coordinate of rotary axis

Illustration



Example

%0001

G43.4H1

G54 (Spindle head 1, A axis coordinate system 0 degree)

M98P1002

G49

G55 (Spindle head 2, A axis coordinate system 180 degrees)

G43.4H1

M98P1002

G49

M30

%1002

G01 X30Y30Z30A30

G01 X0Y0Z0A0

M99

6.6. 90° Head Bidirectional Tool Length Compensation

Parameter No.	040409
Name	90°head bidirectional tool length compensation
Data unit	

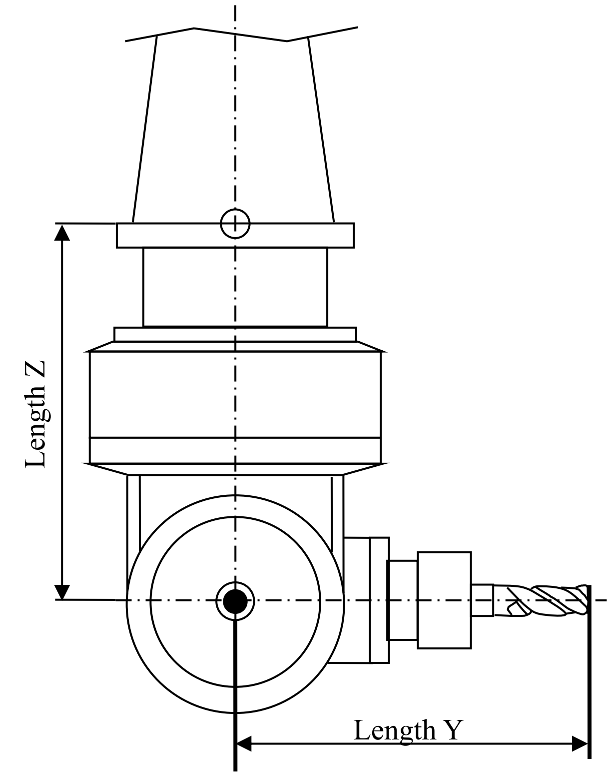
Data type	INT4
Value	0 to 1
Default	0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The 90°angle milling head is an important spindle accessory. It has two right-angled sides forming L-shaped structure. The 5-axis machine's processing technology is enriched and it processing range is expanded by working with the 90°angle head, and the conversion between vertical and horizontal processing is easily achieved. The tool length of the ordinary milling head is only along the spindle direction, and the tool length compensation only needs to be performed on Z axis; while the 90° angle milling head contains two directions for tool length, and bidirectional tool length compensation is required for the 90°angle head.

Set the channel parameter 040409 Bidirectional tool length compensation of 90°angle head to 1 to enable the bidirectional tool length compensation. At this point, the length compensation X and Y values are effective, and system will perform the compensation based on the values.

Illustration



Note

(1) The initial tool direction must be set to (0,0,1) for this function. Channel parameters 040400 to 040402 indicate the initial tool direction.

(2) In tool compensation table, any of X, Y, and Z can set geometric amount, wear amount, and matrix amount.

(3) Generally, the length on two directions of L-shaped 90° angle head need to be set, and the signs are determined based on the direction that tool nose points to spindle center.

6.7. Swivel Head Structure Type

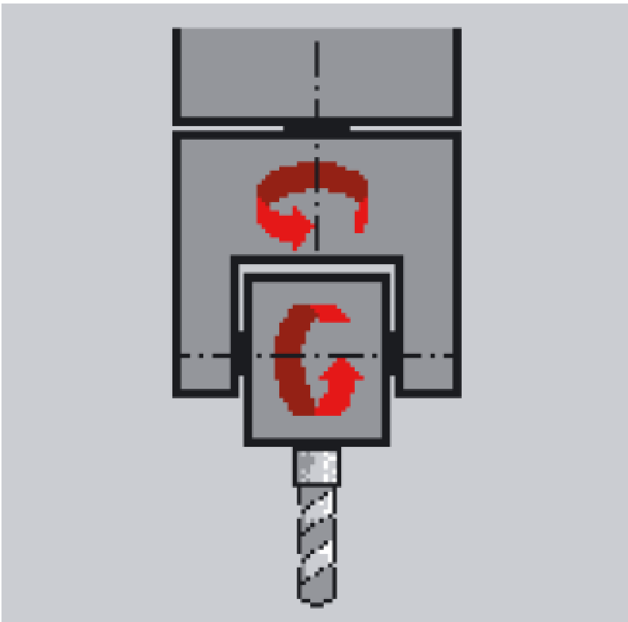
Parameter No.	040410
Name	Swivel head structure type
Data unit	
Data type	STRING[7]
Value	

Default	Z
Access level	CNC manufacturer
Activation	Activation_Reset

Description

The parameter setting works on the rotary axis name of swivel head, and master axis is before slave axis.

Illustration



Dual rotary structure

Example

For the machine tool with CA dual rotary head, if master axis is C axis and slave axis is A axis, then the swivel head structure type is CA.

6.8. Direction Vector of 1st Rotary axis of Swivel Head

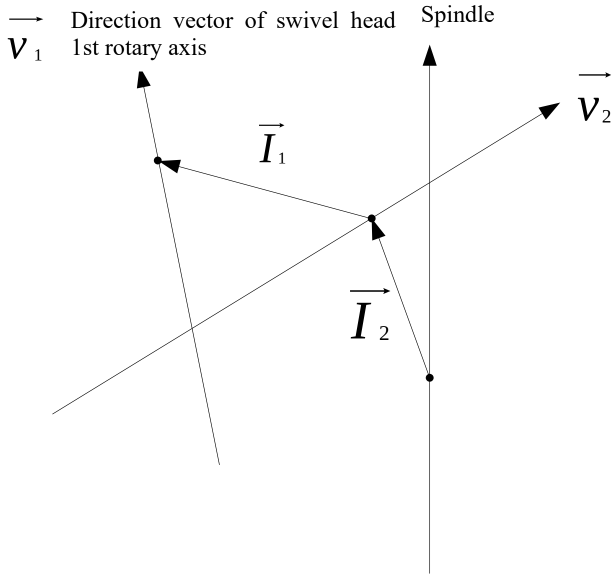
Parameter No.	040411 to 040413
Name	Direction vector of 1st rotary axis of swivel head

Data unit	
Data type	REAL
Value	-21470.0 to 21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the direction vector of the first rotary axis for the swivel head (master axis), and supports any axis direction.

Illustration



Example

For the 5-axis machine tool with CA dual rotary head structure, if the master axis is C axis, then the direction vector of the first rotary axis for the swivel head is set as following,

➢ Parm40411 *Direction vector (X) of 1st rotary axis of swivel head* is set to 0

- Parm40412 *Direction vector (Y) of 1st rotary axis of swivel head* is set to 0

Parm40413 *Direction vector (Z) of 1st rotary axis of swivel head* is set to 1

Example

For the 5-axis machine tool with CA dual rotary head structure, if the master axis is C axis, then the direction vector of the first rotary axis for the swivel head is set as following,

- Parm40411 *Direction vector (X) of 1st rotary axis of swivel head* is set to 0
- Parm40412 *Direction vector (Y) of 1st rotary axis of swivel head* is set to 0
- Parm40413 *Direction vector (Z) of 1st rotary axis of swivel head* is set to 1

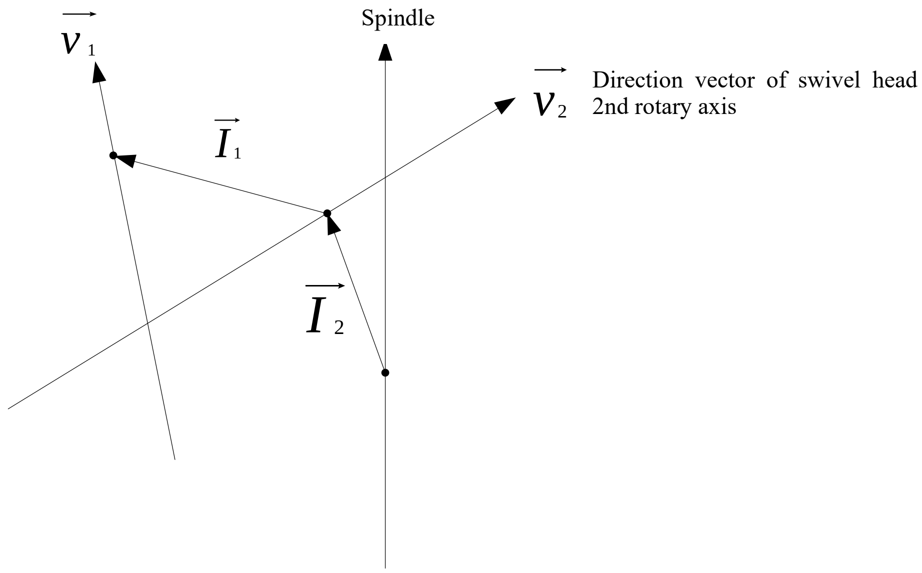
6.9. Direction Vector of 2nd Rotary Axis of Swivel Head

Parameter No.	040414 to 040416
Name	Direction vector of 2nd rotary axis of swivel head
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the direction vector of the first rotary axis for the swivel head (slave axis), and supports any axis direction.

Illustration



Example

For the 5-axis machine tool with CA dual rotary head structure, if the slave axis is A axis, then the direction vector of the second rotary axis for the swivel head is set as following,

- Parm40414 *Directon vector (X) of 2nd rotary axis of swivel head* is set to 1
- Parm40415 *Directon vector (Y) of 2nd rotary axis of swivel head* is set to 0
- Parm40416 *Directon vector (Z) of 2nd rotary axis of swivel head* is set to 0

6.10. Offset Vector of 1st Rotary axis of Swivel Head

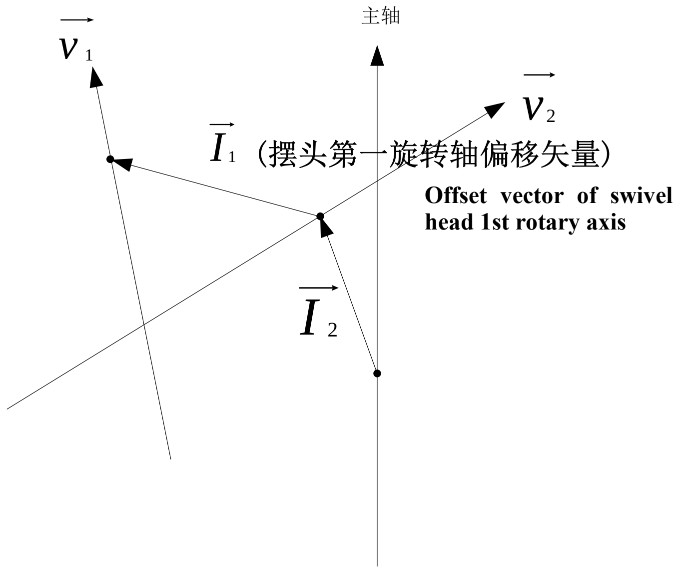
Parameter No.	040417 to 040419
Name	Offset vector of 1st rotary axis of swivel head
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer

Activation	Activation_Reset

Description

This parameter sets the offset vector of the first rotary axis of the swivel head, that is, the offset vector of the master axis relative to the slave axis.

Illustration



Example

For the 5-axis machine tool with CA dual swivel head, if the offset vector of master axis C axis relative to slave axis A axis is (0,-10,0), the direction vector of 2nd rotary axis of swivel head is set as following.

- Parm40417 *Offset vector (X) of 1st rotary axis of swivel head* is set to 0
- Parm40418 *Offset vector (Y) of 1st rotary axis of swivel head* is set to -10
- Parm40419 *Offset vector (Z) of 1st rotary axis of swivel head* is set to 0

6.11. Offset Vector of 2nd Rotary Axis of Swivel Head

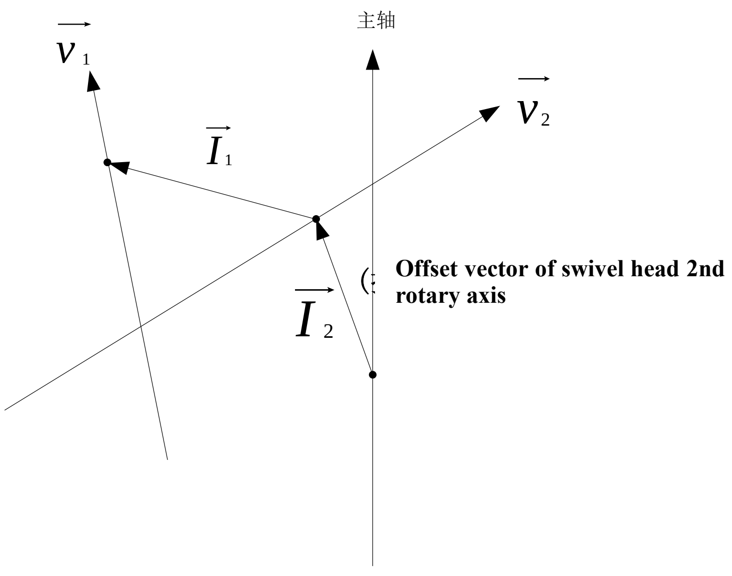
Parameter No.	040420 to 040422
Name	Offset vector of 2nd rotary axis of swivel head
Data unit	

Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the offset vector of the second rotary axis for the swivel head, that is, the offset vector of slave axis relative to the center of spindle end face (control point).

Illustration



Example

For the 5-axis machine with CA dual rotary head, if the offset vector of slave axis A axis relative to spindle end face center (control point) is (-10, -10, 80), the offset vector of the second rotary axis for the swivel head is set as following,

- Parm40420 *Offset vector (X) of 2nd rotary axis of swivel head* is set to -10
- Parm40421 *Offset vector (Y) of 2nd rotary axis of swivel head* is set to -10

➢ Parm40422 *Offset vector (Z) of 2nd rotary axis of swivel head* is set to 80

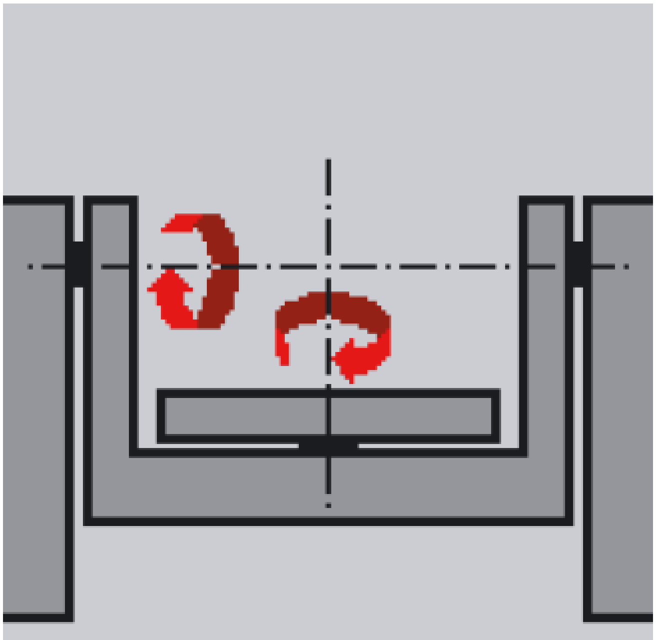
6.12. Rotary Table Structure Type

Parameter No.	040425
Name	Rotary table structure type
Data unit	
Data type	STRING[7]
Value	
Default	YX
Access level	CNC manufacturer
Activation	Activation_Reset

Description

The parameter is to set the name of rotary axis for rotary table, and master axis is before slave axis.

Illustration



Example

The 5-axis machine tool with AC dual rotary table, if master axis is A axis and slave axis is C axis, the rotary table structure type is AC.

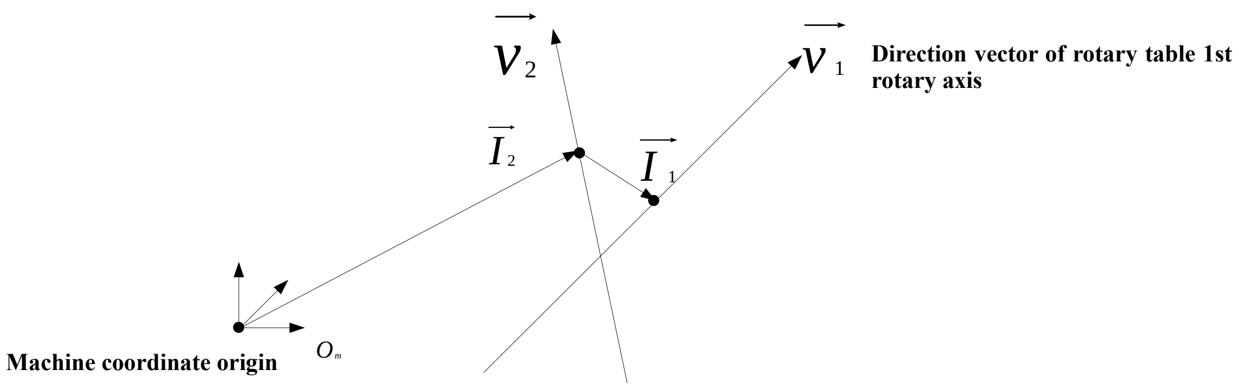
6.13. Direction Vector of 1st Rotary axis of Rotary Table

Parameter No.	040426 to 040428
Name	Direction vector of 1st rotary axis of rotary table
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the direction of master axis for rotary table. The direction is opposite to swivel head direction. It supports direction of any axis.

Illustration



Example

For the 5-axis machine with AC dual rotary table, if the master axis is A axis, the direction vector of 1st rotary axis of rotary table is set as following,

- Parm40426 *Direction vector (X) of 1st rotary axis of rotary table* is set to -1
- Parm40427 *Direction vector (Y) of 1st rotary axis of rotary table* is set to 0
- Parm40428 *Direction vector (Z) of 1st rotary axis of rotary table* is set to 0

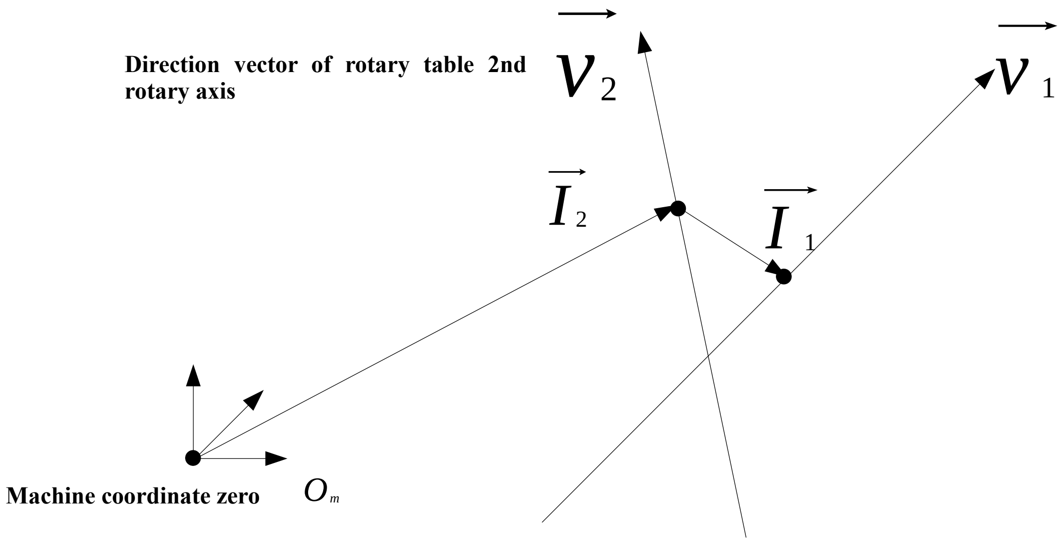
6.14. Direction Vector of 2nd Rotary Axis of Rotary Table

Parameter No.	040429 to 040431
Name	Direction vector of 2nd rotary axis of rotary table
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

This parameter is to set the direction of slave axis for rotary table. The direction is opposite to direction of swivel head. It supports direction of any axis.

Illustration



Example

For the 5-axis machine tool with AC dual rotary table, if slave axis is C axis, direction vector of 2nd rotary axis of rotary table is set as below.

