# HNC-8 System Commissioning Manual (Five-axis)

V2.4 Series

# 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.

# Contents

# 1. Joint Commissioning of Mechanical And Electrical Parameters

## 1.1. Multi-axis Matching Optimization

Whether a CNC machine tool can machine high-quality parts is closely related to the machine tool, servo, system, and process. They are interconnected and the quality of machined parts will be affected if problems exist in any link. The commissioning template is mainly intended to standardize the performance adjustment of five-axis machine tool, guide commissioning personnel to carry out template-based commissioning based on the data index, maximize movement performance of five-axis machine tool and reduce the effect of commissioning personnel's experience. Template commissioning is mainly divided into three parts: Single-axis performance test, multi-axis joint test and machining commissioning.

Note: Before the template is used, please ensure basic mechanical and electrical commissioning of machine tool has been completed, including normal operation of axes (low speed, medium speed, and high speed), positioning accuracy and repeated positioning accuracy meeting the requirements, laser screw pitch error compensation, backlash compensation, normal MCP function and normal operation of magazine. The Commissioning Manual is mainly intended for five-axis CNC machine tool. It can also be referred to for commissioning of three-axis, four-axis, or other multi-axis CNC machine tools.

### 1.1.1. Single Axis Performance Test

The test is intended to optimize servo drive parameters and improve servo rigidity and performance by built-in servo commissioning function and SSTT of system. In principle, when the conditions are met, try to exploit the role of servo drive. From the point of field application, the better servo drive performance is, the higher quality of processed parts is.

#### A. Position loop

Position loop reflects tracking error during actual machining and affects the accuracy of finished parts.

(1) Continuously improve position proportional gain. If vibration noise occurs during axis moving, slightly decrease the parameter value until the noise disappears during axis movement.

(2) Enter the servo commissioning interface, select position loop commissioning, set the travel of axis (machining area), move the axis at 1000mm/min and limit the tracking error displayed to below 0.2mm, 0.1mm at best. If the requirements are not met, decrease position proportional gain.