- Parm40429 *Direction vector (X) of 2nd rotary axis of rotary table* is set to 0
- Parm40430 *Direction vector (Y) of 2nd rotary axis of rotary table* is set to 0
- Parm40431 *Direction vector (Z) of 2nd rotary axis of rotary table* is set to -1

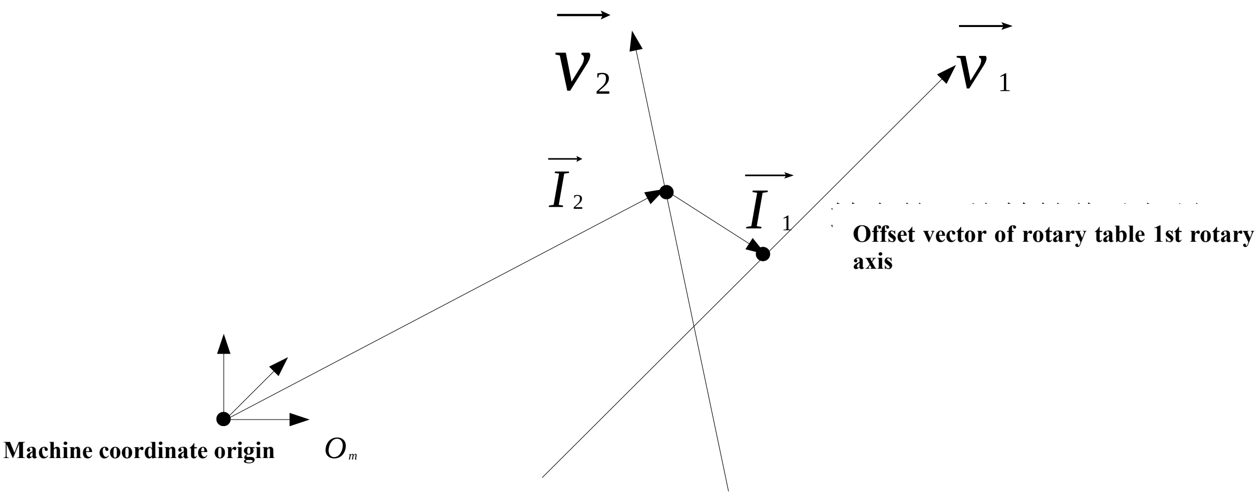
6.15. Offset Vector of 1st Rotary Axis of Rotary Table

Parameter No.	040432 to 040434
Name	Offset vector of 1st rotary axis of rotary table
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0
Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the offset vector of the first axis for rotary table, that is, the offset vector of master axis relative to slave axis.

Illustration



Example

For the 5-axis machine with AC dual rotay table, if the offset vector of master axis A axis relative to slave axis C axis is (0,10,0), then

- Parm40432 *Offset vector (X) of 1st rotary axis of rotary table* is set to 0
- Parm40433 *Offset vector (Y) of 1st rotary axis of rotary table* is set to 10
- Parm40434 *Offset vector (Z) of 1st rotary axis of rotary table* is set to 0

6.16. Offset Vector of 2nd Rotary axis of Rotary Table

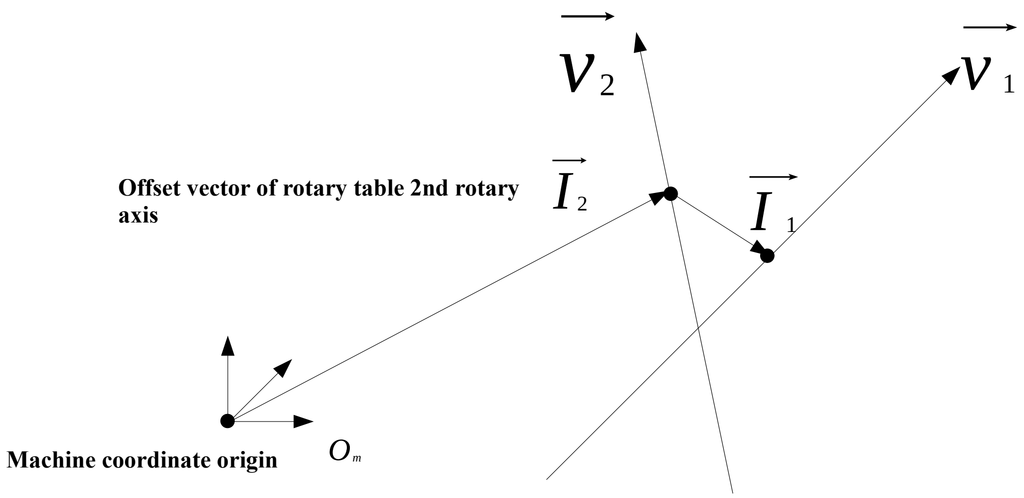
Parameter No.	040435 to 040437
Name	Offset vector of 2nd rotary axis of rotary table
Data unit	
Data type	REAL
Value	-21470.0 to -21470.0

Default	0.0
Access level	Machine manufacturer
Activation	Activation_Reset

Description

The parameter is to set the offset vector of 2nd rotary axis of rotary table, that is, the offset vector of slave axis relative to machine zero.

Illustration



Example

For the machine tool with AC dual rotary table, if the offset vector of slave axis C axis relative to machine zero is (-20,-40,-150), then

- Parm40435 *Offset vector (X) of 2nd rotary axis of rotary table* is set to -20
- Parm40436 *Offset vector (Y) of 2nd rotary axis of rotary table* is set to -40
- Parm40437 *Offset vector (Z) of 2nd rotary axis of rotary table* is set to -150

7. PLC for 5-Axis

Users can easily modify PLC based on difference 5-axis machine structure, which lowers the commissioning of 5-axis machine tool.

For HNC8 5-axis PLC, the following contents are added to the standard milling machine PLC.

- Separate control subprogram of two rotary axes;
- Some alarm and protection functions;
- Manual normal retract module

The added PLC contents are illustrated below

7.1. Added IO Comparison Table (System Panel)

索引	寄存器(I/Q)	IO点(X/Y)	电平	周期	符号名	注释
1	I100.0	X481.0	0	1	自动	
2	I100.1	X481.1	0	1	单段	
3	I100.2	X480.0	0	1	手动	
4	I100.3	X480.1	0	1	手轮	
5	I100.4	X480.2	0	1	回参考点	
6	I100.5	X483.4	0	1	刀具松/紧	
7	I100.6		0	1		
8	I100.7	X480.6	0	1	空运行	
9	I101.0	X480.7	0	1	程序跳段	
10	I101.1	X481.6	0	1	选择停	
11	I101.2		0	1		
12	I101.3	X481.7	0	1	机床锁住	
13	I101.4	X482.6	0	1	防护门	
14	I101.5	X484.7	0	1	机床照明	

IO comparison table of system panel

In IO comparison table of system panel, the annotation information of the register (I/Q) and the logic position in the ladder diagram have been defined, and there is no need to modify it. X input point corresponds to I register, and Y output point corresponds to Q register according to point of each function button on the MCP panel (such as auto, single block, jog, etc.). The IQ register address of the system ranges from 80 to 127.

The mapping between IQ and XY can be separately generated as an IO comparison table (system panel) (*.IOTS), which is output or loaded into a PLC file by the PLC editing software HNCLadder, such as IO comparison table of 808D milling panel. IOTS, 818D standard milling panel without MPG. IOTS, etc.

7.2. Added IO Comparison Table (User)

In IO comparison table of user, the annotation information of the register (I/Q) and the logic position in the ladder diagram have been defined, and there is no need to modify it. X input point corresponds to I register, and Y output point corresponds to Q register according to the definition of field IO device.

The IQ register address of the system ranges from 0 to 79.

The mapping between IQ and XY can be separately generated as an IO comparison table (user input&output) (*.IOTU), which is output or loaded into a PLC file by HNCLadder. The IO comparison table (user input&output) can be saved as a dedicated IO comparison table for a certain user.

Note: In the V2.4 version, the I and Q registers defined in the two IO comparison tables support up to 256 groups. If it is exceeded, the startup fails when loading the PLC to the system.

索引	寄存器(I/Q)	IO点(X/Y)	电平	周期	符号名	注释
1	I0.0		0	0	紧刀到位	
2	I0.1		0	0	松刀到位	
3	I0.2		0	0	扣刀	
4	I0.3		0	0	原点	
5	I0.4		0	0	刹车	
6	I0.5		0	0		
7	I0.6		0	0	刀库计数	
8	I0.7		0	0	刀库零位	
9	I1.0		0	0	前进/到刀	
10	I1.1		0	0	后退/回刀	
11	I1.2		0	0		
12	I1.3		0	0	润滑报警	
13	I1.4		0	0		
14	I1.5		0	0	气压报警	

$1EMG

User IO comparison table

7.3. Added K Parameter Switch

地址	注释	7	6	5	4	3	2	1	0
K0	面板类型	0	0	0	0	0	0	0	1
K1	刀库类型	0	0	0	0	0	0	0	1
K2	刀库调试1	0	0	0	0	0	0	0	0
K3	刀库调试2	0	0	0	0	0	0	0	0
K4	主轴功能	0	0	0	0	0	0	0	0
K5	轴移动选择	0	0	0	0	0	0	0	0
K6	进给轴	0	0	0	0	0	0	0	0
K7	手摇功能	0	0	0	0	1	0	1	0
K8	润滑功能	0	0	0	0	0	0	0	0

308D面板(K0.0)

808D面板

Added K parameter switch

K parameter switch is used to replace PLC switch. Each group of K parameter has point 1 to 7, a total of 8 points. 16 groups have 128 points in total. It is divided into following groups based on functions of K parameter: K0 group: panel type; K1 group: magazine type; K2, K3 groups: magazine debugging; K4 group: spindle function, etc. The K parameter planning in the new version of PLC is as follows:

Group	A address	Symbol name	Annotation
Panel type	K0.0	808D panel	Standard 808D milling MCP panel
	K0.1	818D standard w/o MPG	Standard 818D milling MCP panel without MPG
	K0.2	848D (reserved)	5-axis 848D panel
	K0.3	818D drilling tapping w/o MPG	818D drilling tapping MCP panel without MPG
	K0.4	818D 6-axis w/o MPG	818D 6-axis drilling tapping MCP panel without MPG
	K0.5	818D with MPG	818D drilling tapping MCP panel with MPG
Magazine type	K1.0	Arm (disc type) magazine	Arm (disc-type) magazine of machining center
	K1.1	Umbrella type magazine	Umbrella type magazine of machining center
	K1.2	Drilling tapping counting magazine	Magazine counting signal of drilling tapping center
	K1.3	PMC drilling tapping magazine	PMC tool selection magazine of drilling tapping center
	K1.4	TDS servo magazine	TDS third-party point controls servo magazine
	K1.7	Magazine masking	0: Mask magazine function is, 1: Not mask magazine function
Magazine debugging 1	K2.0	Magazine debugging	Magazine debugging function is, 0: turned off; 1: turned on
	K2.1	Arm debugging	ATC arm debugging is, 0: turned off; 1: turned on

	K2.2	Magazine counting signal conversion	Magazine counting signal selection. 0: normally open; 1: normally closed
	K2.3	Tool position timing limit	Timing of single-tool-position rotation is enabled
	K2.4	12TOOL drilling tapping	Magazine capacity of 12 tools, only used for drilling tapping center
	K2.5	16TOOL	Magazine capacity of 16 tools
	K2.6	21TOOL	Magazine capacity of 21 tools
	K2.7	24TOOL	Magazine capacity of 24 tools
Magazine debugging 2	K3.0	Magazine single-step debugging	Single step function of magazine is, 0: turned off; 1: turned on
	K3.1	Next tool Enable	Next tool calling function of MCP panel is turned on
	K3.2	Large tool Enable	Large/small tool management is turned on
	K3.3	Tool clamping/release of spindle	0: There is not tool clamping/release in position signal of spindle; 1: There is a tool clamping/release in position signal of spindle
	K3.4	Magazine rearrangement Enable	When magazine is at zero, M38 tool number arrangement is enabled
	K3.5	Position orientation	When drilling tapping magazine Z axis position >0, auto orientation is 0: not allowed; 1: is allowed
	K3.6	Tool release button on panel	1: Tool change on panel is allowed. Both tool clamping/release button and external tool release button are valid; 0: only external tool release button is valid
	K3.7	Tool post upside down	When the preselected tool is in position with T command, the tool post upside down function is, 0: disabled; 1: enabled
Spindle function	K4.0	Spindle G01 protection	0: G01 protection is enabled when spindle doesn’t rotate 1: G01 protection is disabled when spindle doesn’t rotate

	K4.1	Spindle overload protection	Overload protection of spindle load current is 0: Disabled; 1: Enabled (coordinate axis parameter 105087 needs to be set)
	K4.2	Spindle current limiting	0: Disabled; 1: Enabled
	K4.3	Spindle speed arrival check	M3 and M4 spindle speed arrive check is 0: enabled; 1: disabled
Axis traverse selection	K5.0	X-axis reference point return direction	Reference point return on X is 0: in positive direction; 1: in negative direction
	K5.1	Self-lock button of rapid traverse button	0: Disabled; 1: Enabled
4th rotary axis	K6.0	Valid 4-axis	Four-axis is, 0: Invalid; 1: Valid
	K6.1	Enable is not interrupted for 4-axis	Enable is not interrupted for when 4-axis is locked
	K6.2	Enable interrupt for 4-axis	Enable is interrupted for when 4-axis is locked (automatic clamping and release)
	K6.3	External 4-axis clamping/release button ON	The external 4-axis clamping/release button is enabled (when K6.1=1)
	K6.4	Non-signaled 4-axis clamping/release check	It is not signaled for 4-axis release/clamping in position check
	K6.5	4-axis clamping/release in position switching	Exchange of 4-axis clamping and release signals
	K6.6	4-axis single step debugging	Single-step debugging for 4-axis automatic clamping/release (when K6.2=1)
	K6.7	Separate 4th axis clamping and release signal	K6.7=1: Two separate signals are used form 4th axis clamping and release respectively K6.7=0: A common input signal is used for both 4th axis release and clamping
MPG function	K7.0	Point-type MPG	MPG axis selection and magnification selection are

			point-to-point signals
	K7.1	Combined-type MPG	MPG axis selection and magnification selection are combined-point switches (e.g. 848D panel)
	K7.2	MCP panel MPG	MPG of MCP panel is, 0: not selected; 1: selected
	K7.3	MPG interrupt	0: Disabled; 1: Enabled
	K7.4	MPG precutting	0: Disabled; 1: Enabled
Lubrication	K8.0	PLC lubrication	Lubrication of PLC control output is, 0: disabled; 1: enabled
	K8.1	Lubrication timing mode	0: Start timing when power on; 1: Start timing when axis moves
	K8.2	Whether automatic lubrication is allowed when power on	0: Automatic lubrication is disabled when power on; 1: Automatic lubrication is Enabled when power on
	K8.3	Lubrication pressure alarm shielding	0: Disabled; 1: Enabled
	K8.4	Lubricant pressure shielding	0: Disabled; 1: Enabled
Function selection switch 1	K9.0	Chip removal CW/CCW	Chip removal is controlled by two buttons: clockwise rotation and counterclockwise rotation
	K9.1	Chip removal CW/stop/CCW	Chip removal is controlled by three buttons: clockwise rotation, stop, and counterclockwise rotation
	K9.2	Water out of spindle center	0: Disabled; 1: Enabled
	K9.3	Chip blower	0: Disabled; 1: Enabled
	K9.4	Rear flushing	Flushing of machine tool at the rear is, 0: disabled; 1: enabled
	K9.5	Watchdog X9/Y9	0: is not selected; 1: HIO-1000 IO watchdog is selected

	K9.6	Watchdog X19/Y19	0: is not selected; 1: HIO-1200 IO watchdog is selected
	K9.7	IO watchdog alarm	0: Disabled; 1: enabled
Function selection switch 2	K10.0	Protective door	0: Disabled; 1: Enabled
	K10.1	Air pressure check	0: Disabled; 1: Enabled
	K10.2	M30 workpiece count selection	0: Workpiece counting is performed with M64; 1: Workpiece counting is performed with M30
	K10.3	MST_LOCK	0: Disabled; 1: Enabled
	K10.5	Hardware limit switch	0: Disabled; 1: Enabled
	K10.6	Yellow light control	0: Disabled (no output); 1: Enabled
Function selection switch 3	K11.0	Intelligence	PLC sampling of intelligent function is, 0: disabled; 1: enabled
	K11.1	Manual intervention of return	0: Disabled; 1: Enabled. Users must set to No. 59 of channel parameter to 1007; otherwise, an alarm will be issued
	K11.3	Auto measurement of tool length	0: Disabled; 1: Enabled. Users must first enable the K14.3 tool setting function to enable the automatic tool length measurement function
5th rotary axis	K12.0	Valid 5-axis	Five -axis is, 0: invalid; 1: Valid
	K12.1	Enable is not interrupted for 5-axis	Enable is not interrupted when he fifth axis is locked
	K12.2	Enable is interrupted for 5-axis	Enable is interrupted when he fifth axis is locked (automatic release/clamping)
	K12.3	External 5-axis clamping/release ON	The external 5-axis clamping/released is enabled (when K12.1=1)
	K12.4	Non-signaled 5th axis clamping/release check	It is not signaled for clamping/release in position check of the fifth axis
	K12.5	5th axis	Exchange of release and clamping signals of the fifth

		clamping/release switching	axis
	K12.6	5th axis signal-step debugging	Single debugging of automatic clamping/release for the fifth axis (when K12.2=1)
	K12.7	Separate 5th axis clamping signal and clamping signal	K2.7=1: Two separate signals are used for 5th axis release and 5th axis clamping K12.7=0: A common input signal is used for both 5th axis release and clamping
Signal handling	K13.0	MPG emergency stop shielding	0: Enabled; 1: Disabled
	K13.1	Panel emergency stop shielding	0: Enabled; 1: Disabled
Measurement function	K14.0	Workpiece measurement	0: Disabled; 1: Enabled
	K14.1	Workpiece probe pulse	Workpiece probe signal is, 0: the level signal; 1: the pulse signal
	K14.3	Tool setter	0: Disabled; 1: Enabled
	K14.4	Probe protection	0: Disabled; 1: Enabled

7.4. Simple Commissioning Steps

(1) Enter the system by the permission of machine manufacturer or higher. Press Diagnosis→Ladder→Ladder information→IO comparison table→IO to enter the user IO setting interface. Fill XY output and input signals of definition of machine user IO into the user IO comparison table to form a mapping.

1	I0.0	X2.1	0	1	紧刀到位
2	I0.1	X2.0	0	1	松刀到位
3	I0.2	X3.5	0	1	扣刀
4	I0.3		0	1	刀臂原点
5	I0.4	X2.2	0	1	刀臂刹车
6	I0.5		0	0	
7	I0.6	X0.5	0	0	刀库计数
8	I0.7	X0.4	0	1	刀库零位
9	I1.0	X2.6	0	0	前进/倒刀...
10	I1.1	X2.5	0	0	后退/回刀...
11	I1.2		0	0	
12	I1.3		0	0	润滑报警
13	I1.4		0	0	
14	I1.5	X1.1	0	0	气压报警

Custom IO mapping configuration

(2) Enter the system by the permission of machine manufacturer or higher. Press Diagnosis→Ladder→Ladder information→K parameter to enter the K parameter setting interface and set the K parameter of PLC based on machine magazine type, spindle function, and feed axis function. Machine magazine commissioning and activation of function are completed by K parameter. See PLC User Manual for specific reference.

地址	注释	7	6	5	4	3	2	1	0
K0	面板类型	0	0	0	0	0	0	0	1
K1	刀库类型	0	0	0	0	0	0	1	0
K2	刀库调试1	0	0	0	0	0	0	0	0
K3	刀库调试2	0	0	0	0	0	0	0	0
K4	主轴功能	0	0	0	0	0	1	0	0
K5	回零方式	0	0	0	0	0	0	0	1
K6	进给轴	0	0	0	0	0	1	0	0
K7	排屑吹气	0	0	0	0	0	0	0	1
K8	润滑功能	0	0	0	0	0	0	0	1

808D面板(K0.0)

Custom K parameter configuration

(3) Auxiliary function configuration

a) If HIO-1000 is used, please turn on K9.5 watchdog X9/Y9; if HIO-1200 is used, please turn on K9.6 watchdog X19/Y19.

b) According to whether MPG is used and whether MPG emergency stop signal is valid, choose to turn on or off the K13.0 MPG emergency stop shielding.

c) After checking user IO comparison table and setting K parameter, release emergency stop on panel for commissioning of other functions.

7.5. PLC of Key Functions

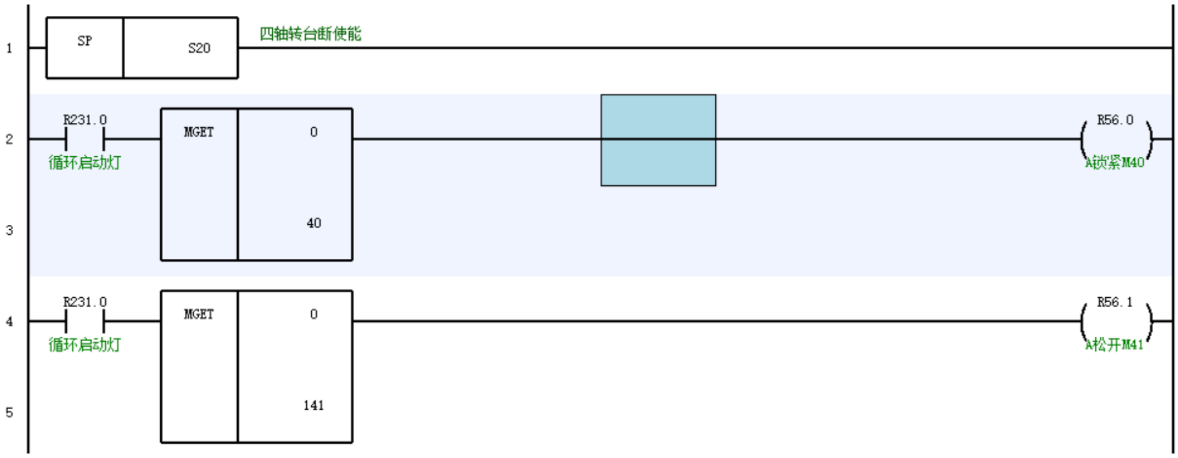
7.5.1. Subprogram module of 4th and 5th functions

Five-axis machine tools have two more axis controls than ordinary milling machines. In order to facilitate PLC debugging, the newly added axis is written in a separate subprogram for PLC control.

The following is the function description of the fourth axis and the fifth axis

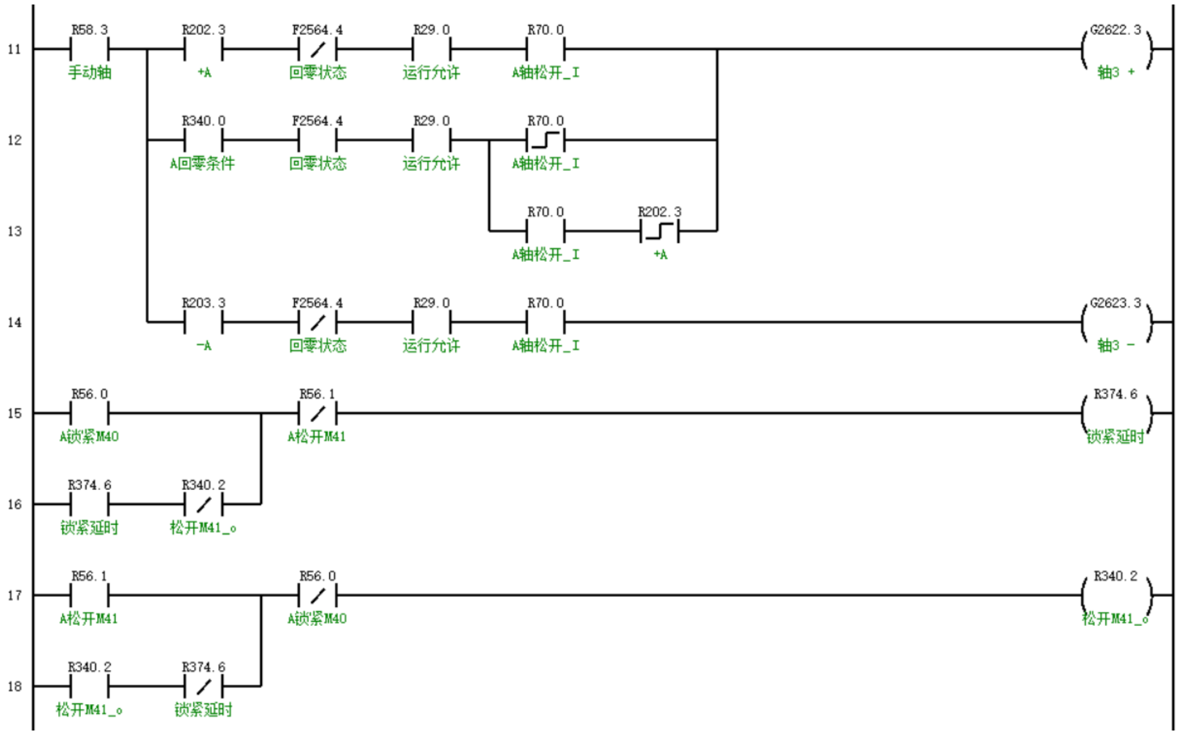
Rotary axis	Clamping	Release
4th axis	M40	M41
5th axis	M45	M46

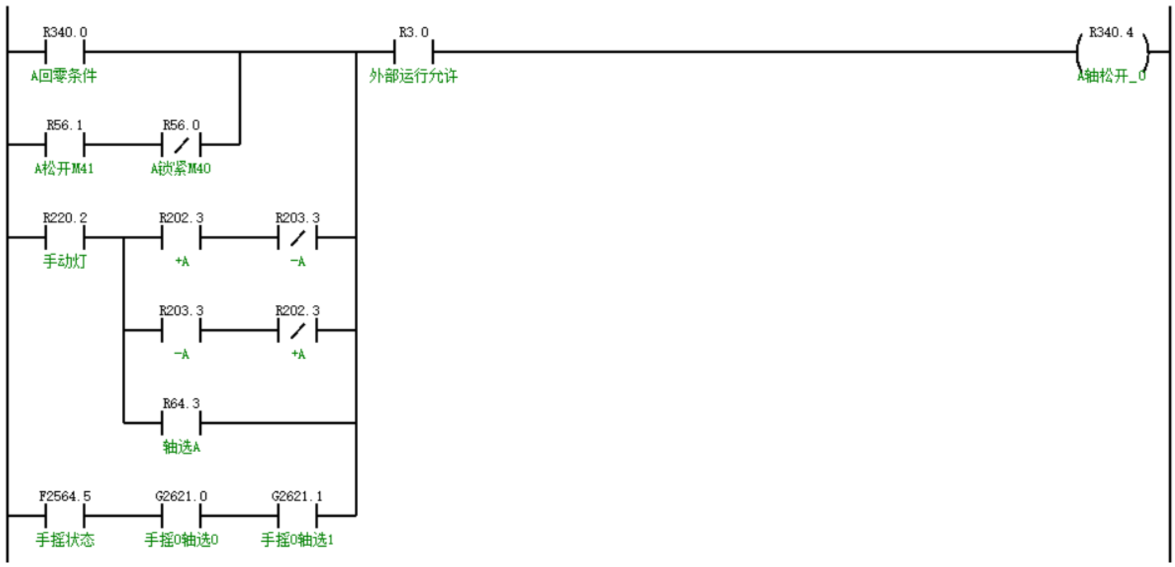
The functions of the fourth axis and the fifth axis are in the subprograms S20 and S21. Here is the brief introduction of subprogram of A axis function

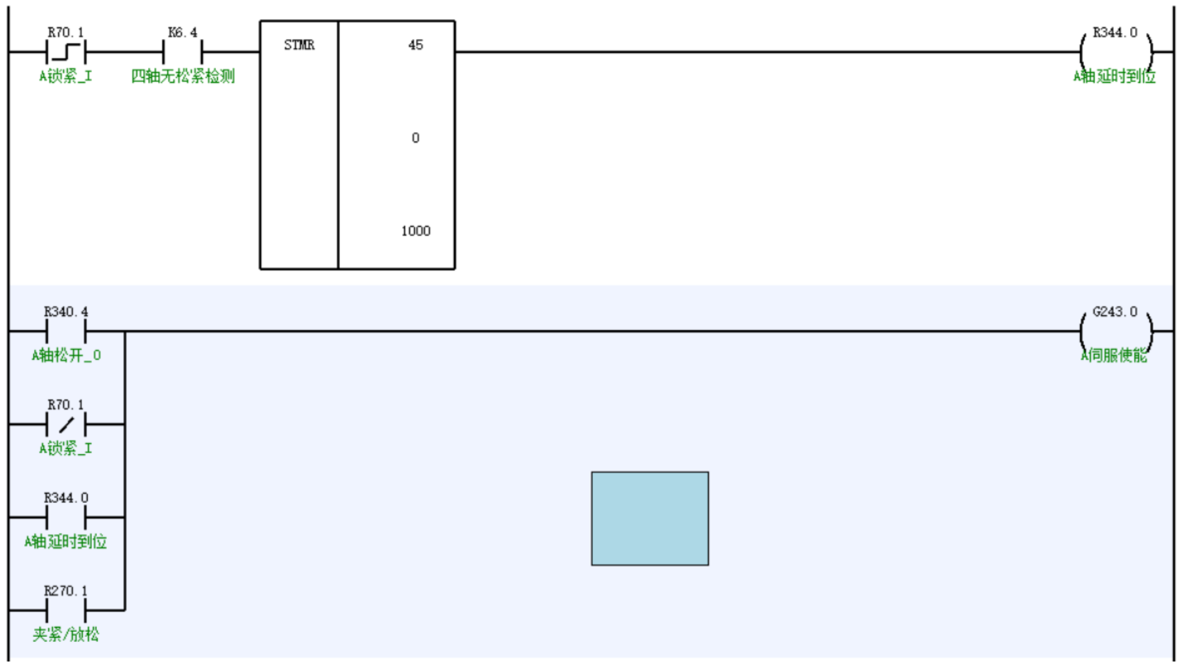


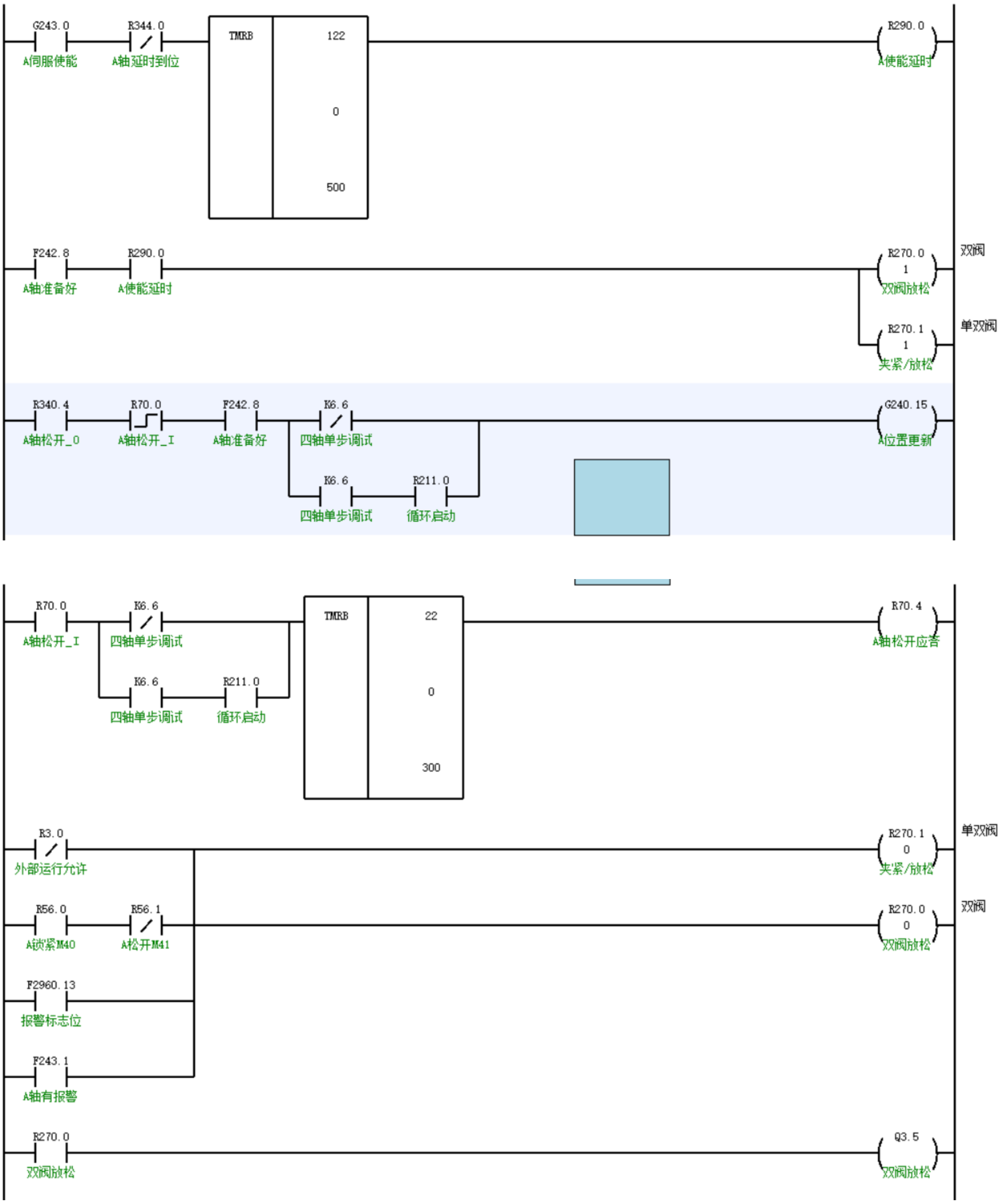
The above is the auto clamping/release code of A axis. It is required to modify the signal points of the release and clamping in position, and modify the release and clamping alarm delay time;

Axis enable signal is in subprogram;




R340.0
A回零条件
R3.0
外部运行允许
R340.4
A轴松开_0
R56.1
A松开M41
R56.0
A锁紧M40
R220.2
手动灯
R202.3
+A
R203.3
-A
R203.3
-A
R202.3
+A
R64.3
轴选A
F2564.5
手摇状态
G2621.0
手摇0轴选0
G2621.1
手摇0轴选1


R70.1
A锁紧_I
K6.4
四轴无松紧检测
STMR
45
0
1000
R344.0
A轴延时到位
R340.4
A轴松开_0
G243.0
A伺服使能
R70.1
A锁紧_I
R344.0
A轴延时到位
R270.1
夹紧/放松



The manual control of A axis is also added to the subprogram module. The first requirement for axis movement is to turn on the release M code of A axis. In this subprogram, in JOG mode, A axis is in the release state by default, and is automatically clamped when switching to auto or single block;

7.5.2. Correspondence of MCP panel point

（1）X input point and Y output point of 818BM panel

	0	1	2	3	4	5	6	7
X480	Auto	Single block	Jog	Increment	Reference point return	Tool change enable	Tool clamping/release	Dry run
X481	Block skip	Optional stop	Z axis lock	Machine lock			Magazine CW	Magazine CCW
X482	X	Y	Z	0%	25%	Spindle CW	Spindle stop	Spindle CCW
X483	Working light	A	B	C	50%	100%	Spindle orientation	Spindle jog
X484	Spindle brake	Protective door	7	8	9	F1	F2	Cooling
X485	Lubrication	Chip blower	Auto power off	—	Rapid traverse	+	F3	F4
X486	Chip removal CW	Chip removal stop	Chip removal CCW	Overtravel release	Cycle start	Feed hold		
X487	Spindle override							
X488	MPG emergency stop, MPG axis selection, MPG magnification							
X489	Feedrate override							
X490	Incremental pulses per period of MPG							
X491								

（2）X input point and Y output point of 848

	0	1	2	3	4	5	6	7
X480	Auto	Single block	Jog	Spindle CW	Spindle stop	Spindle CCW	Custom	Custom
X481	Custom	MDI	Block skip	Reference point return	Spindle orientation	Cooling	Spindle jog	Custom
X482	Custom	Custom	Optional stop	MPG	Machine lock	X	Y	Z
X483	Custom	Custom	Custom				A	B

X484	C	Custom	Custom	Custom	Rapid traverse ×0	Rapid traverse ×25	Protective door	Axis 7
X485	Axis 8	Cycle start	Feed hold	Axis 9	Rapid traverse ×50	Rapid traverse ×100	Working light	JOG—
X486	Rapid traverse	JOG+						
X487	Spindle override magnification							
X488	Feed axis override magnification							
X489								
X490								
X491								
X497	MPG							

8. Highlights

1. When using this system, please set the parameter 000353 Five-axis function application to 1 to ensure that all five-axis functions can be used normally.

2. This system supports five-axis machining programs generated by mainstream CAM manufacturers such as CATIA, UG and Cimatron, and provides users with corresponding CAM five-axis post-processing packages.

3. Only Cimatron fully supports the post-processing of the inclined surface machining cycle, and the generated program can be executed directly. Programs generated by other manufacturers need to be modified manually for inclined surface machining.

4. When perform machining in any line mode, it is necessary to correctly set the parameter [040114] Axis in-position sequence in any line based on the current axis configuration, otherwise it cannot be used normally. For example, for a machine tool with AC dual swivel head structure, this parameter is set to 101211. The scanning mode can be used directly and normally, and the RTCP function needs to be turned on in the MDI mode for machining in the non-scanning mode.

Appendix A AC Dual Rotary Table Calibration

1. Detection before calibration of machine tool

1) Ensure geometric accuracy of X, Y and Z axes of machine tool have been measured.

2) Measure axis line of A axis, axis line of C axis, and zero position of A axis.

◆ **Axis line of A axis**

Axis line of A axis must be parallel with X axis, and dial indicator should be installed on the spindle head. Identify an XZ plane on the worktable and press the gauge on the plane. Then, rotate A axis and observe whether the value changes. Normally, the value changes within 0.02mm.

◆ **Axis line of C axis and zero position of A axis**

Install the dial indicator on the rotatable part of the spindle and make indicator contact the worktable, rotate the spindle by hand, observe change of reading, adjust the rotary table so that error in X direction is within the allowable range and angle of A axis so that error in Y direction is within the allowable range, and set the position of A axis as the zero position. (This measurement is used suppose the workbench is vertical is vertical to axis line of C axis)

Axis line of C axis must be parallel with Z axis and dial indicator should be installed on the spindle head. At the zero position of A axis, identify an XY plane on the worktable and make indicator contact the plane. Then, rotate C axis and observe whether the value changes. Normally, the value changes within 0.02mm.

3) Check positional accuracy of A axis and C axis

◆ **Position of A axis**

Step 1:

a) Reference point return of A axis and C axis.

b) Install dial indicator on the spindle head and make the indicator contact the worktable.

c) Move the worktable along Y direction.

d) Observe change of the value. Normally, the value changes within 0.02mm.

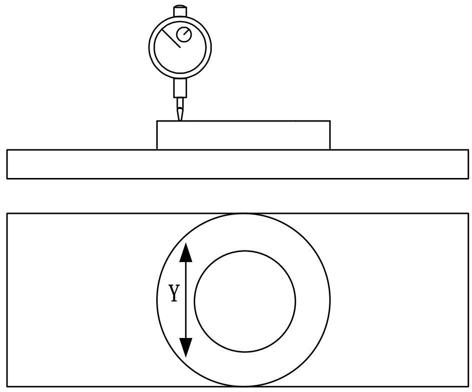


Figure 0-1 A axis accuracy test

Step 2:

a) 90° of A axis and 0° of C axis.

b) Install dial indicator on the spindle head and make the indicator contact the upper surface of the worktable.

c) Move the spindle head along Z direction.

d) Observe change of the value. Normally, the value changes within 0.02mm.

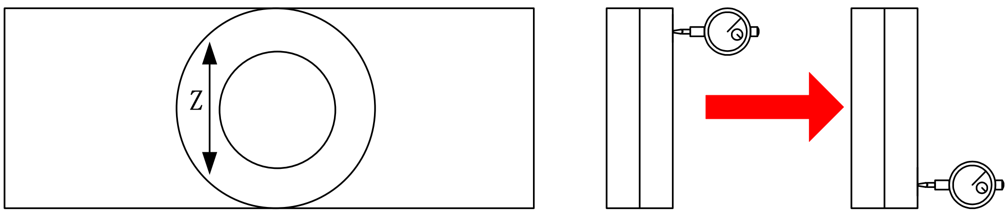


Figure 0-2 A axis accuracy test

- **Position of C axis**

a) Install square gage on the rotary table. When C axis is 0°, make a plane of square gage parallel with YZ plane;

b) Rotate C axis 0°, 90°, 180° and 270° respectively;

c) Install dial indicator on the spindle, make the indictor contact the vertical plane, move the spindle up and down, and observe change of reading.

On the condition that positional accuracy of A and C axes is qualified, RTCP parameters calibration can be performed; otherwise, adjust or compensate rotary axis accordingly.

2. RTCP parameter calibration of machine tool

RTCP parameters to be calibrated include C rotary table center vector and AC axis offset vector.

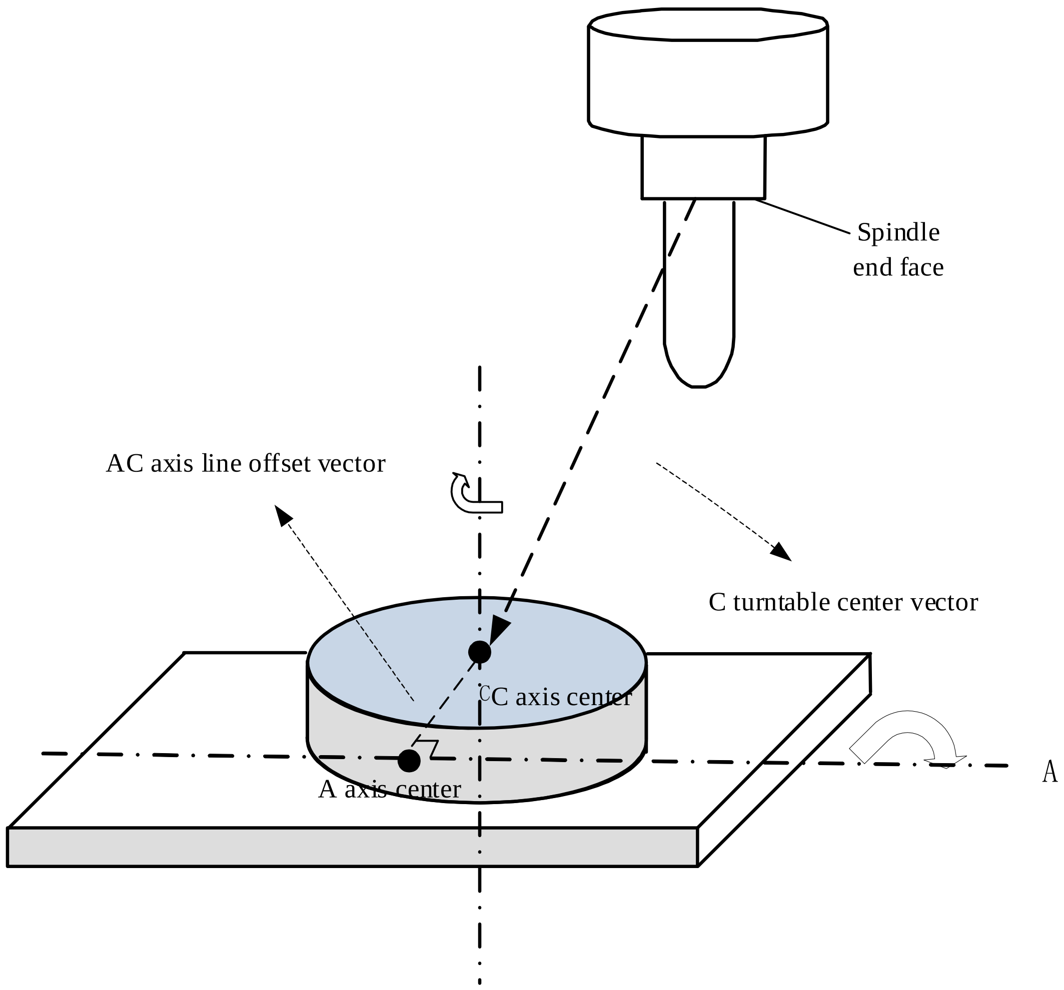


Figure 0-3 RTCP parameter diagram

(1) X and Y offset vectors of C rotary table center

Step 1:

a) 0° of A axis and 0° of C axis.

b) Install edge finder on the spindle head and position the contact head close to the height of surface of C axis.

c) Position the contact head to the left and right edges of inner diameter of C axis, and record coordinates of X axis as X1 and X2.

d) Calculate X coordinate of C axis: Xc=(X1+X2)/2.