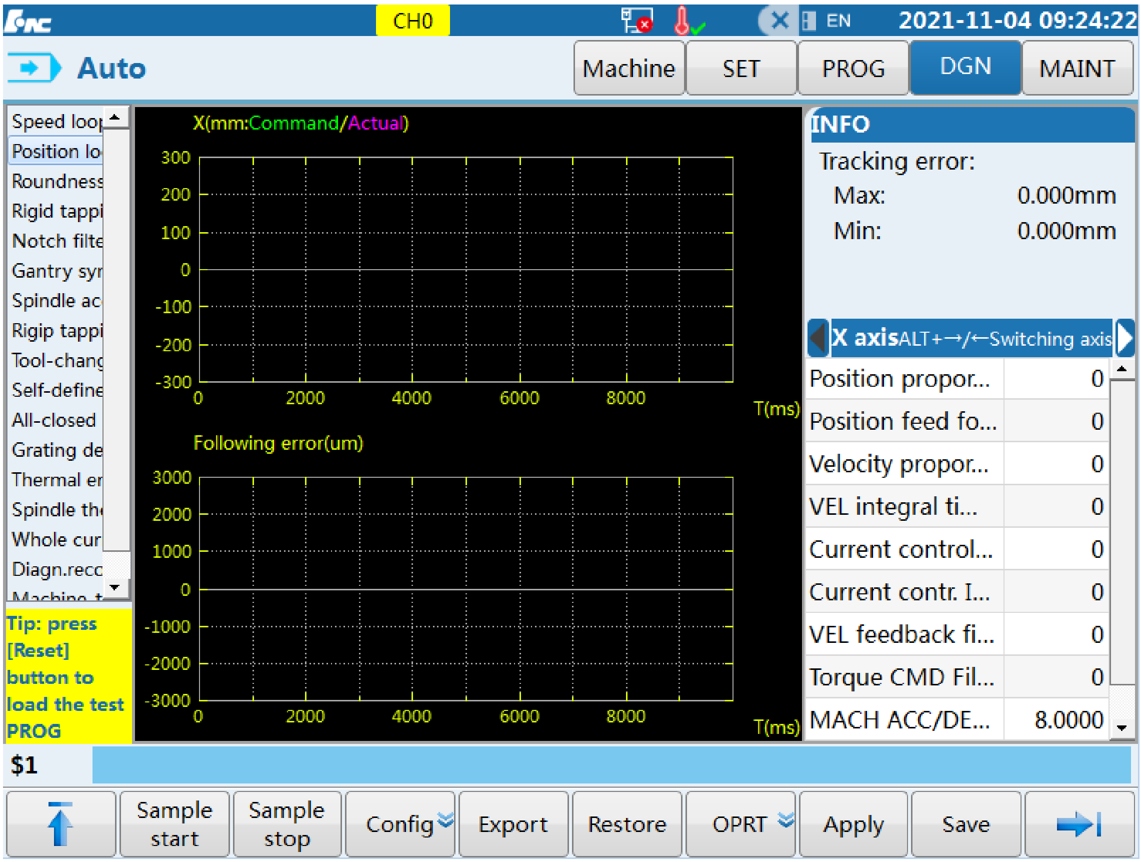



Figure 1-1 Position loop test

## B. Speed loop

Speed loop reflects speed fluctuation during actual machining, namely tracking error fluctuation. Common vertical grain and chatter mark incurred during machining are caused by great speed fluctuation. The principle of speed loop commissioning is to improve rigidity of servo drive and limit the speed fluctuation to a reasonable range.

(1) Improve speed proportional gain. If noise occurs when an axis moves, slightly decrease the parameter value until the noise disappears during axis movement.

(2) Enable function of notch filter to adjust the trap frequency of axis. For adjustment method, refer to relevant commissioning documents. Further improve rigidity of speed loop. If vibration occurs, adjust the parameters [Speed loop feedback filtering factor] and [Torque command filter time]. Try not to use the second notch filter, but if the expected effect cannot be achieved, users could use the second notch filter.

(3) Enter the servo commissioning interface, select speed loop commissioning, set the travel of axis (machining area), and set the movement speed as the specified speed of finish machining F. 3000mm/min is the test speed by default. Speed fluctuation should be 10mm/min, and maximum value and minimum value of acceleration should be the same. If there is a big difference, it means that the machine is stressed unevenly in two directions and needs to be adjusted properly. If the requirements are not met, decrease speed loop proportional gain.