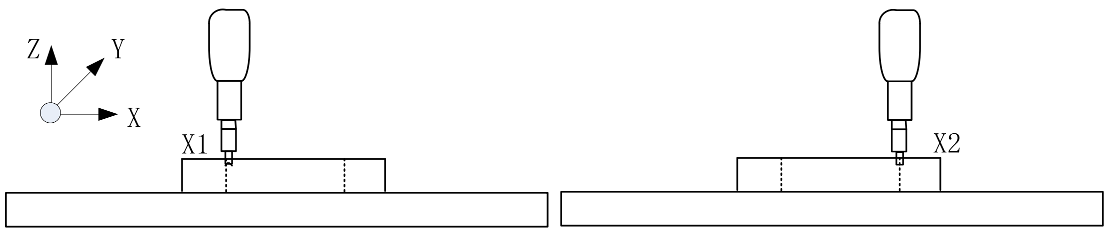


Figure 0-4 Rotation center in X direction of C axis

e) Position the contact head to front and back edges of inner diameter of C axis, and record coordinates of Y axis as Y1 and Y2.

f) Calculate Y coordinate of C axis: Yc =(Y1+Y2)/2.

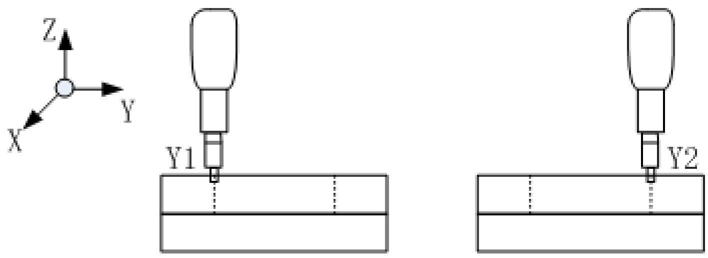


Figure 0-5 Rotation center in Y direction of C axis

Step 2:

a) Install the dial indicator on the spindle head.

b) Adjust X and Y coordinates of the worktable to the position calculated in Step 1.

c) Install test arbor on the spindle head.

d) Adjust X and Y coordinates so that reading of the dial indicator is limited to 0.02mm when C axis rotate a revolution.

e) Record current coordinates Xc and Yc as the X and Y coordinates of C axis.

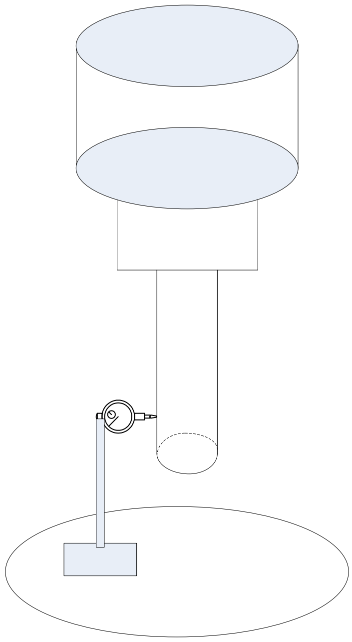


Figure 0-6 Test of rotation center of C axis

(2) Offset vector of AC axis lines

Only offset vector in Y direction of AC axis should be calibrated. Adopt different calibration methods based on travel range of A axis.

- **A axis can rotate 90° and -90°**

Step 1:

a) A axis rotates -90°.

b) Install the dial indicator on the spindle head.

c) Make the dial indicator contact the highest point of outer diameter of the C rotary table.

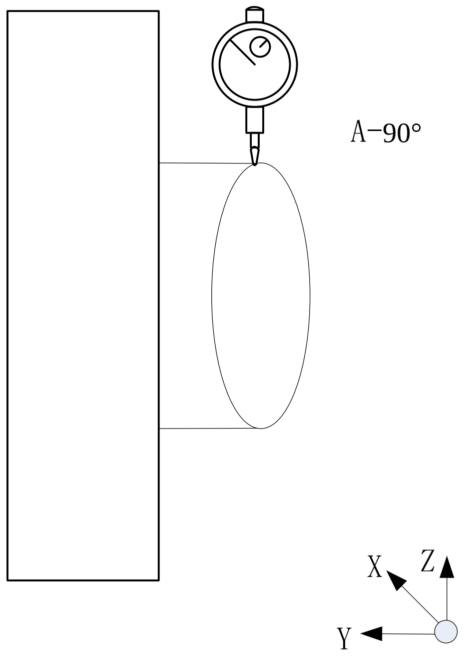


Figure 0-7 Offset vector of AC axis when A axis can rotate ±90°

d) Relative clear of Z axis.

Step 2:

a) A axis rotates 90°.

b) Make the dial indicator contact the highest point of outer diameter of the C rotary table and keep the reading the same with that in Step 1.

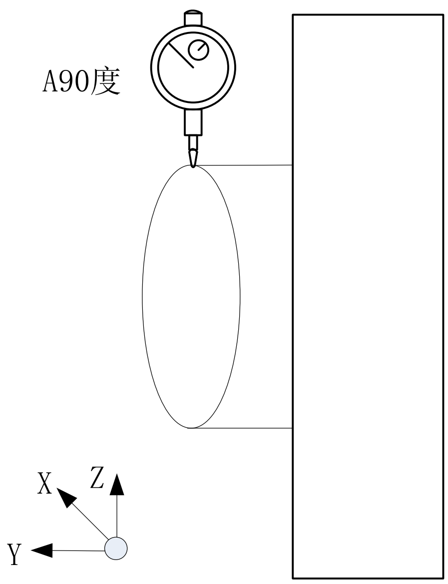


Figure 0-8 Offset vector of AC axis when A axis can rotate ±90°

c) Read relative coordinate of Z axis and record it as Z1.

d) Then, OFFY = Z1/2.

- **A axis cannot rotate 90° or -90°**

Select the position closest to plus and minus travel of A axis. ±40° of A axis is selected in this example.

Step 1:

a) Install the dial indicator on the spindle head.

b) 0° of A axis, identify the highest point in X axis direction of the worktable, and perform relative clear of X axis.

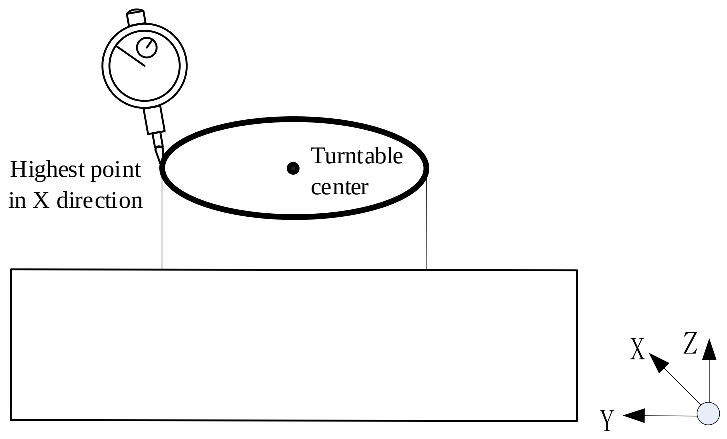


Figure 0-9 Offset vector of AC axis when A axis cannot rotate ±90°

Step 2:

a) -40° of A axis and 0° of C axis.

b) Ensure relative zero position of X axis, move Y and Z axes, and have the dial indicator towards the edge of the worktable as shown below. Relative clear of Z axis.

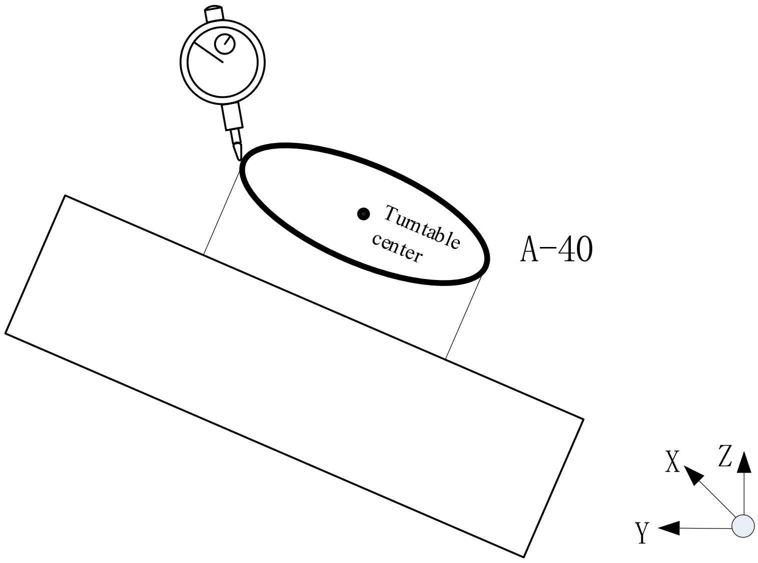


Figure 0-10 Offset vector of AC axis when A axis cannot rotate ±90°

Step 3:

a) 40° of A axis and 0° of C axis.

b) Ensure relative zero position of X axis, move Y and Z axes, and have the dial indicator towards the edge of the worktable as shown below. Keep the reading identical to that in Step 2.

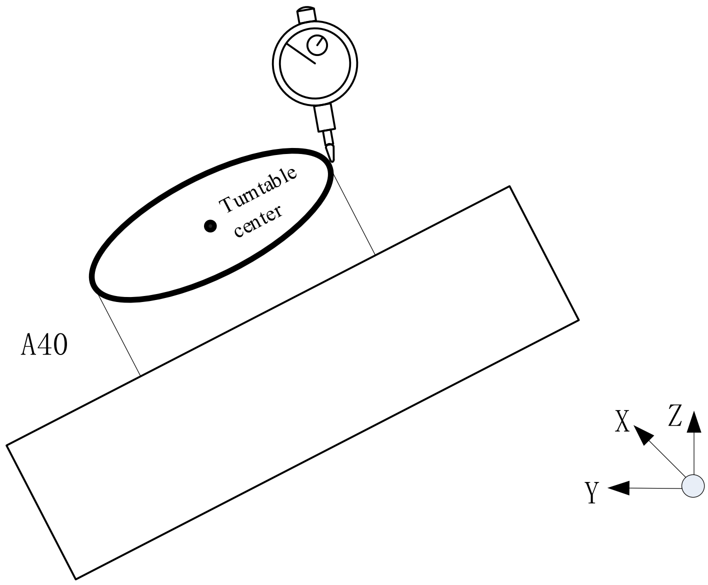


Figure 0-11 Offset vector of AC axis when A axis cannot rotate for ±90°

c) Read current relative coordinate of Z axis and record it as Z1.

d) Then, AC_OFFY = Z1/2.

Note: It is inconvenient to calibrate offset vector in Y direction of AC axis manually, since the value is normally small. Temporarily set AC_OFFY as 0 and make corrections through datum sphere

(3) Offset vector in Z direction of C axis center

Step 1:

a) 0° of A axis and 0° of C axis.

b) Install the workblank on the worktable and install the tool.

c) Position the worktable on the position of X and Y offset vectors of C axis center, and perform relative clear of Y axis coordinate.

d) Position the tool nose on the workpiece surface and record current Z coordinate as Z1.

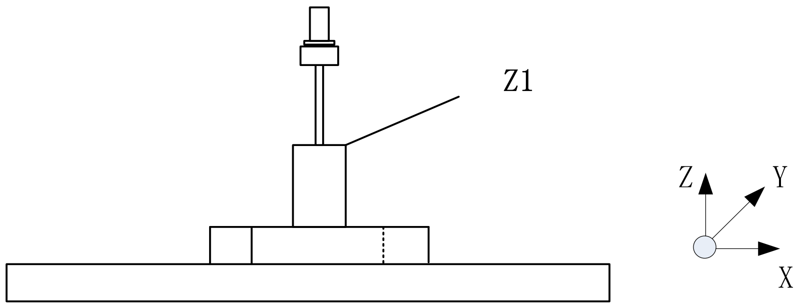


Figure 0-12 Offset vector in Z direction of C axis center

Step 2:

a) -90° of A axis and 0° of C axis.

b) Perform the tool setting, and position the tool edge on the workpiece surface.

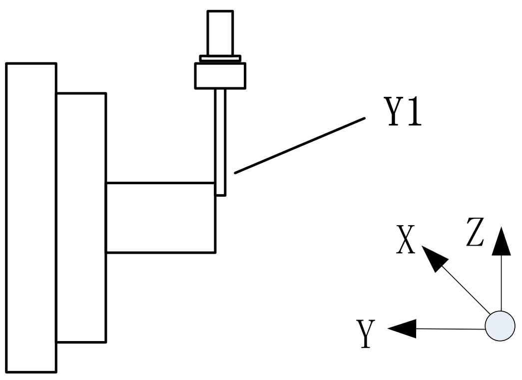


Figure 0-13 Offset vector in Z direction of C axis center

c) Record current Y coordinate as Y1.

d) Distance from the upper surface of the workpiece to A axis H = Y1-AC_OFFY+R (where, AC_OFFY is the offset distance in Y direction of AC axis and R is the radius of tool).

Step 3:

a) Install the tool on the spindle.

b) Install the dial indicator on the worktable.

c) Position the dial indicator probe on the tool nose and reset perform the relative clear of Z axis.

d) Position the dial indicator probe on the spindle end face and record current Z coordinate as Z2.

e) Tool length L = -Z2.

f) Offset vector in Z direction of C axis Zc = Z1+H – L.

After the aforesaid calibration process is completed, fill the calibration data in channel parameters in the following table.

040400	Initial direction of tool (X)	0.0
040401	Initial direction of tool (Y)	0.0
040402	Initial direction of tool (Z)	1.0
040425	Rotary table structure type	AC
040426	Vector X of the first rotary axis direction of rotary table	-1.0
040427	Vector Y of the first rotary axis direction of rotary table	0.0
040428	Vector Z of the first rotary axis direction of rotary table	0.0
040429	Vector X of the second rotary axis direction of rotary table	0.0
040430	Vector Y of the second rotary axis direction of rotary table	0.0
040431	Vector Z of the second rotary axis direction of rotary table	-1.0

040432	Vector X of the first rotary axis offset of rotary table	0.0
040433	Vector Y of the first rotary axis offset of rotary table	AC_OFFY
040434	Vector Z of the first rotary axis offset of rotary table	0
040435	Vector X of the second rotary axis offset of rotary table	Xc
040436	Vector Y of the second rotary axis offset of rotary table	Yc
040437	Vector Z of the second rotary axis offset of rotary table	Zc

Note: Direction vector must be set based on the rotation direction of current rotary axis during parameter setting.

3. Effectiveness Test of RTCP Function of Machine Tool

(1) RTCP function test of C axis

a) 0° of C axis

b) Install datum sphere, measure length L and radius R of datum sphere and fill the data L-R in the tool compensation table.

c) Install the dial indicator in X negative direction as shown below.

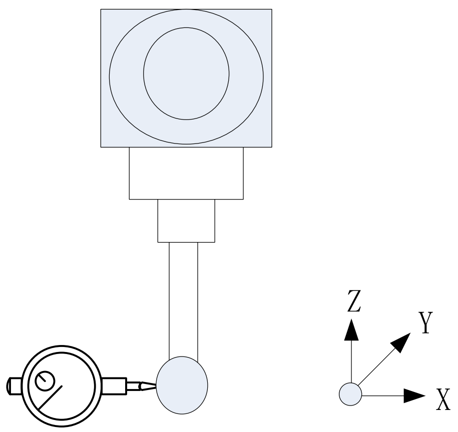


Figure 0-14 RTCP accuracy test of C axis

d) Write G code test program, as shown below:

G54

F500

G43.4H1（enable RTCP function）

G90C0

C90

C180

C270

G49

M30

e) Observe value change of the dial indicator. Normally, the value changes within 0.02mm when C axis rotates a revolution. If reading is greater than 0.02mm, calibration parameters can be corrected according to reading.

- Observe data of X direction. With C0° as the basis, the difference of the dial indicator between C0° and C180° is dx, then the adjusted value of vector X of the second rotary axis offset of the rotary table is:

$$Xc = Xc + dx/2$$

- Observe data of Y direction. With C90° as the basis, the difference of the dial indicator between C90° and C270° is dy, the adjusted value of vector Y of the second rotary axis offset of the rotary table is:

$$Yc = Yc - dy/2$$

Note: The aforesaid dx and dy have positive and negative directions. If users don't know how to calculate it, fill in a data for test using trial-and-error method. If results are incorrect, it means that data is compensated reversely. Compensate data until basic test requirements are met.

(2) RTCP function test of A axis

a) 0° of A axis.

b) Install datum sphere, measure length L and radius R of datum sphere and fill data L-R in the tool compensation table.

c) Install the dial indicator as shown below.

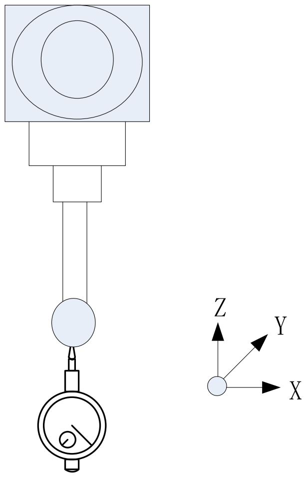


Figure 0-15 RTCP accuracy test of A axis

d) Measure effectiveness of Y offset distance of AC line and create the test program in G code, as shown below:

G54

F500

G43.4H1（enable RTCP function）

G90A0

A30

A0

A-30

G49

M30

Observe the value change of dial indicator and compare reading at A30 and A-30. If reading is limited to the range of 1 thread, the value of AC_OFFY needs not adjusted. If reading is not limited to the range, reading at A30 should prevail. The difference of reading is doffy and the adjusted value is:

AC_OFFY = AC_OFFY+ doffy/2

e) Measure effectiveness of Z offset of C axis center and edit test program for G code, as shown below:

G54

F500

G43.4H1（enable RTCP function）

G90A0

A-90

M30

Observe the value change of dial indicator and compare the readings at the time of A0 and A-90. If reading is in the range of 0.02mm, the value of Zc needs not adjusted. If reading is out of the range, and the difference between readings of these two is dz, then the adjusted value is:

$$Zc = Zc - dz$$

Note: doff and dz have a positive value and a negative value like effectiveness test of C axis. If users don't know how to calculate it, fill in a data for test using trial-and-error method. If results are incorrect, it means that data is compensated reversely. Compensate data until basic test requirements are met.

f) Re-execute the test program in step e and observe the moving of dial indicator pointer during the entire movement process

Appendix B BC Dual Swivel Head Calibration

1. Detection before calibration of machine tool

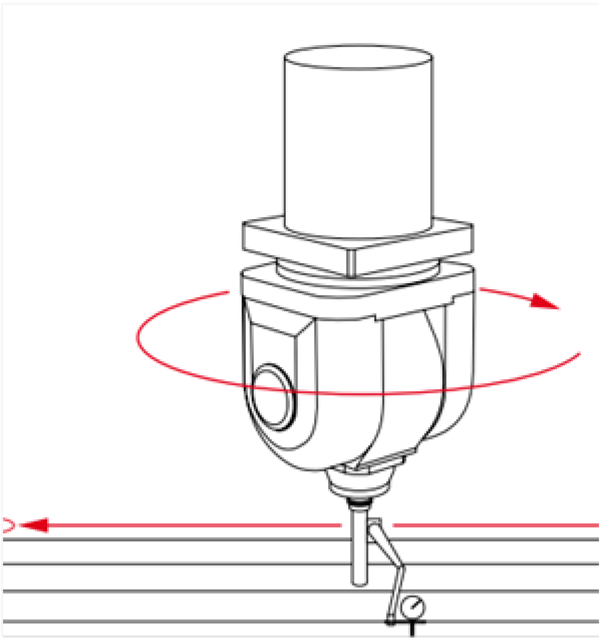
(1) Ensure geometric accuracy of X, Y and Z axes of machine tool has been measured.

(2) Measure axis line and zero position of B/C axis.

C axis is a master axis and it is not affected by B axis, so C axis should be measured first.

◆ **Axis line of C axis**

Axis line of C axis must be parallel with Z axis and dial indicator should be installed on the spindle head. Identify an XY plane on the worktable, B axis swings an angle and make the indicator touch the plane. Then, rotate C axis and observe whether the value changes. Normally, the value changes within 0.02mm. Otherwise, C axis should be adjusted mechanically to ensure accuracy.



Accuracy test of C axis

◆ **Axis line of B axis and zero position of C axis**

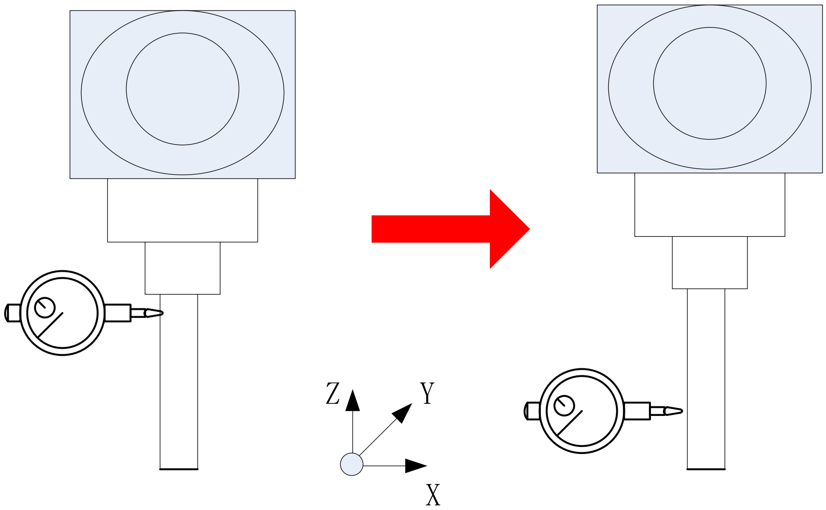
Axis line of B axis must be parallel with Y axis and dial indicator should be installed on the spindle head. Identify an XZ plane on the worktable and have the indicator touch the plane. Then, rotate B axis and observe whether the value changes. Adjust C axis so that the reading is within the permissible error range, and set the position of C axis as the its zero position. Normally, the value changes within 0.02mm.

◆ **Zero position of B axis**

a） Reference point return of B axis and C axis.

b) Install test arbor, make the dial indicator contact the test bar and vertical to XY plane of machine tool.

c） Move Z axis up and down.



B axis zero test

d) Adjust C axis so that reading is within the permissible error range, and set the position of B axis as the zero position. Normally, the value changes within 0.02mm.

(3) Check positional accuracy of B axis and C axis

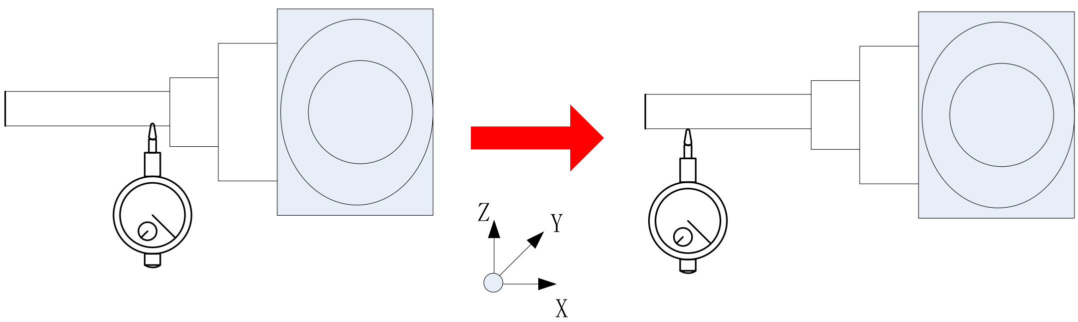
◆ **Position of B axis**

Step 1:

a) 90° of B axis

b) Make the dial indicator contact the test arbor and vertical to XY plane of machine tool.

c) Move X axis back and forth.



90° positioning accuracy test of B axis

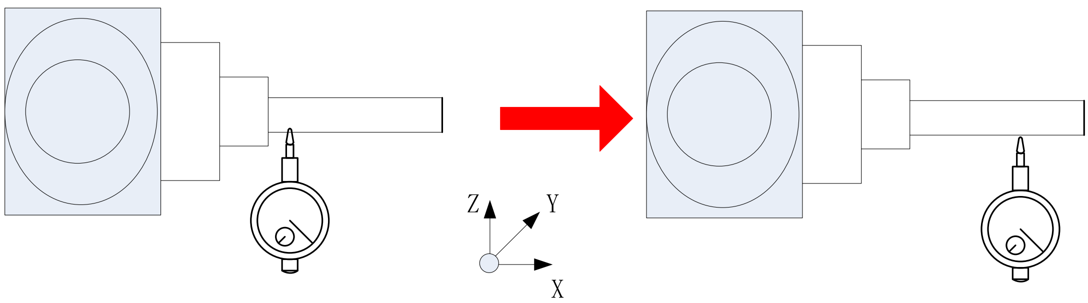
d) Observe changes of the value. Normally, the value changes within 0.02mm.

Step 2:

a) -90° of B axis

b) Make the dial indicator contact the test arbor and vertical to XY plane of machine tool.

c) Move X axis back and forth.



90° positioning accuracy test of B axis

Observe changes of the value. Normally, the value changes within 0.02mm.

◆ **Position of C axis**

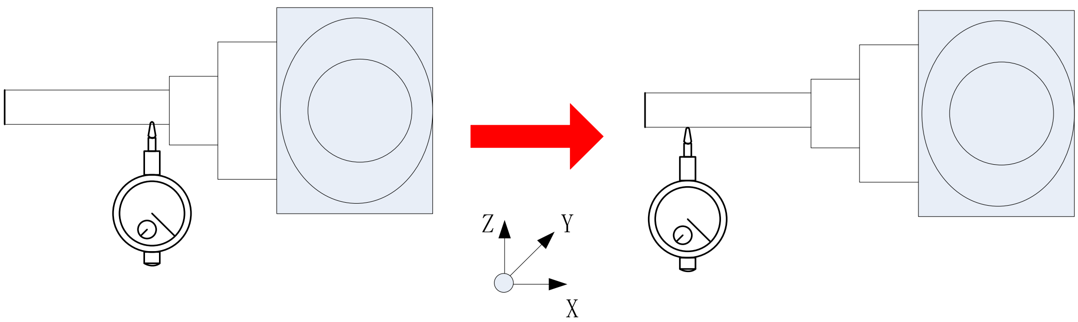
90° or -90° of B axis

Step 1:

a) 0° of C axis

b) Make the dial indicator contact the test arbor and vertical to XY plane of machine tool.

c) Move X axis back and forth.



Positioning accuracy test of C axis

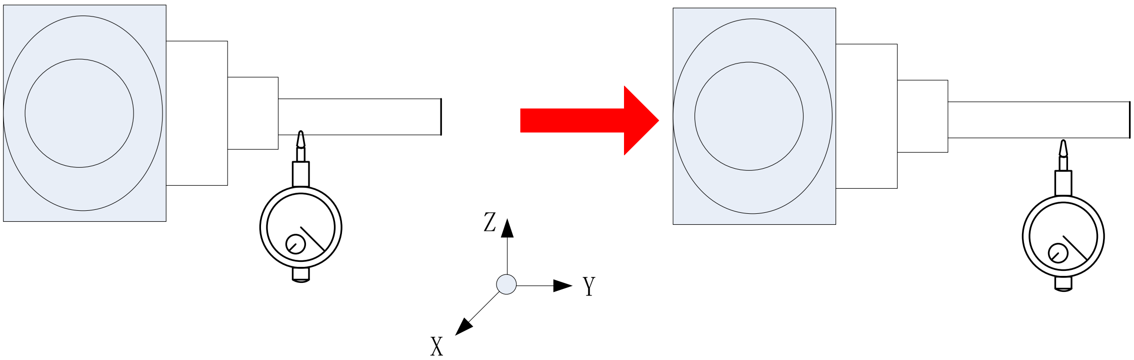
d) Observe changes of the value. Normally, the value changes within 0.02mm.

Step 2:

a) 90° of C axis

b) Make the dial indicator contact the test arbor and vertical to XY plane of machine tool.

c) Move Y axis back and forth.



Positioning accuracy test of C axis

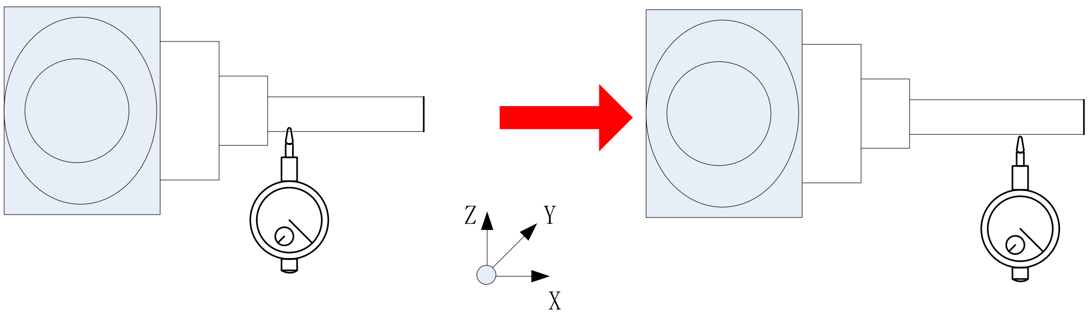
d) Observe changes of the value. Normally, the value changes within 0.02mm.

Step 3:

a) 180° of C axis

b) Make the dial indicator contact the test arbor and vertical to XY plane of machine tool.

c) Move X axis back and forth.



Positioning accuracy test of C axis

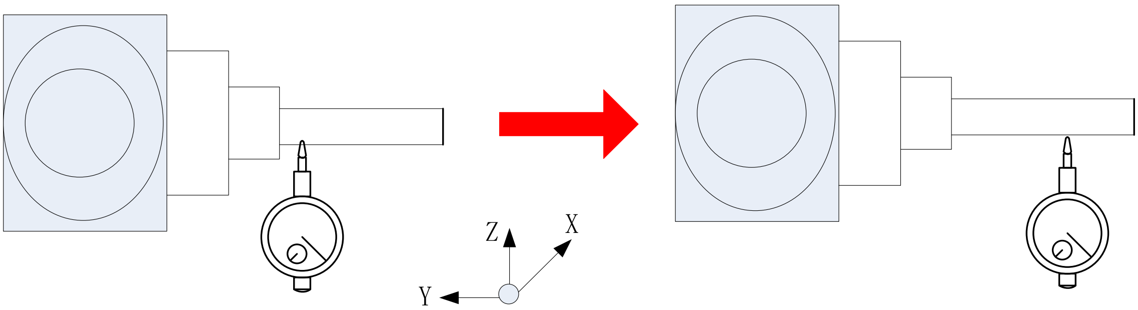
d) Observe changes of the value. Normally, the value changes within 0.02mm.

Step 4:

a) 270° of C axis

b) Make the dial indicator contact the test arbor and vertical to XY plane of machine tool.

c) Move Y axis back and forth.



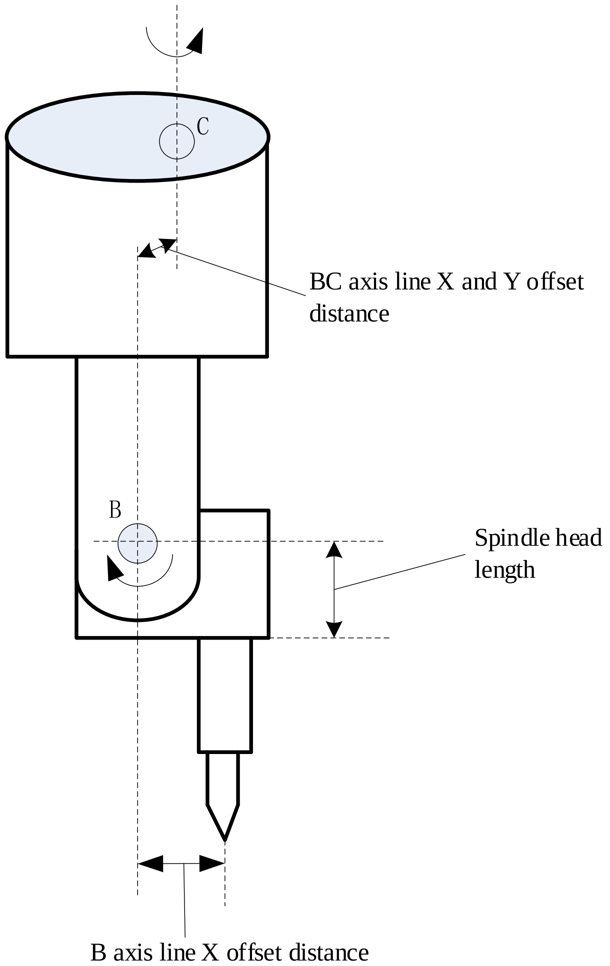
Positional accuracy test of C axis

Observe changes of the value. Normally, the value changes within 0.02mm.

On the condition that the positioning accuracy of A and C axes is qualified, RTCP parameter calibration is allowed; otherwise, adjust or compensate the rotary axis accordingly.

(4) RTCP parameter calibration of machine tool

RTCP parameters to be calibrated include X offset distance of B axis line, Y offset distance of B axis line, offset distance of BC axis lines, and length of spindle head.



RTCP parameter calibration diagram

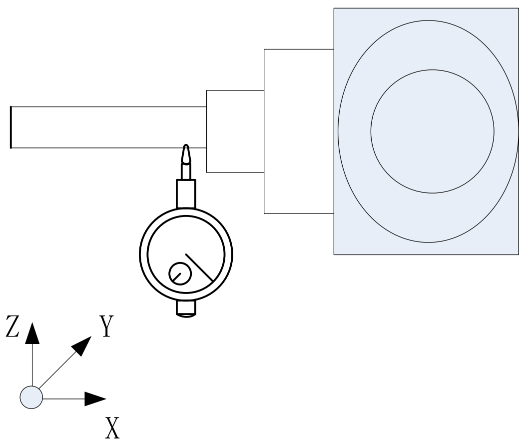
1）X offset distance of B axis line

- **X offset distance of B axis line**

Step 1:

a) 90° of B axis and 0° of C axis.

b) Place the dial indicator on the bottom of test arbor (major diameter).



Offset distance in X direction of B axis

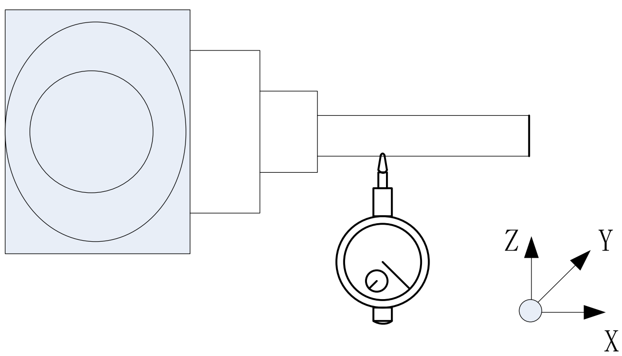
c) Set reading of the dial indicator as 0.

d) Relative clear of Z axis.

Step 2:

a) -90° of B axis and 0° of C axis.

b) Place the dial indicator on the bottom of test bar (major diameter) again and ensure the same reading as that in step 1.



Offset distance in X direction of B axis

c) Set reading of the dial indicator as 0.

d) View relative coordinate value of Z axis and record it as Z1.

e) Calculate X offset distance of B axis relative to the spindle:

$$SPOFFBx = Z1/2$$

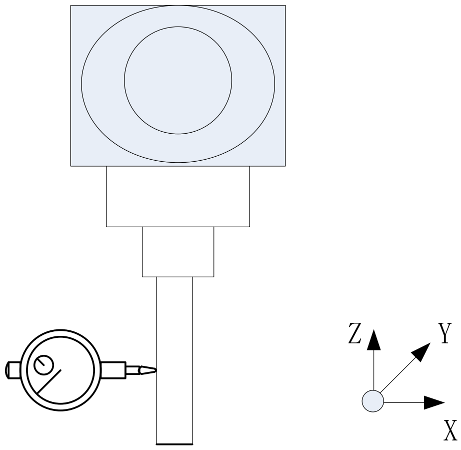
2) X and Y offset distances of BC axes

◆ **X offset distance of BC axes**

Step 1:

a) 0° of B axis and 0° of C axis.

b) Install the dial indicator in the X negative direction of test arbor and make the dial indicator contact the test arbor. (X direction)



X offset distance of BC axis

c) Set reading of the dial indicator as 0.

d) Relative clear of X axis.

Step 2:

a) 0° of B axis and 180° of C axis.

b) Make the dial indicator contact the X negative direction of test arbor again and ensure reading is the same with that in step 1

c) View relative coordinate value of X axis and record it as X1.

d) Calculate X offset distance of C axis relative to the spindle:

$$SPOFFCx=(-x1)/2。$$

e) Calculate X offset distance of BC axes

$$BCOFFx = SPOFFCx - SPOFFBx$$

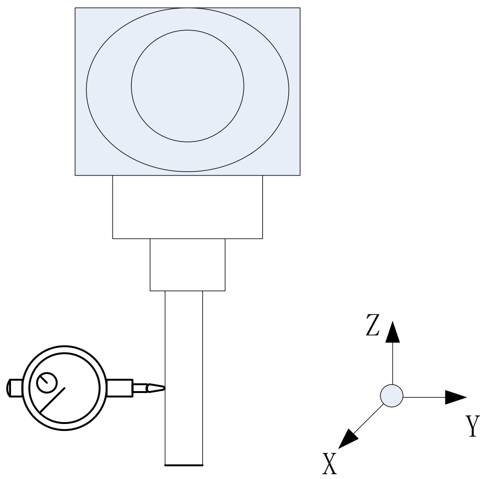
◆ **Y offset distance of BC axes**

If the set control point of B axis is the same as the center point of spindle on Y, Y offset distance of BC axes is equal to Y offset distance of C axis relative to the spindle.

Step 1:

a) 0° of B axis and 0° of C axis.

b) Install the dial indicator in the Y negative direction of test arbor and make the dial indicator contact the test arbor. (Y direction)



Offset distance in Y direction of BC axes

c) Set reading of the dial indicator as 0.

d) Relative clear of Y axis.

Step 2:

a) 0° of B axis and 180° of C axis.

b) Make the dial indicator contact the Y negative direction of test arbor again and ensure reading is the same with that in step 1.

c) View relative coordinate value of Y axis and record it as Y1.

d) Calculate Y offset distance of B axis relative to the spindle:

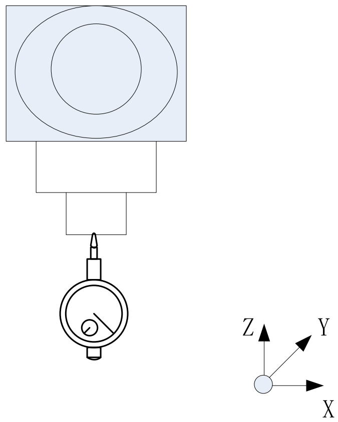
$$BCOFFy = (-y1)/2$$ 。

3) Length of spindle head

Step 1:

a) 0° of B axis and 0° of C axis.

b) Make the dial indicator contact the spindle end face.



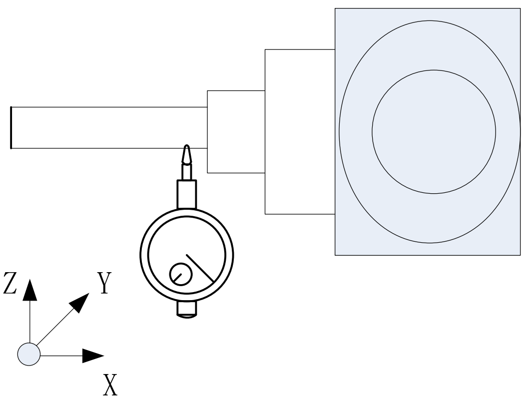
Spindle head length measurement

c) Relative clear of Z axis.

Step 2:

a) 90° of B axis and 0° of C axis.

b) Have the dial indicator contact the lowest position of test arbor (major diameter) and ensure the same reading as that in step 1.