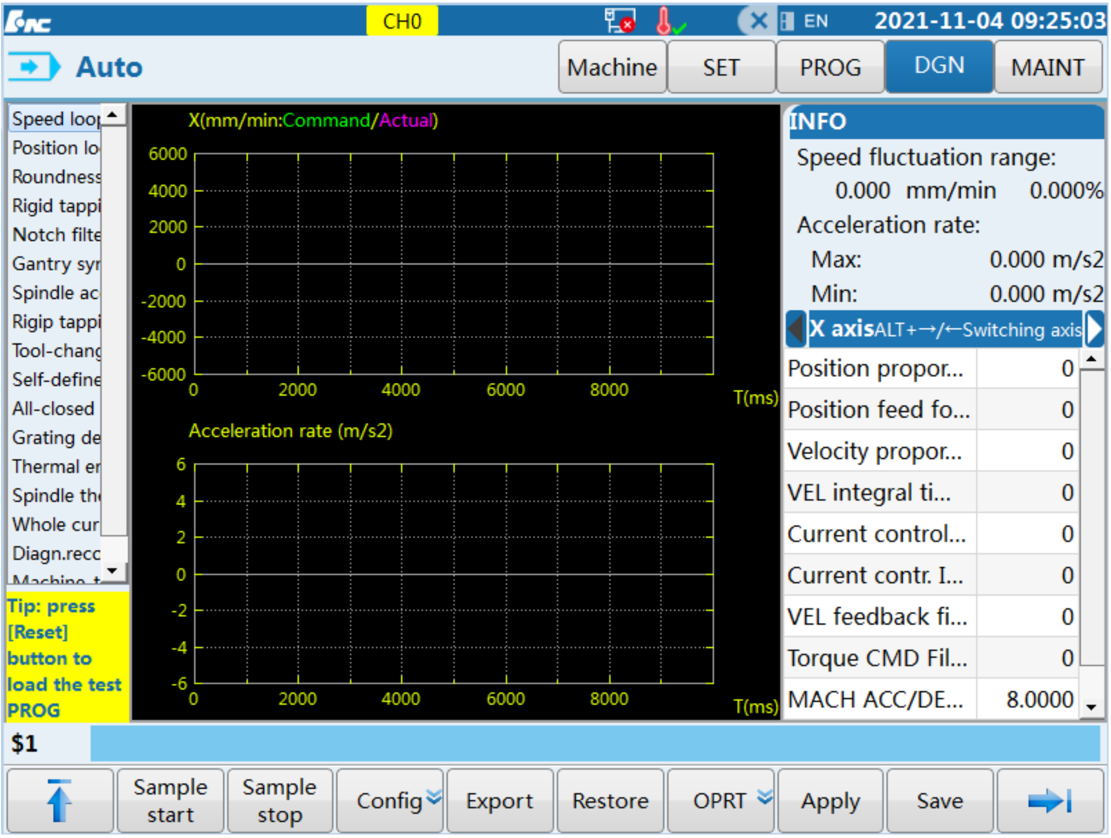



Figure 1-2 Speed loop test

## C. Current loop

PA27 servo current loop gain of hardware current loop is 6000, and PA27 of common servo current loop is often 3000-4000.

## D. Overshoot test (low speed and high speed)

According to previous commissioning experience, there is an overshoot of the actual speed of gravity axis, bearing axis and hard guide axis at the end of acceleration or deceleration, which is especially obvious under full closed-loop control mode. The key of overshoot test is to focus on the speed fluctuation in the end of acceleration or deceleration, and the reverse performance of axis is optimized by adjusting speed loop parameters and acceleration/deceleration parameters of axis. To observe speed fluctuation curve easily, users can use SSTT tool to collect command speed and actual speed waveforms of tested axis.

(1) Set the travel (machining area) of axis, perform the reciprocating motion, and select low speed, medium speed, and high speed for test.

(2) Observe waveform features of actual velocity curve in this area after sampling and calculate maximum speed fluctuation of wavepeak and wavevalley.

(3) Fluctuation should be below 40mm/min. Otherwise, decrease speed loop gain and increase acceleration time constant and jerk time constant of axis.

**Note:**

a) The overshoot test speed includes F100, F200, F500, F1000 and F3000 for users' reference. Users can adjust the selected speed as required, but the speed should cover low speed, medium speed and high speed.

b) Under full-closed loop control mode, especially at low speed, it is hard to adjust speed fluctuation to the ideal range. At this time, the measurement standard can be relaxed, and 100mm/min cannot be exceeded at most.