Spindle head length test

c) Set reading of the dial indicator as 0.

d) View relative coordinate value of Z axis and record it as Z1.

e) Length of spindle head:

SPLEN = -(z1+d/2) - SPOFFBx。(X direction offset of B axis is SPOFFBx, diameter of test arbor is d)

After the aforesaid calibration process is completed, fill calibration data in the channel parameters in the following table. (The structure type of the rotary table is guaranteed to be empty)

040400	Initial direction of tool (X)	0.0
040401	Initial direction of tool (Y)	0.0
040402	Initial direction of tool (Z)	1.0
040410	Structure style of swivel head	CB
040411	Vector X of the first rotary axis direction of swivel head	0.0
040412	Vector Y of the first rotary axis direction of swivel head	0.0
040413	Vector Z of the first rotary axis direction of swivel head	1.0
040414	Vector X of the second rotary axis direction of swivel head	0.0
040415	Vector Y of the second rotary axis direction of swivel head	1.0
040416	Vector Z of the second rotary axis direction of swivel head	0.0
040417	Vector X of the first rotary axis offset of swivel head	BCOFFx
040418	Vector Y of the first rotary axis offset of swivel head	BCOFFy
040419	Vector Z of the first rotary axis offset of swivel head	0.0
040420	Vector X of the second rotary axis offset of swivel head	SPOFFBx
040421	Vector Y of the second rotary axis offset of swivel head	0.0
040422	Vector Z of the second rotary axis offset of swivel head	SPLEN

Note: Direction vector must be set based on rotation direction of current rotary axis during parameter setting. For specific observation of direction, please refer to five-axis parameter description.

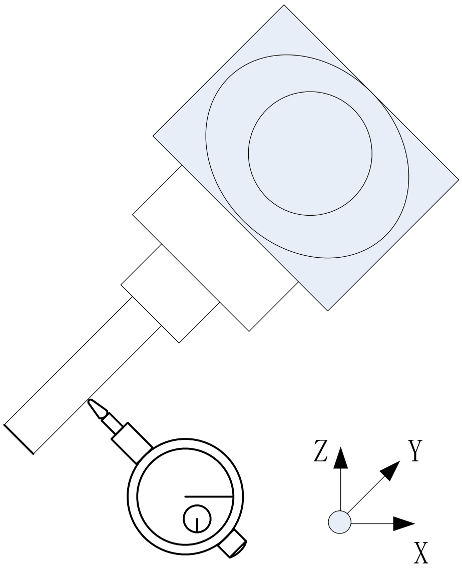
4. Effectiveness Test of RTCP Function of Machine Tool

1) Positioning accuracy test of B axis

Detect the positioning accuracy of B axis using normal tool feed and retract.

a) B axis rotates an angle.

b) Have the indicator contact the straight bar.



Positioning accuracy test of B axis

c) Move the tool along the normal direction through normal tool feed and retract command G53.3L_. It should be noted that RTCP function should be enabled before this command is used, call G43.4 command.

d) Observe changes of values of the dial indicator. Normally, the value changes within 0.02mm.

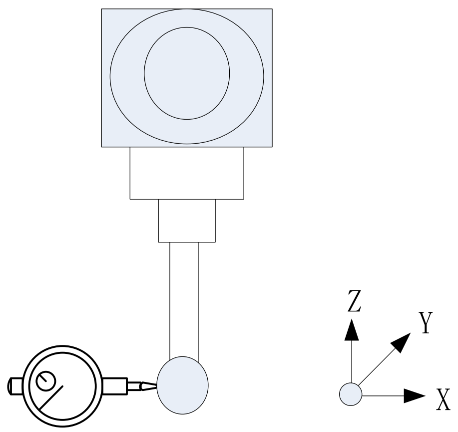
2) RTCP function test

- **RTCP function test of C axis**

a) 0° of C axis

b) Install datum sphere, measure length L and radius R of datum sphere and fill data L-R in the tool compensation table.

c) Install the dial indicator in X negative direction as shown below.



RTCP accuracy test of C axis

Write G code test program, as shown below:

```
G54
F500
G43.4H1（enable RTCP function）
G90C0
C90
C180
C270
 G49
 M30
```

d) Observe change of values of the dial indicator. Normally, the value changes within 0.02mm when C axis rotates a revolution. If the reading is greater than 0.02mm, calibration parameters can be corrected according to the reading.

- Observe data of X direction. The difference of the dial indicator between C0° and C180° is dx taking C0° as the reference, the adjusted value of offset vector X of the second rotary axis of the rotary table is:

$$Xc = Xc - dx/2$$

- Observe data of Y direction. Taking C90° as the reference, the difference of the dial indicator between C90° and C270° is dy, the adjusted value of offset vector Y of the second rotary axis of the rotary table is:

$$Yc = Yc + dy/2$$

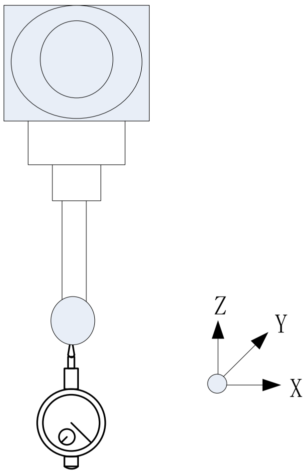
Note: Whereas the aforesaid dx and dy have positive and negative directions, if they cannot be calculated, trial-and-error can be adopted. Fill in a data to test using trial-and-error. If results are incorrect, it means that data is compensated reversely. Compensate data until basic test requirements are met.

◆ **RTCP function test of B axis**

a) 0° of B axis.

b) Install datum sphere, measure length L and radius R of datum sphere and fill data L-R in the tool compensation table.

c) Install the dial indicator as shown below.



RTCP accuracy test of B axis

d) Write G code test program, as shown below:

```
G54
F500
G43.4H1（enable RTCP function）
G90b0
B90
B0
B-90
G49
```

e) Observe changes of the dial indicator. Normally, the value changes within 0.02mm when B axis moves. If the reading is greater than 0.02mm, calibration parameters can be corrected according to the reading.

- Compare reading of B90° and B-90°. With B90° as the reference, the difference of the dial indicator between B90° and B-90° is dx, the adjusted value of offset vector Y of the second rotary axis of the swivel head is:

  SPOFFBx = SPOFFBx + dx/2

  After SPOFFBx is corrected and accuracy requirements are met, the value of BCOFFx should be corrected

  BCOFFx = BCOFFx – dx/2

- Compare reading of B0° and B90°. With B0° as the referemce, the difference of the dial indicator between B0° and B90° is dz, the adjusted value of offset vector Z of the second rotary axis of the swivel head is:

  SPLEN = SPLEN - dz

Note: Whereas the aforesaid dx and dz have positive and negative directions. If they cannot be calculated, fill in a data to test by trial-and-error. If results are incorrect, it means that data is compensated oppositely. Compensate data until basic test requirements are met.

Supplementary instruction:

(1) For AC dual swivel head machine tool, refer to this file. The main difference is that the symbols of calculation formulas are reverse for the two: offset distance of B axis relative to spindle for BC dual swivel head and offset distance of A axis relative to spindle for AC dual swivel head.

SPOFFAy= -Z1/2

(2) While performing RTCP test using datum sphere, symbols of computational formula of the corrected value should also be inverted:

SPOFFBAy= SPOFFAy - dy/2

BCOFFy = BCOFFy + dy/2

Appendix C Calibration of B Swivel Head C Rotary Table

1. Detection before calibration of machine tool

（1）Ensure geometric accuracy of X, Y and Z axes of machine tool has been measured.

（2）Measure axis line and zero position of B axis and axis line of C axis.

- **Axis line of B axis**

Axis line of B axis must be parallel with Y axis and dial indicator should be installed on the spindle head.

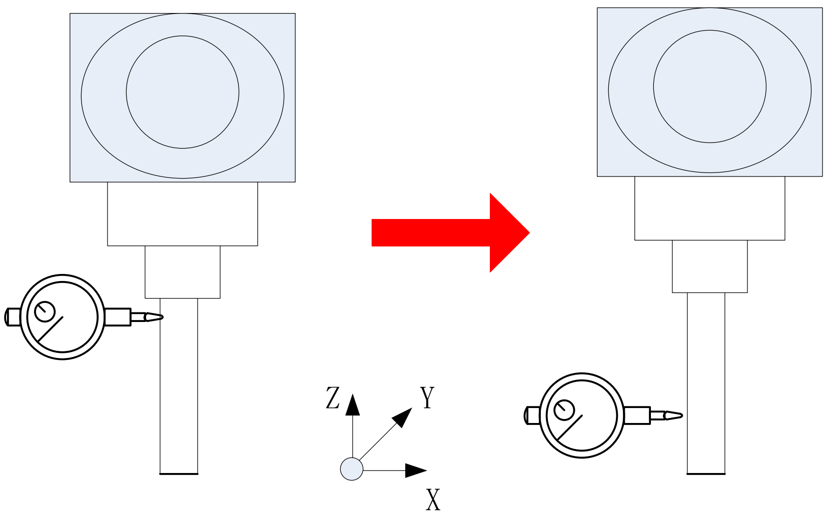
Identify an XZ plane and have the indicator contact the plane. Then, rotate B axis and observe whether the dial indicator changes.

- Zero position of B axis

Reference point return of B axis and C axis.

a) Install test arbor, have the indicator contact the test arbor and vertical to XY plane of machine tool.

Move Z axis up and down.



B axis zero detection

b) Adjust B axis to make reading within the permissible error range and identify the position of B axis as the zero position. Normally, the value changes within 0.02mm.

- **Axis line of C axis**

Axis line of C axis must be parallel with Z axis and dial indicator should be installed on the spindle head. Identify an XY plane on the worktable and have the indicator contact the plane. Then, rotate C axis and observe whether the value changes. Normally, the value changes within 0.02mm.

（3）Check positioning accuracy of B axis

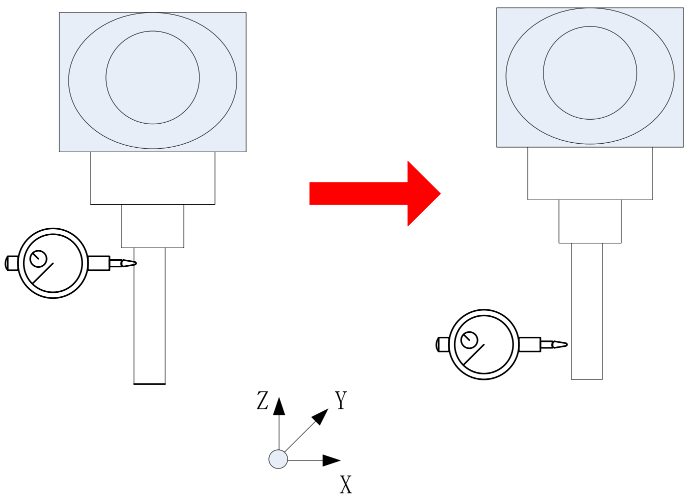
- **Position of B axis**

Step 1:

a) Reference point return of B axis.

b) Install test arbor, make the dial indicator contact X negative direction of the test arbor.

c) Move Z axis up and down.



Positioning accuracy test of B axis

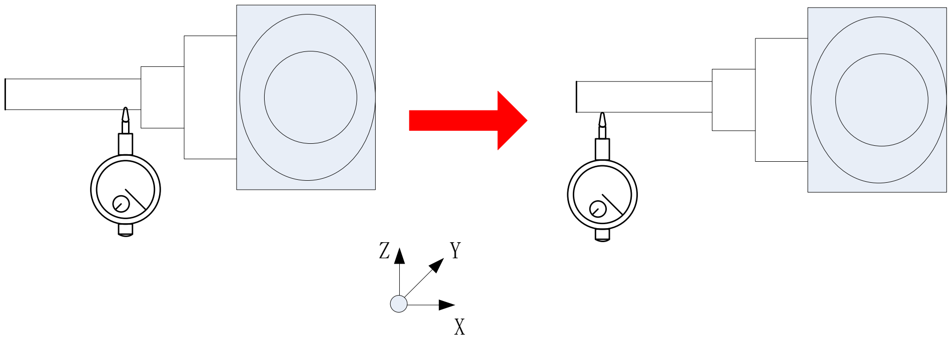
d) Observe changes of the value. Normally, the value changes within 0.02mm.

Step 2:

a) 90° of B axis.

b) Have the dial indicator contact the test arbor.

c) Move X axis back and forth.



Positioning accuracy test of B axis

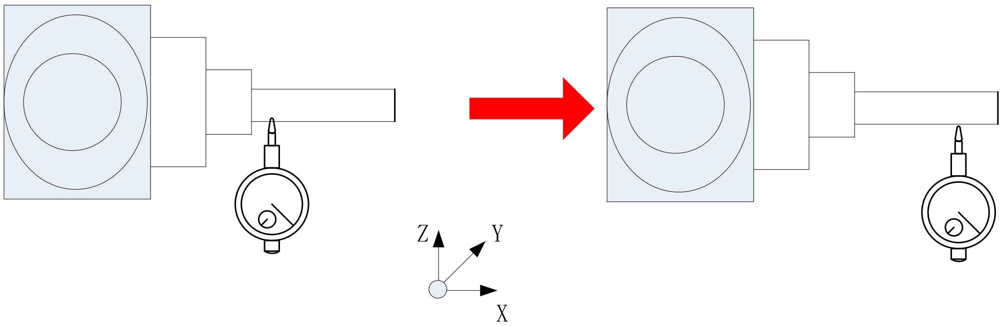
d) Observe changes of the value. Normally, the value changes within 0.02mm.

Step 3:

a) -90° of B axis.

b) Have the dial indicator contact the test arbor.

c) Move X axis back and forth.



Positioning accuracy test of B axis

d) Observe change of the value. Normally, the value changes within 0.02mm

- **Position of C axis**

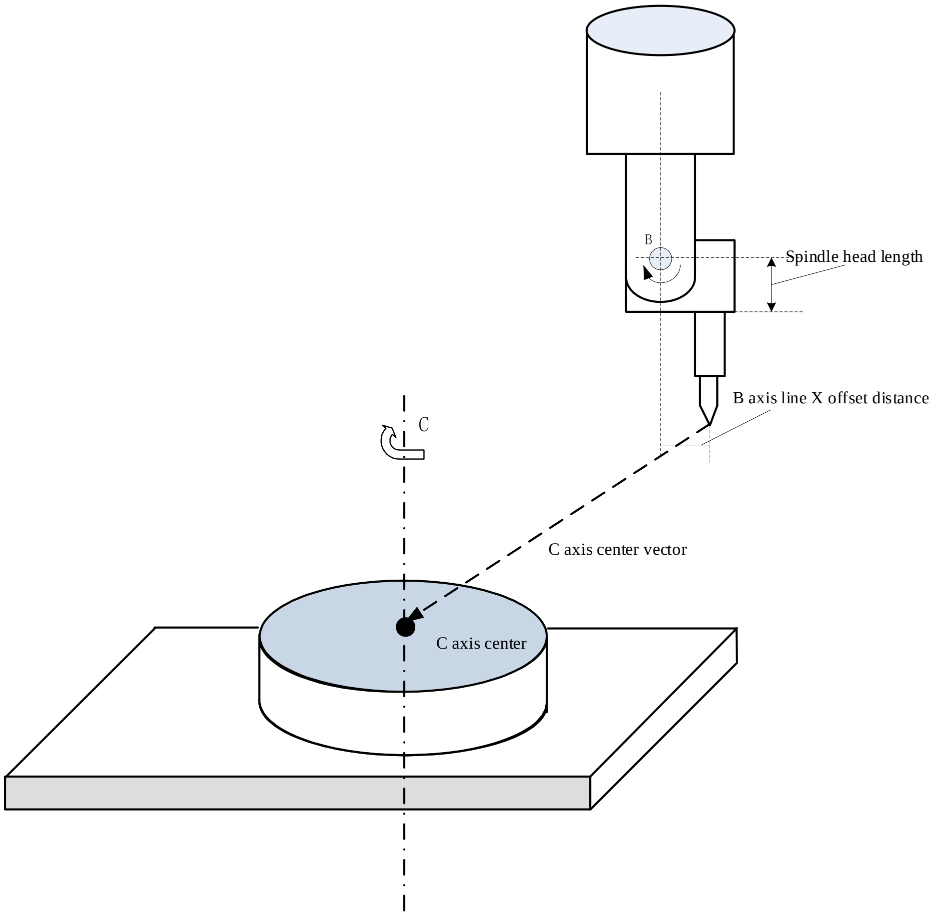
a) Install square gage on the rotary table. When C axis is 0°, make a plane of square gage parallel with YZ plane;

b) Rotate C axis 0°, 90°, 180° and 270° respectively;

c) Install dial indicator on the spindle, make the indicator contact the vertical plane, move the spindle up and down and observe changes of the reading.

2. RTCP parameter calibration of machine tool

RTCP parameters to be calibrated include X offset distance of B axis line, length of spindle head, and position of C axis center.



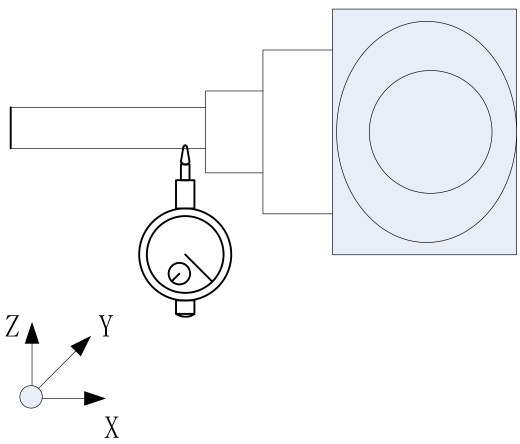
RTCP parameter diagram of machine tool

（1）X offset distance of B axis line

Step 1:

a) 90° of B axis

b) Make the indicator contact the lowest position of test arbor (major diameter).



Offset distance in X direction of B axis

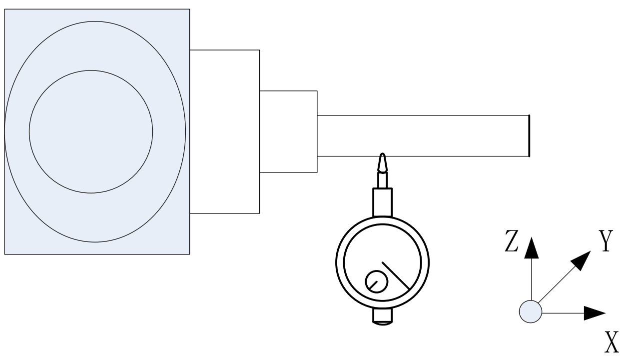
c) Set the reading of the dial indicator as 0.

d) Reset relative coordinates of Z axis.

Step 2:

a) -90° of B axis and 0° of C axis.

b) Make the dial indicator contact the test arbor (major diameter) again and ensure the same reading as that in step 1.



Offset distance in X direction of B axis

c) Set the reading of the dial indicator as 0.

d) View relative coordinate value of Z axis and record it as Z1.

e) Calculate X offset distance of B axis line relative to the axis line of the spindle:

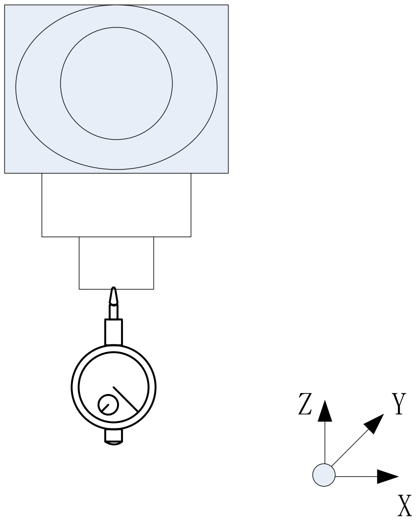
SPOFFBx = Z1/2

（2）Length of spindle head

Step 1:

a) 0° of B axis and 0° of C axis.

b) Make the dial indicator contact the spindle end face.



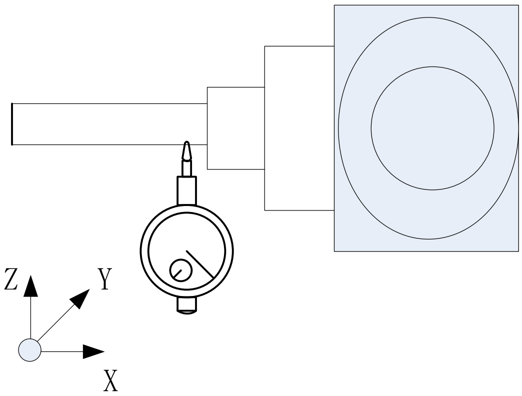
Spindle head length measurement

c) Relative clear of Z axis.

Step 2:

a) 90° of B axis and 0° of C axis.

b) Make the indicator contact the test arbor (major diameter) and ensure the same reading as that in step 1.



Spindle head length measurement

c) Set the reading of the dial indicator as 0.

d) View relative coordinate value of Z axis and record it as Z1.

e) Length of spindle head:

$SPLEN = -(z1+d/2) - SPOFFBx$。

(X offset of B axis is SPOFFBx, diameter of test arbor is d)

（3）X and Y offset vectors of C axis center

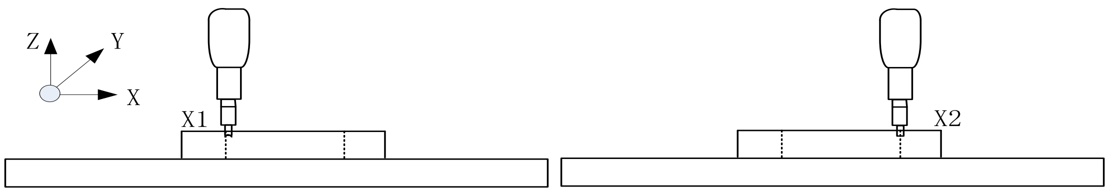
Step 1:

a) 0° of A axis and 0° of C axis.

b) Install edge finder on the spindle head and make the probe contact the edge of C axis.

c) Make the probe contact the left and right edges of inner diameter of C axis and record coordinates of X axis as X1 and X2.

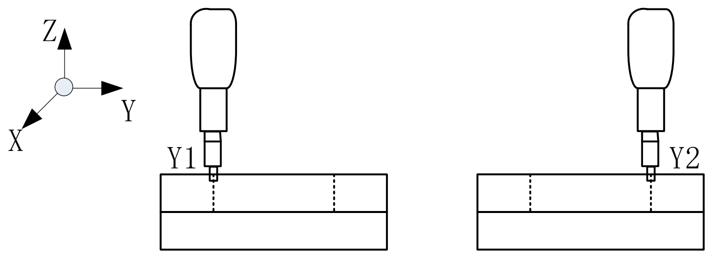
d) Calculate X coordinate of C axis: Xc=(X1+X2)/2.



Offset distance in X direction of C axis center

e) Make the probe contact the front and rear edges of inner diameter of C axis and record coordinates of Y axis as Y1 and Y2.

f) Calculate Y coordinate of C axis: Yc =(Y1+Y2)/2.



Offset distance in Y direction of C axis center

Step 2:

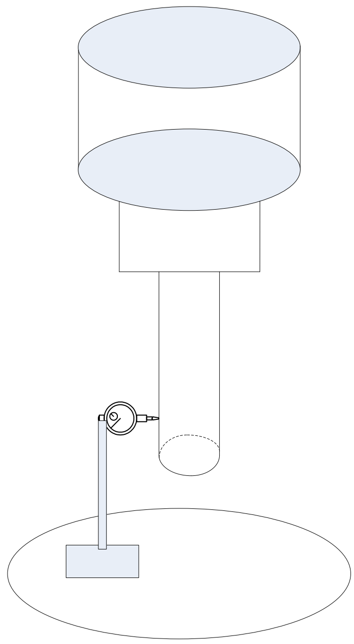
a) Install the dial indicator on the spindle head.

b) Adjust X and Y coordinates of the worktable to the position calculated in Step 1.

c) Install test arbor on the spindle head.

d) Slightly adjust X and Y coordinates to make reading of the dial indicator is limited within 0.02mm when C axis rotates a revolution.

e) Record current coordinates Xc and Yc as X and Y coordinates of C axis.



C axis center position test

◆ **Z offset vector of C axis center**

Z coordinate needs not calibrated. It can be any value, just set it to 0.

After the aforesaid calibration is completed, fill the calibration data in the channel parameters in the following table.

040400	Initial direction of tool (X)	0.0
040401	Initial direction of tool (Y)	0.0
040402	Initial direction of tool (Z)	1.0
040410	Style of swivel head structure	B
040411	Vector X of the first rotary axis direction of swivel head	0.0
040412	Vector Y of the first rotary axis direction of swivel head	0.0
040413	Vector Z of the first rotary axis direction of swivel head	0.0
040414	Vector X of the second rotary axis direction of swivel head	0.0
040415	Vector Y of the second rotary axis direction of swivel head	1.0
040416	Vector Z of the second rotary axis direction of swivel head	0.0
040417	Vector X of the first rotary axis offset of swivel head	0.0
040418	Vector Y of the first rotary axis offset of swivel head	0.0
040419	Vector Z of the first rotary axis offset of swivel head	0.0
040420	Vector X of the second rotary axis offset of swivel head	SPOFFBx
040421	Vector Y of the second rotary axis offset of swivel head	0.0
040422	Vector Z of the second rotary axis offset of swivel head	SPLEN
040425	Rotary table structure type	C
040426	Vector X of the first rotary axis direction of rotary table	0.0
040427	Vector Y of the first rotary axis direction of rotary table	0.0

040428	Vector Z of the first rotary axis direction of rotary table	0.0
040429	Vector X of the second rotary axis of rotary table	0.0
040430	Vector Y of the second rotary axis of rotary table	0.0
040431	Vector Z of the second rotary axis of rotary table	-1.0
040432	Vector X of the first rotary axis offset of rotary table	0.0
040433	Vector Y of the first rotary axis offset of rotary table	0.0
040434	Vector Z of the first rotary axis offset of rotary table	0.0
040435	Vector X of the second rotary axis offset of rotary table	Xc
040436	Vector Y of the second rotary axis offset of rotary table	Yc
040437	Vector Z of the second rotary axis offset of rotary table	0

Note: Direction vector must be set according to rotation direction of current rotary axis during parameter setup. Please refer to five-axis parameter description for how to observe rotation direction.

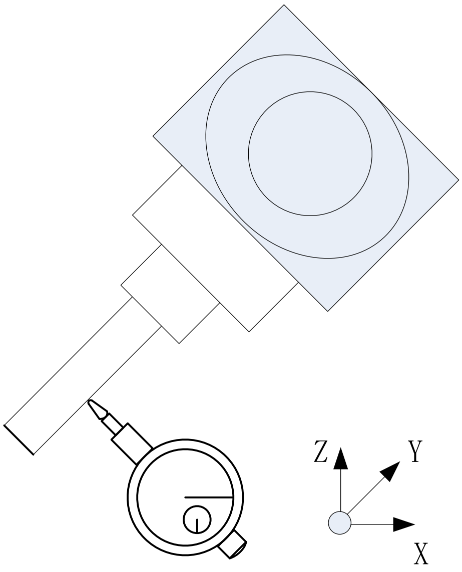
3. Validity test of RTCP function of machine tool

(1) Positioning accuracy test of B axis

Detect positioning accuracy of B axis of the system using normal feed and retract function.

a) B axis rotates an angle.

b) Make the indicator contact the straight bar.



Positioning accuracy test of B axis

c) Move the tool axis in the normal direction through normal feed and retract command G53.3L_. It should be noted that RTCP function should be enabled before this command is used, call G43.4 command.

d) Observe changes of the dial indicator. Normally, the value changes within 0.02mm.

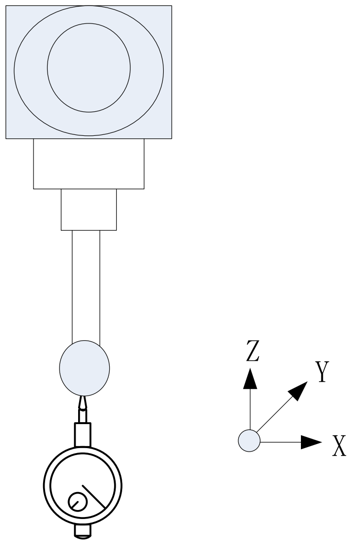
(2) RTCP function test

◆ **RTCP function test of B axis**

a) 0° of B axis.

b) Install datum sphere, measure length L and radius R of datum sphere, and fill data L-R in the tool compensation table.

c) Install the dial indicator as shown below.



RTCP accuracy test of B axis

d) Write G code test program, as shown below:

```
G54
F500
G43.4H1（enable RTCP function）
G90 B0
B90
B0
B-90
 G49
```

e) Observe changes of the dial indicator. Normally, the value changes within the range of 2 threads when B axis moves. If reading is greater than 2 threads, calibration parameters can be corrected according to reading.

- Compare reading at B90° and B-90°. With B90° as the basis, the error value of the meter between B90° and B-90° is dx, the adjusted value of offset vector Y of the second rotation axis of the swing head is:

  SPOFFBx = SPOFFBx + dx/2

- Compare the reading of B0° and B90°. With B0° as the reference, the difference between B0° and B90° is dz, and the adjusted value of offset vector Z of the second rotary axis of the swivel head is:

  SPLEN = SPLEN - dz

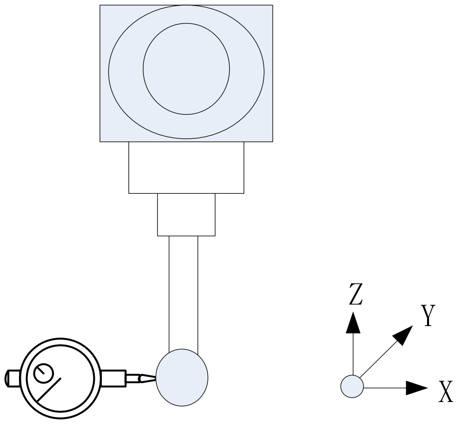
Note: Whereas the aforesaid dx and dy have positive and negative directions. If they cannot be calculated, trial-and-error can be adopted. If results are incorrect, it means that data is compensated oppositely. Compensate data until basic test requirements are met.

◆ **RTCP function test of C axis**

a) 0° of C axis

b) Install datum sphere, measure length L and radius R of datum sphere, and fill data L-R in the tool compensation table.

c) Install the dial indicator in X negative direction as shown below.



RTCP accuracy test of C axis

d) Write G code test program, as shown below:

```
G54
F500
G43.4H1（enable RTCP function）
G90C0
C90
C180
C270
G49
M30
```

e) Observe changes of the dial indicator. Normally, the value changes within 0.02mm when C axis rotates a revolution. If the reading is greater than 0.02mm, calibration parameters can be corrected according to the reading.

➢ Observe data of X direction. With C0° as the reference, the difference of the dial indicator between C0° and C180° is dx, the adjusted value of offset vector X of the second rotary axis of the rotary table is:

$$Xc = Xc + dx/2$$

➢ Observe data of Y direction. With C90° as the basis, the difference of the dial indicator between C90° and C270° is dx, the adjusted value of offset vector Y of the second rotary axis of the rotary table is:

$$Yc = Yc - dy/2$$

Note: Whereas the aforesaid dx and dy have positive and negative directions. If they cannot be calculated, fill in a data and test by trial-and-error. If results are incorrect, it means that data is compensated oppositely. Compensate data until basic test requirements

Appendix D Case for Automatic Calibration of AC Dual Rotary Table

1. Calibration of machine tool

AC dual rotary table machine tool

2. Calibration tool

Renishaw OMP-600 probe

Datum sphere: Renishaw diameter 25mm

主轴测头

3. Calibration process

A. Installation of Renishaw probe

In HNC-8, M40 and M41 commands are developed to turn on and off the probe. Figure 1 below is the electronical wiring diagram of probe signal receiver, in which the dark green end represents status input of probe and is connected to I/O input X point of the CNC system. When the probe deforms, the signal receiver receives the deformation state, the dark green end outputs high level from low level and G31 block skip command of the CNC system is activated by signal on the rising edge. PLC ladder of G31 command activation is shown in Figure 2 below, in which X4.5 represents that the dark green end is connected to I/O point.

The white is the input end of probe activation and is connected to I/O output Y point of the CNC system. When Y point outputs high level, the signal receiver gives an activation signal of probe and the probe starts to work. PLC for turning on probe M70 and turning off probe M71 is shown in below Figure 3.

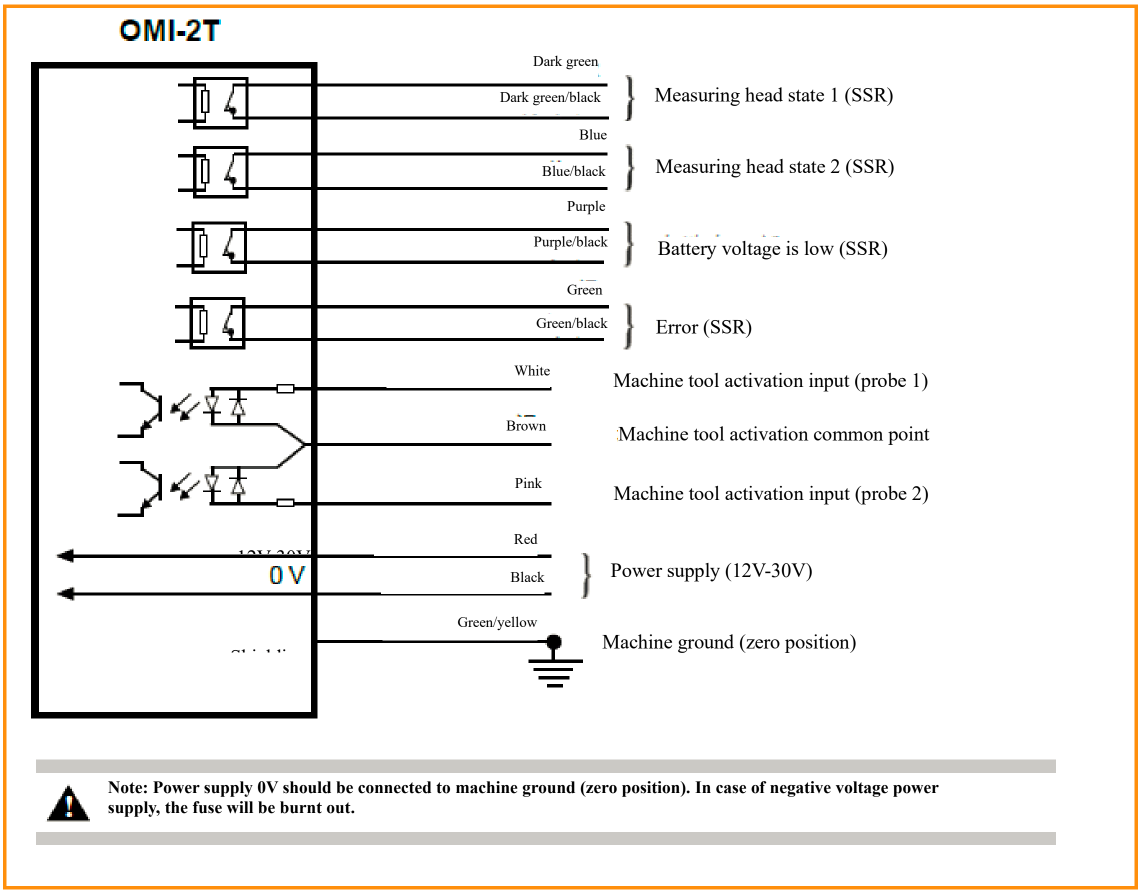


Figure 1 Electronical wiring of probe signal receiver

M70 and M71 commands are added to HNC-8 to turn on and off probe, and G31 block skip is activated. PLC is shown below.

Note: Note: PLC module of G31 block skip activation should be placed in PLC1.

Figure 2 Activation of G31 block skip command

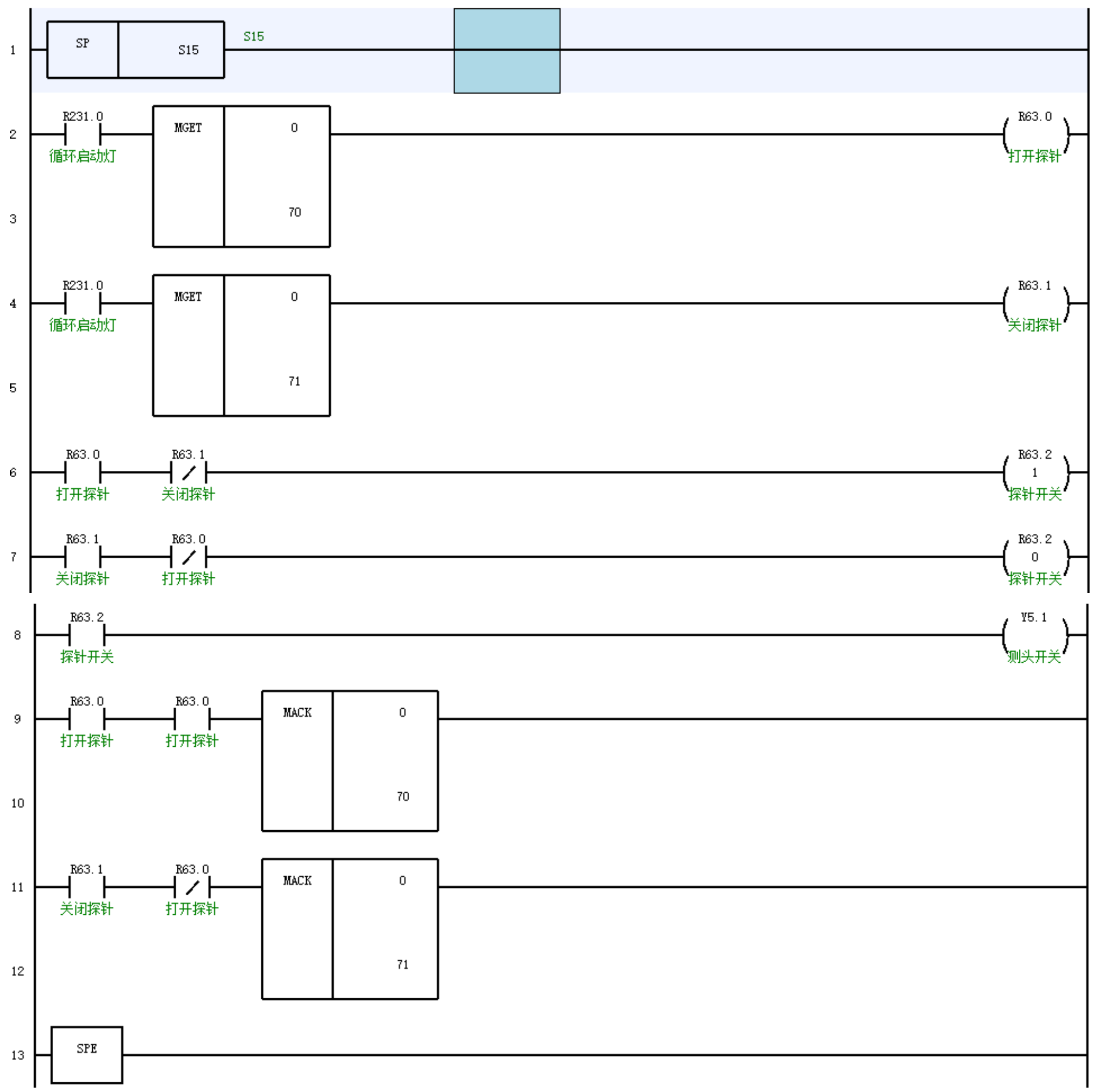


Figure 3 Turning on and off of M70 and M71 probes

G31 function test

The datum sphere is placed on C rotary table and the probe slowly approaches the surface of the datum sphere in X direction. When it is about to approach, change it to the minimum magnification. When the signal light just lights up, machine coordinate X is -228.1001. With this value as the actual machine coordinates of the collision point, use the following G code automatically and return to the latched machine coordinate value.

Test G code:

```
G54
G31 L1 G01
G54
G31 L1 G01 G90 X-229.1001 F100
```

G01 G91 X20

M30

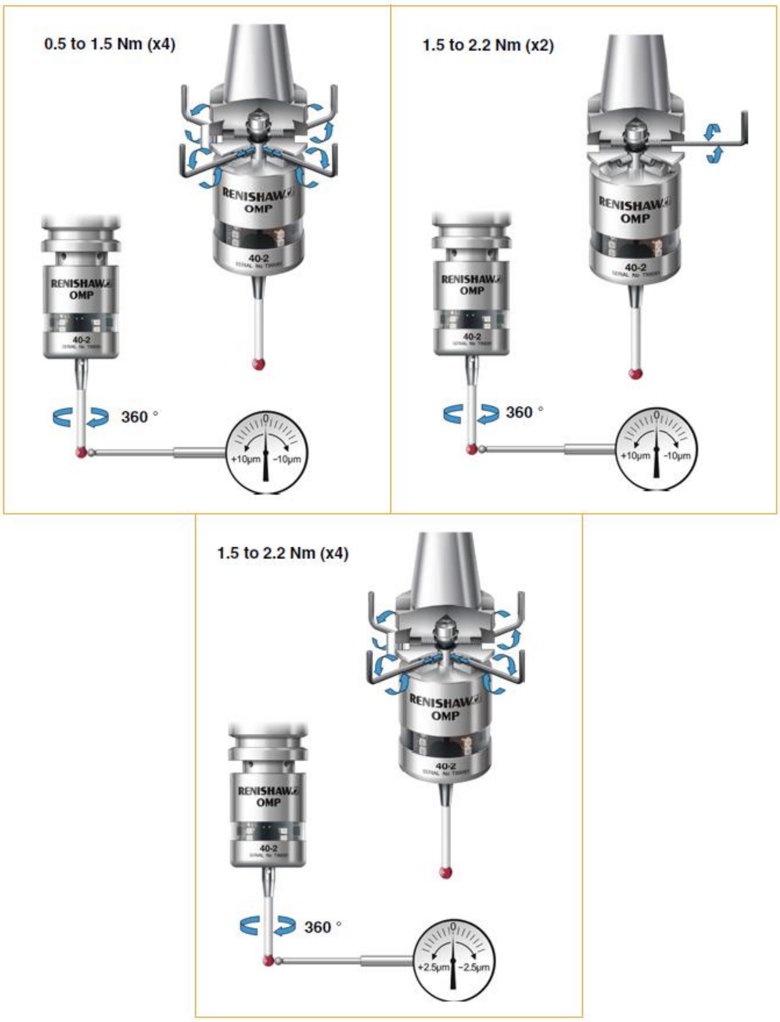
Coordinate record: -228.1031

Actual coordinates: -228.1001

Conclusion: G31 command is correct, PLC connection is correct, and working condition of probe is correct

Spindle concentricity calibration

Rotate the spindle and the dial indicator reading is within 2um



Probe calibration

Automatic Calibration Test

(1) Click on "Maintain"- "Machine calibration" to enter the automatic machine calibration interface.