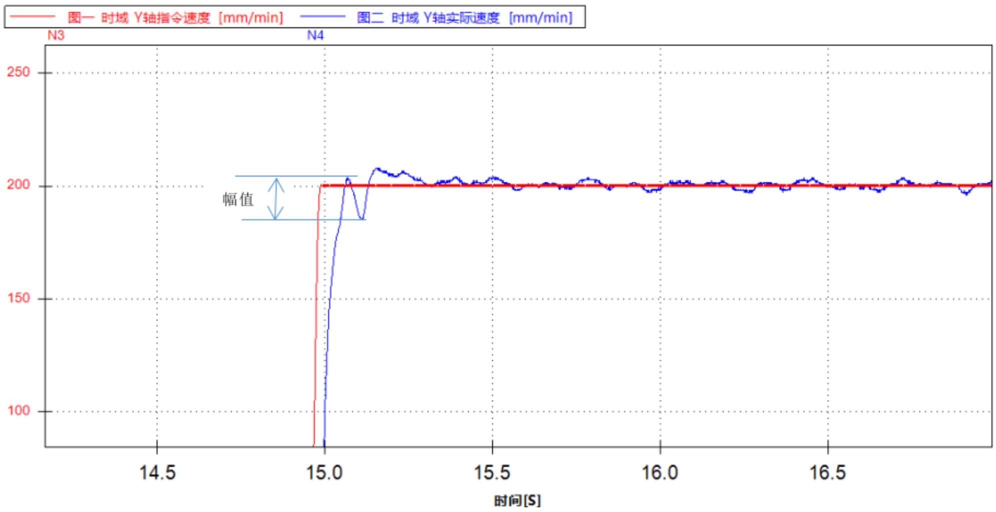


Figure 1-3 Overshoot test

### 1.1.2. Debugging of Simultaneous Multiaxis Motion

Under simultaneous multi-axis control mode, servo matching between axes determines contour accuracy and surface quality of processed parts. In this test, the servo matching between axes is performed through roundness measurement function of system and RTCP simultaneous-motion measurement function. The roundness measurement is intended for servo matching between 2 axes and to measure over quadrant jump error. The RTCP simultaneous-motion measurement is intended for servo matching between RTCP five axes.

Note: If servo matching measurement function of the system cannot satisfy multiaxis matching, based on previous commissioning experience, it is just required to ensure consistency of position loop parameter PA0 of axes.

#### A. Roundness measurement

With roundness measurement function, the system can measure servo matching among X, Y and Z feed axes and over quadrant jump error. Besides, SSTT software has the roundness measurement function of linear axis and rotary axis. If conditions permit, try to measure roundness using ballbar. Users can measure roundness by roundness measurement function firstly and then remeasure it using ballbar.

Recommended: Roundness radius is 100 and F speed is 1000mm/min.

(1) The reference range of servo mismatch between two axes is 【-1.0,1.0】. If the range is exceeded, tune the PA0 parameter.

(2) Measure the backlash with dial indicator and fill the compensation value in the parameter list. This step may be ignored if bi-directional pitch error compensation is used.

(3) Measure over quadrant jump error. Disable the backlash compensation, measure reverse jump, delay time and acceleration/deceleration time using built-in servo commissioning tool of HNC-8 system and fill the data in the compensation table, and then perform the remeasurement using ballbar. If the requirements are not met, correct the compensation data. Generally, the jump value 0.01mm can meet the requirements. There are stricter requirements for small machine tool (drilling machine, high gloss polishing machine) and the jump value should reach the micron level.

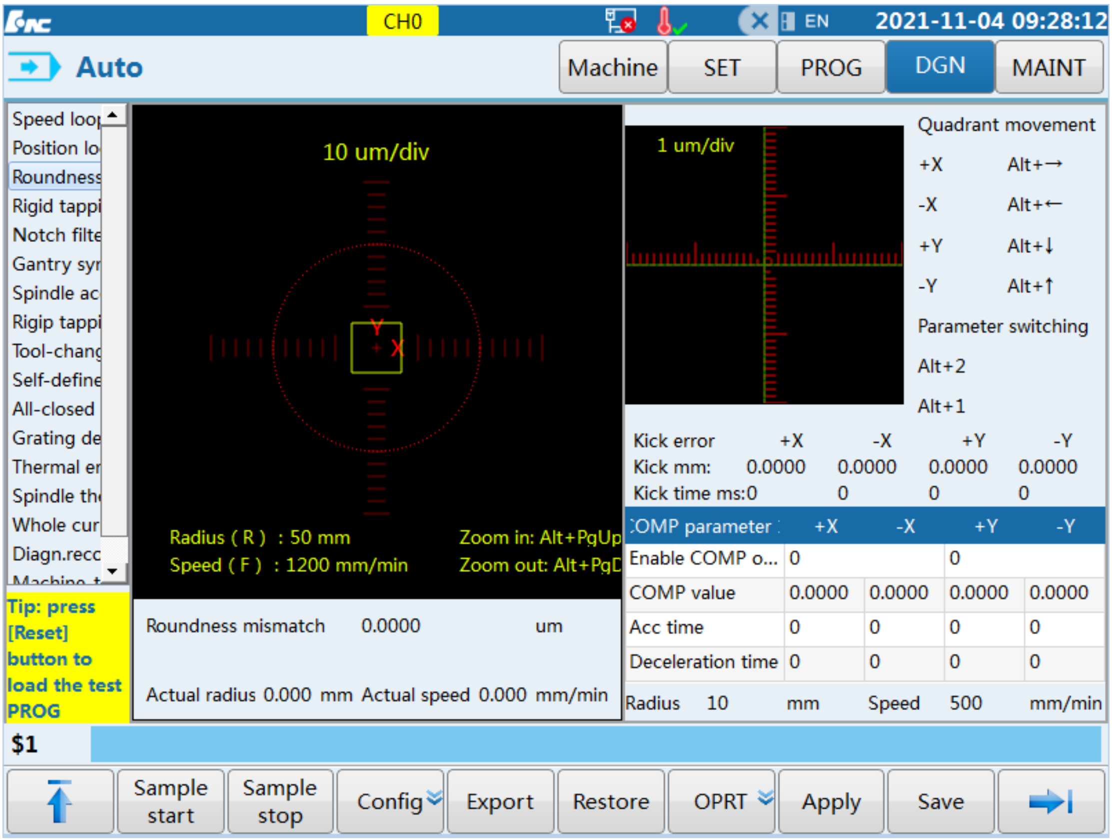



Figure 1-4 Roundness measurement

### B. RTCP simultaneous-motion measurement

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*.

(1) The parameters include rotary axis, feedrate, tool compensation number, initial angle of rotary axis, end angle of rotary axis, and coordinates of reference point. The feedrate is recommended as 1000mm/min, the reference point coordinates should be the coordinates of the initial point of machine tool before RTCP simultaneous-motion. There are two options: Current tool position and custom reference point.