Automatic machine calibration interface

The software interface includes machine coordinate area, parameter setup area, macroprogram area, and RTCP parameter area, and software function includes "Macroprogram", "Current position", "Measurement result", and "Parameter import". Areas and functions are described as below.

Area description:

a) Machine coordinate area: To display current machine coordinates and relative coordinates;

b) Parameter setup area: To input basic parameters (measurement type, rotary axis display sequence, rotary axis name, safety height, positioning speed, intermediate speed, trigger speed, datum sphere radius, tool length, and tool radius) and multiple teaching points of master axis and slave axis.

c) Macroprogram area: To display measurement macroprogram;

d) RTCP parameter area: To display measurement results of RTCP parameter.

Function description:

a) Current position: To acquire teaching points of master axis and slave axis and 5 machine coordinates, and set them in parameter table corresponding to "Parameter setup area";

b) Macroprogram: Measurement software loads corresponding measurement macroprogram based on "Measurement type" in "Basic parameters" and displays it in "Macroprogram area";

c) Measurement result: Macroprogram controls collision between probe and datum sphere and latches machine coordinates of collision point. The system fits and calculates RTCP parameters based on coordinates of collision point and displays them in "RTCP parameter area";

d) Parameter import: Input measurement results of RTCP parameters into the CNC system to take effect.

(2) Fill corresponding parameters in the calibration parameter column based on actual need

Notes to automatic calibration parameters

Parameter name	**Notes to parameters**
Calibration type	0: Dual swivel head structure 1: Dual rotary table 2: 1 swivel head and 1 rotary table
Display sequence of rotary axis	0: The second rotation axis is displayed in front 1: The first rotation axis is displayed in front
Name of rotary axis	The first rotary axis is in front of the second rotary axis
Safety height	After the probe rapidly approaches the datum sphere at positioning speed F1, it should keep a safe distance from the vertex of the datum sphere in Z direction
Positioning speed F1	Speed of the probe rapidly approaching the datum sphere
Intermediate speed F2	Speed of retracting after probe collides with datum sphere when the safety height is exceeded. It is smaller than the positioning speed
Trigger speed F3	Speed of continuing colliding with the datum sphere and accurately collecting position after retracting at intermediate speed. It is smaller than intermediate speed
Radius of datum sphere	Radius of datum sphere
Tool length	Distance from spindle end face to datum sphere center
Tool radius	Radius of probe

Tool length of probe (datum sphere is installed on the spindle and tool length refers to the distance from the datum sphere center to the spindle end face): 236.6080mm (three measurements)

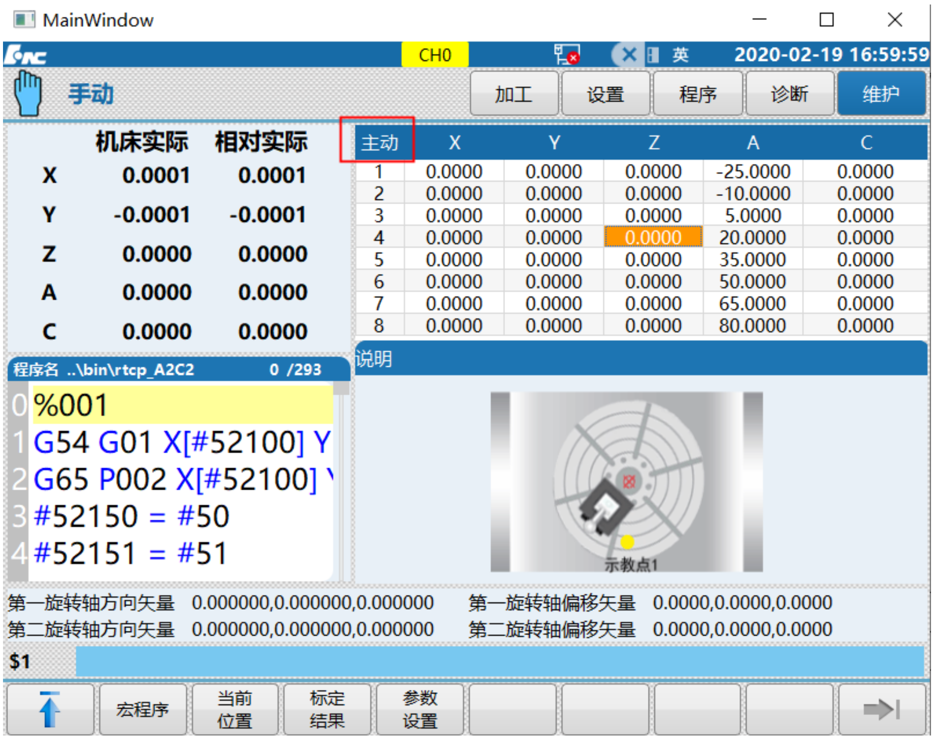
Positioning speed 2000mm/min

Intermediate speed 200mm/min

Trigger speed 50mm/min

(3) Click on PageDown to collect teaching points of the first rotary axis

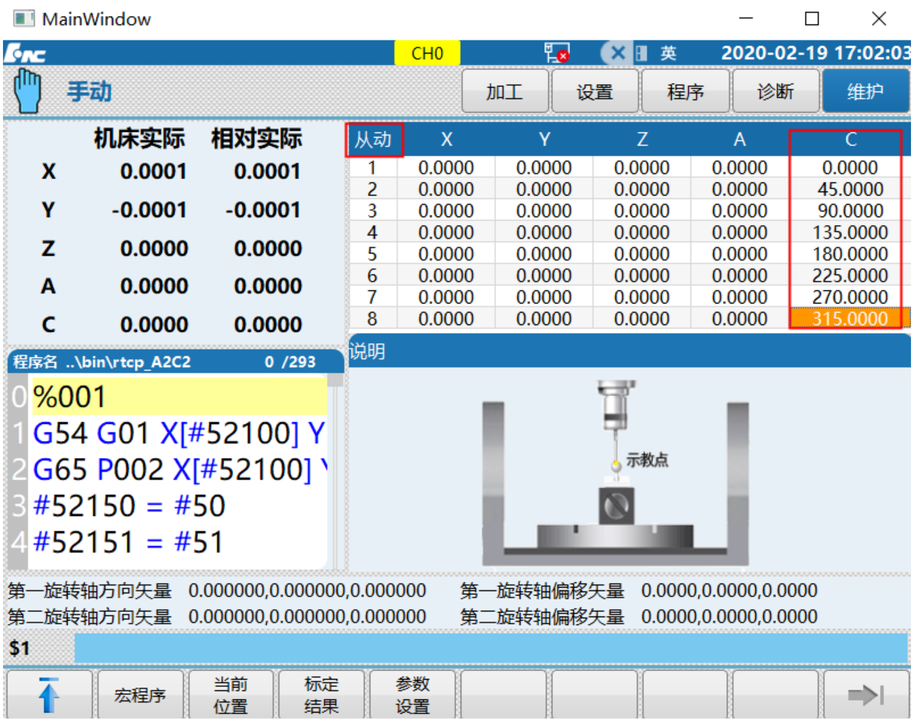
Install probe on the spindle of AC dual rotary table and place datum sphere on the rotary table. Select a point every 45° on 360° of C rotary table, and select a point every 15° for A rotary table ranging from -25° to 80° due to travel limit.



Acquisition of teaching point of the first rotary axis

(4) Click on "PageDown" to collect teaching points of the second rotary axis

Install probe on the spindle of AC dual rotary table and place datum sphere on the rotary table. Select a point every 45° on 360° of C rotary table, and take a point every 15° for A rotary table ranging from -25° to 80° due to travel limit.



Acquisition of teaching point of the second rotary axis

(5) Click on Macroprogram to generate measurement program

Generation of measurement program

(6) Press Cycle start button and the system starts measurement automatically

Generated calibration results

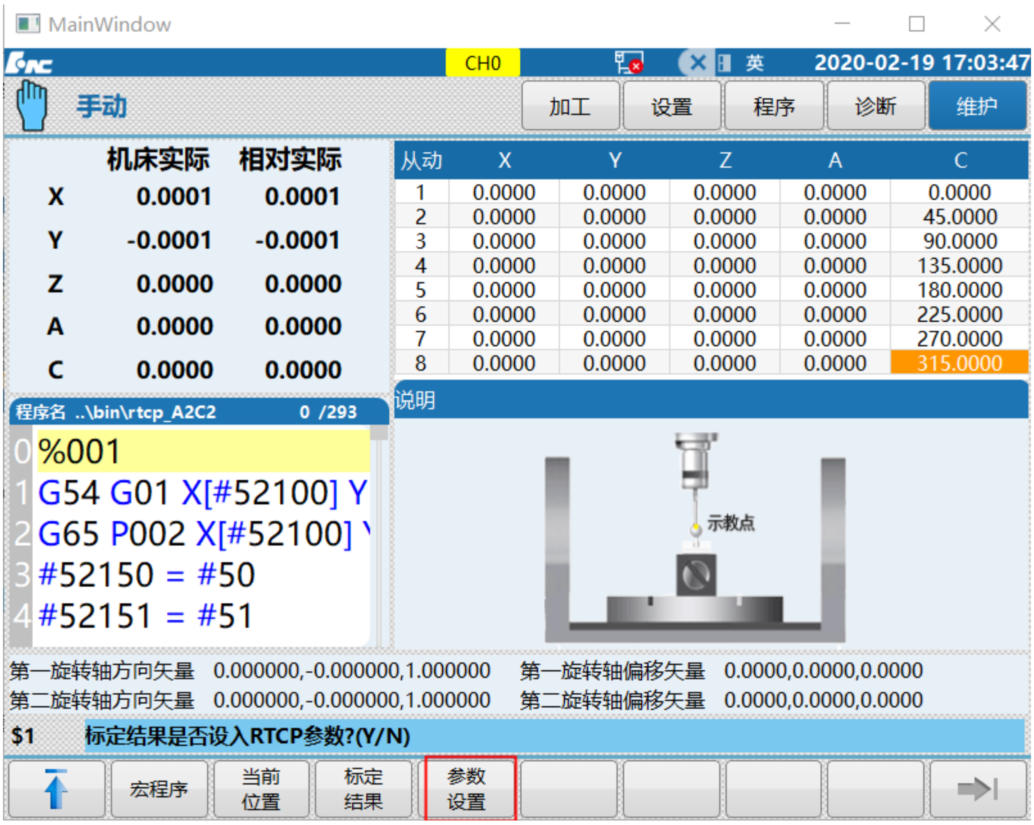
Direction vector of A axis -1.000000, 0.000000, 0.000093

Direction vector of C axis 0.000200, 0.000010, -1.000007

Offset vector of A axis -0.0000, 0.1028, -0.0000

Offset vector of C axis -200.3605, -190.1897, -516.0118

(7) After measurement, click on Parameter setup to write calibration results into system



Writing of calibration result

Users can refer to results in system parameter setting:

Direction vector of A axis -1, 0, 0

Direction vector of C axis 0.0002, 0, -1

Offset vector of A axis 0, 0.1032, 0

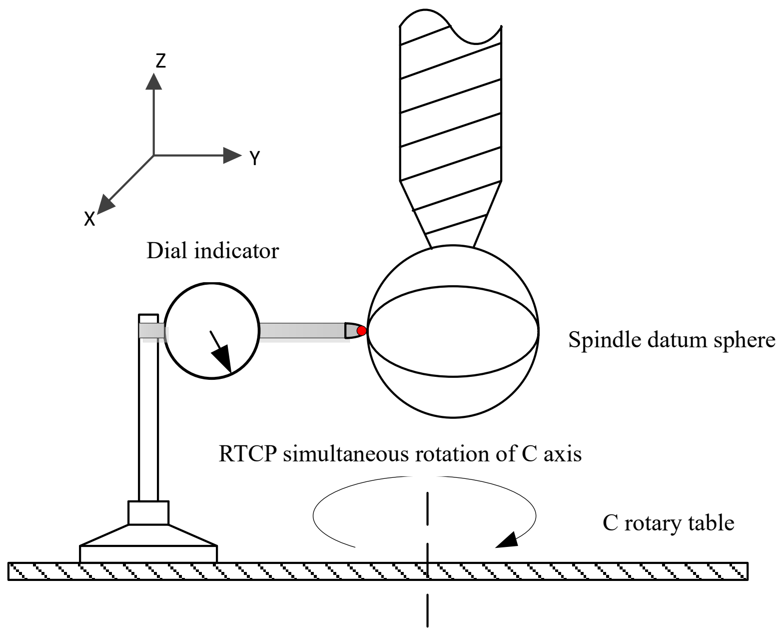
Offset vector of C axis -200.3521, -190.1937, -516.0228

(8) RTCP data accuracy test:

a) Accuracy test of C rotary table:

Press the probe of dial indicator to spin the needle to 0, and pause it for 10s every 45°. Pressing the indicator obtains plus readings, and releasing the indicator obtains minus readings.

RTCP accuracy test of C axis is tested using datum sphere and dial indicator of the spindle. Install datum sphere on the spindle and dial indicator on C rotary table. Have the indicator probe point to the equator of the datum sphere, and press it to a certain reading. C axis rotates simultaneously. Record the readings of indicator at different corners and evaluate RTCP positioning accuracy of simultaneous motion for C axis.



RTCP accuracy test of C axis for AC dual rotary table

Test G code:

G54

G43.4 H1

G01 C45

G04 X10

G01 C90

G04 X10

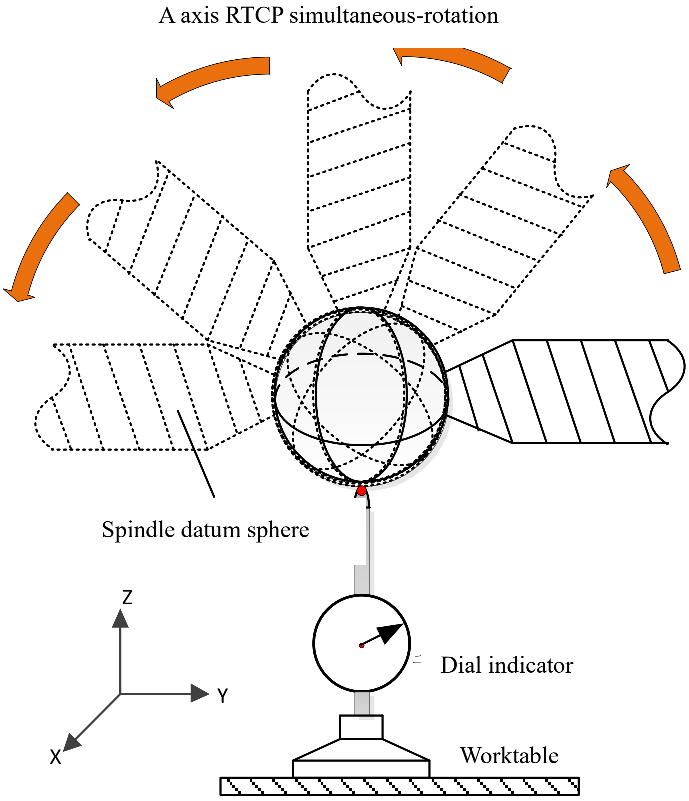
…

Test data:

Angle	0°	45°	90°	135°	180°	225°	270°	315°	360°
Reading (um)	0	-2	-10	-18	-20	-12	-14	+4	-3

b) Accuracy test of A rotary table:

Press the probe of dial indicator to spin the needle to 0, and perform the test of A axis +/- 90°. Pressing the indicator obtains plus readings, and releasing the indicator obtains minus readings.



RTCP accuracy test of A axis for AC dual rotary table

Test G code:

G54

G43.4 H1

G01 A45

G04 X10

G01 A90

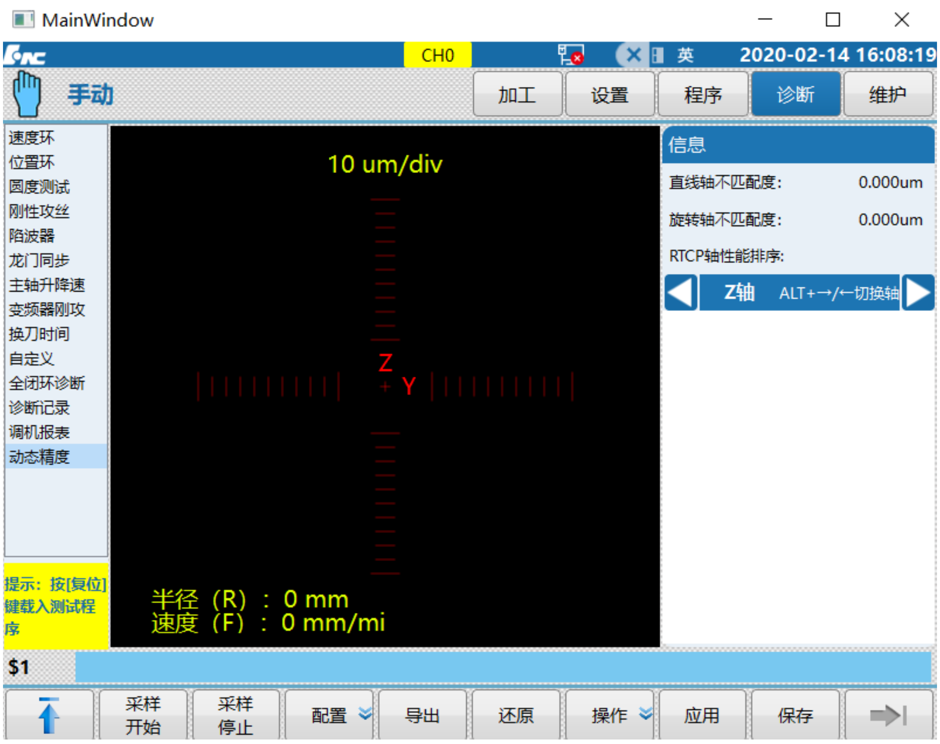
```
G04 X10
G01 A45
G04 X10
G01 A0
G04 X10
…
```

Test result:

Angle	0°	-25°	-10°	5°	20°	35°	50°	65°	80°
Reading (um)	0	-8	+8	+14	+22	+20	+22	+24	+20

Appendix E Case for 5-Axis Dynamic Precision

Before RTCP simultaneous-motion matching is measured, it is necessary to calibrate structural parameters of five-axis machine tool and test RTCP action simply. Servo commissioning tool and SSTT software of the system have the five-axis dynamic performance measurement function to measure servo matching at the time of RTCP simultaneous motion of five axes. Users can tune servo parameters based on that, and mainly tune the PA0 parameter *position proportional gain*.

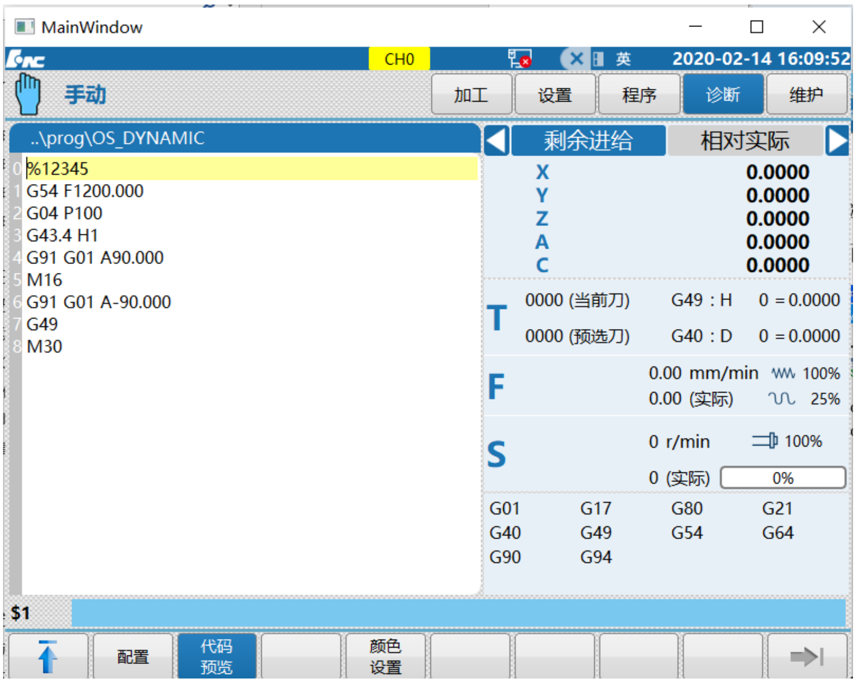


Dynamic Precision Debugging Interface

(1) The parameters include rotary axis, feedrate, tool compensation number, initial angle of rotary axis, end angle of rotary axis, and sampling period. The feedrate is recommended as 1000mm/min.

Dynamic accuracy parameter setting

(2) Manually move the rotary axis to the reference point (any machine position). Set parameters on Configuration interface, generate G code, and click the code preview to view.



Dynamic precision test G code

(3) Press cycle start, the system will automatically perform sampling and start drawing. After the

measurement program is completed, the system automatically calculates the straightness mismatch and the rotary axis mismatch. If the mismatch is within 1.5um, the requirement is met. Otherwise, users need to sort according to the servo performance of axis, and adjust the PA0 parameter based on the weakest axis