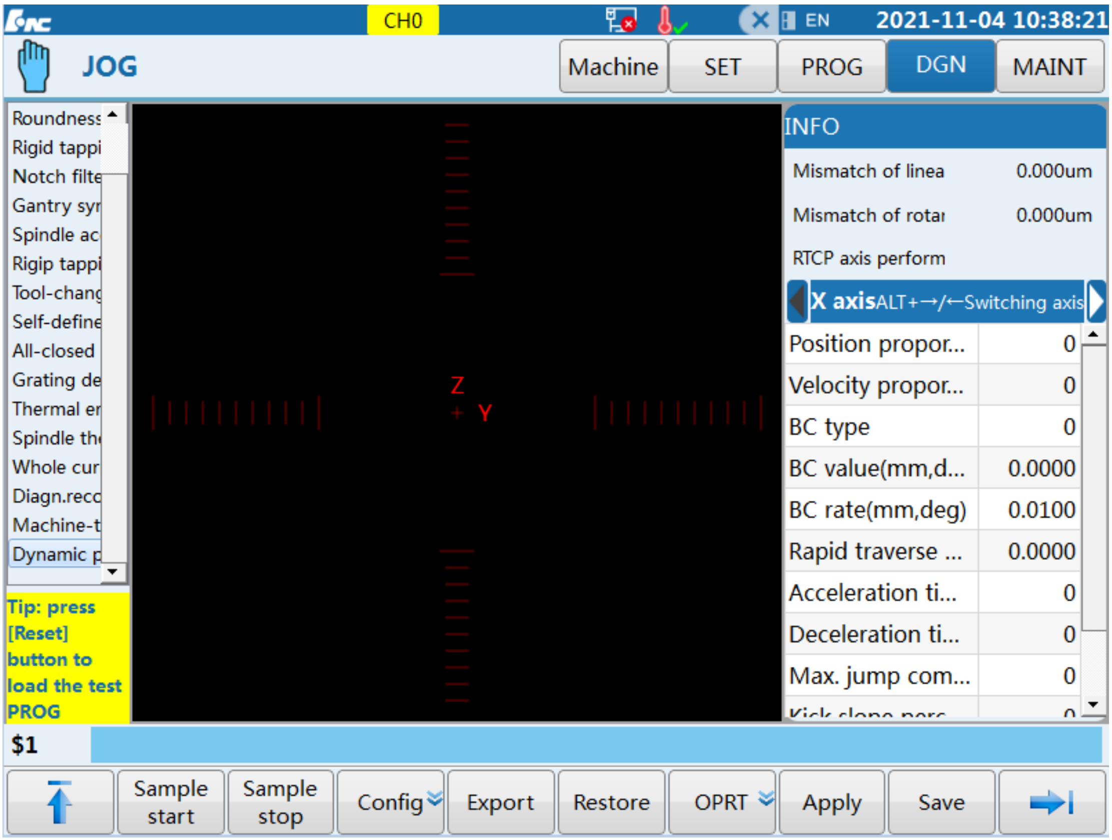



Figure 1-5 Dynamic precision test

(2) After the parameters are set, press【Confirm to test G code】, and the program of G code measurement will be generated automatically.

```
%12345;
G54 X30.71277 Y131.08677 Z195.01965 F3000
G01 C0
G43.4 H1
M00
G91 C360
M00
G90G01 C0
M30
```

(3) Press cycle start and enable sampling function when the first M00 is executed. Then press cycle start.

(4) Straightness mismatch and rotary axis mismatch will be obtained after measurement. The mismatch should be limited to 1um. Otherwise, tune PA0 parameter with the weakest axis as the basis according to servo table.

## 1.2. Spatial Error Compensation

Spatial error of machine tool refers to initial spatial position error and movement error caused by inaccurate control of manufacturing, installation and motion and tool, machine bed, thermal deformation as well as other factors. Geometric error and error arising from thermal deformation account for about 53% of total error of machine tool, so spatial error compensation of CNC machine tool is of great significance for improving machining precision.

In view of the current research status in the field of three-dimensional error compensation of CNC machine tools, 3D spatial error measurement, modeling and compensation technology of CNC machine tool is studied through theoretical research, computer simulation and experimental research, offering solutions for improving machine performance and machining accuracy.

Geometric error of CNC machine tool includes linear displacement error, straightness error, perpendicularity error, and angle error, which determine the precision performance of CNC machine tool. For CNC machine tool containing three mutually vertical linear axes, suppose error occurs repeatedly and can be measured, users can use the software for spatial error compensation, and then overall machining precision of machine tool can be greatly improved.

Movement of machine axis is a geometric motion, which is described using 6 degrees of freedom. There are 6 error factors including 1 linear positioning error, 2 straightness errors and 3 angular displacement errors. Thus, for three-axis machine tool, there will be 21 geometric errors (including 3 perpendicularity errors), which are collectively referred to as spatial geometric error or volumetric error of machine tool. If these three linear feed axes are represented by X axis, Y axis and Z axis respectively, these 21 errors can be expressed as follows:

(1) Linear displacement error: Dx(x), Dy(y), and Dz(z)

(2) Straightness error in the vertical plane: Dy(x), Dx(y), and Dx(z)

(3) Linearity error in the horizontal plane: Dz(x), Dz(y), and Dy(z)

(4) Roll angle error: Ax(x), Ay(y), and Az(z)

(5) Pitch angle error: Ay(x), Ax(y), and Ax(z)

(6) Swing angle error: Az(x), Az(y), and Ay(z)

(7) Perpendicularity error: Φxy, Φyz, and Φxz

D represents linear error, A angle error, Φ perpendicularity error. The subscript indicates the direction of error, and the bracketed content is the motion axis that produces error.

In order to improve machining precision of CNC machine tool, the CNC system must have error compensation function. e.g.: manufacturing tolerance of screw rod of machine tool will cause the pitch error, and linear displacement error compensation will be needed. For large gantry milling machine, due to a large beam span which is easily bent and deformed under the influence of gravity, the deflection compensation (straightness compensation) for the beam needs to be performed.

Presently, the simpler spatial error measurement method for machine tool is based on 21-geometric-error measurement compensation method, in which 21 geometric errors of machine tool can be accurately measured by 6D laser interferometer (three linear displacement errors, six straightness errors, three perpendicularity errors, and nine angle errors). Compensation of these 21 errors can meet the precison requirements of most CNC machine tools.

With motion of Y axis as the example, in actual movement process Y axis will generate spatial errors in 6 degrees of freedom, including three translation errors: Linearity (positioning) error of Y axis in Y direction $\delta_y(y)$, straightness error of Y axis in X direction $\delta_x(y)$, and straightness error of Y axis in Z direction $\delta_z(y)$; three corner errors: Elevation error caused by rotation around axis X $\varepsilon_x(y)$, roll angle error caused by rotation around axis Y $\varepsilon_y(y)$, and swing angle error $\varepsilon_z(y)$ caused by rotation around Z axis. In the above error symbols, letter y in the bracket means that the error is caused by motion of Y axis, and the error value is related to position of Y axis; superscript of translation error is represented by $\delta$, subscript is represented the direction of error, superscript of corner error is represented by $\varepsilon$, and subscript represents center rotation axis. Y axis error element is shown in Fig. 1-6:

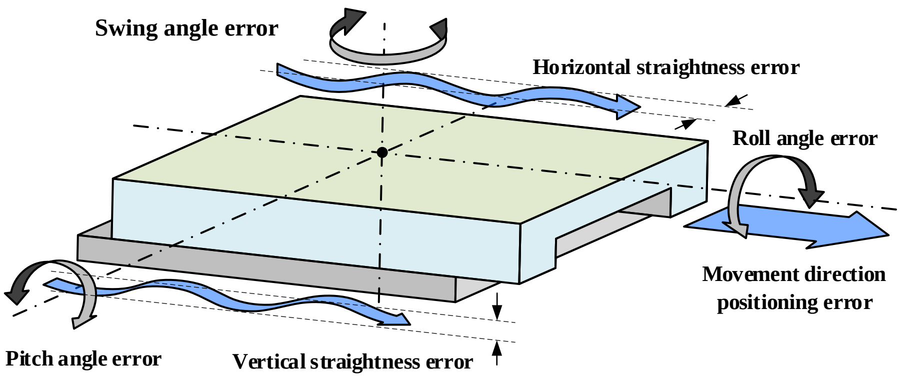



Figure 1-6 Y axis error element

### 1.2.1. Spatial Geometric Compensation Principle

### A. Linear displacement error compensation

Linear displacement error is mainly caused by pitch error of screw rod of machine tool, error value is related to current position of axis, and motion axis and compensation axis are the same axis. In order to achieve linear displacement error compensation, compensation value sequence must be built based on the error curve, and compensation value sequence is obtained according to error curve sampling which is a collection of compensation values at uniformly-spaced points within the travel range.

After compensation value sequence is obtained, compensation values of current motion axis at positions is calculated using linear interpolation, as shown in Fig. 1-7.

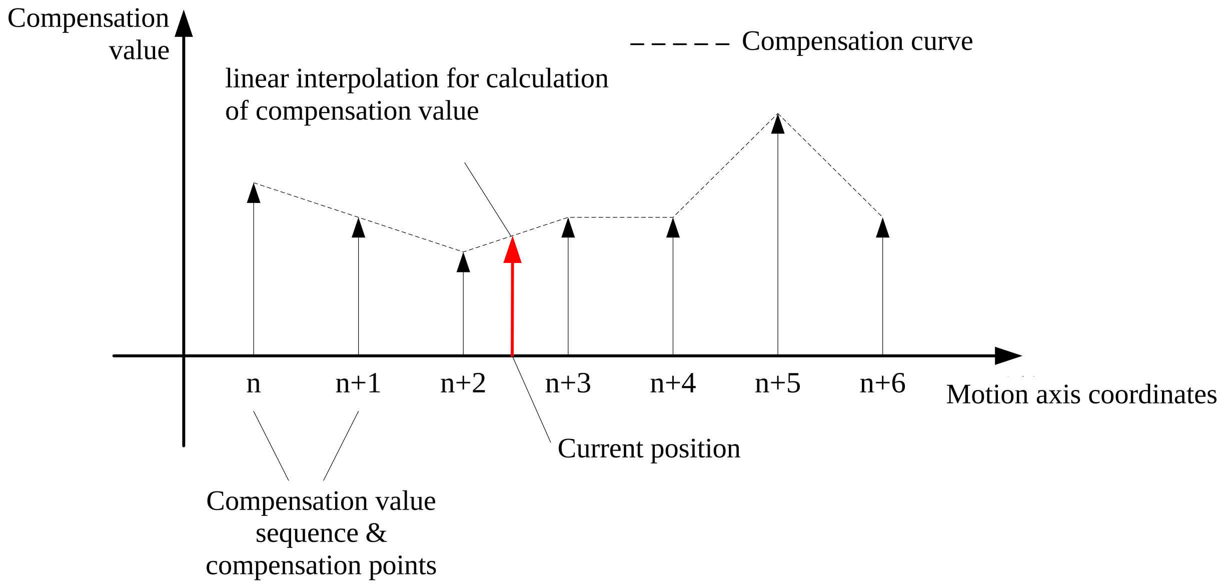



Figure 1-7 Calculation of current position compensation value using linear interpolation

During compensation, compensation value will be superposed on command coordinates of current motion axis. Thus, current axis will move in positive direction when compensation value is positive.

### B. Straightness error compensation

Straightness error compensation is similar to linear displacement error compensation. The difference is that motion axis of straightness error compensation and compensation axis are not the same axis. Compensation value calculated according to position of current motion axis will be superposed on command coordinates of specified compensation axis.

### C. Perpendicularity error compensation

Perpendicularity error is shown in Fig.1-8:

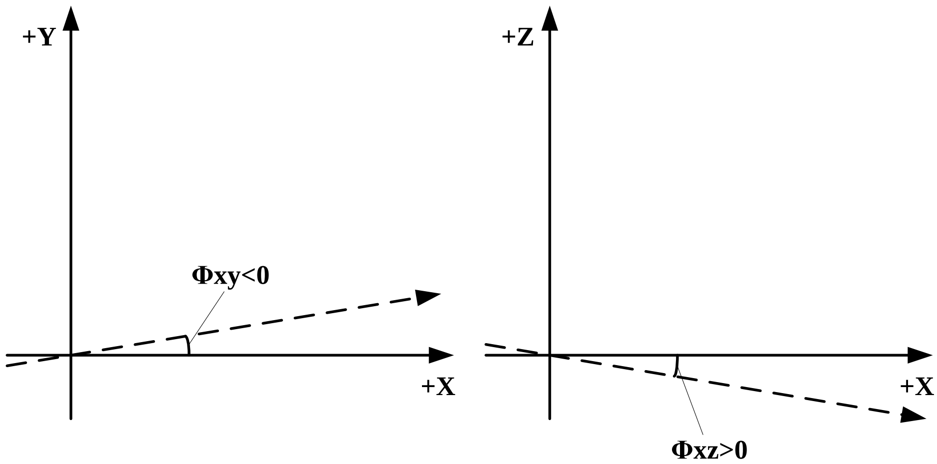



Figure 1-8 Perpendicularity error

It can be seen in the figure that perpendicularity error is positive when the positive inclined angle between two axes is greater than 90° and negative when the inclined angle is less than 90°. Because perpendicularity error is unrelated to motion axis of machine tool, it is unnecessary to build compensation value sequence. Machine displacement compensation amount specific to perpendicularity error is figured out by the below formula:

Compensation value=Coordinates of motion axis * perpendicularity error

Suppose perpendicularity error of X axis and Y axis of machine tool is Φxy, when Y axis is the compensation axis, compensation value $Comp_y = x * \Phi xy$. And x is the coordinates of current position of motion axis X.

## 1.2.2. Introduction to HNC-8 Spatial Error Compensation Module

### A. Interpretation of spatial geometric error compensation parameters

Error compensation is used to describe parameter set of geometric error of machine tool, stored in user memory area of the CNC system (the CNC system allocates storage area of error compensation table in advance), and offers necessary interfaces for users to input compensation data. Besides, compensation algorithm is used to calculate displacement compensation amount of machine tool axes through querying error compensation table.

(1) Linear displacement error and straightness error compensation table

In geometric errors of machine tool, both linear displacement error and straightness error need to establish a sequence of compensation values to be accurately described. Thus, compensation tables of the two types of errors are shown in Fig. 1-9, which are error compensation tables containing compensation value sequence.

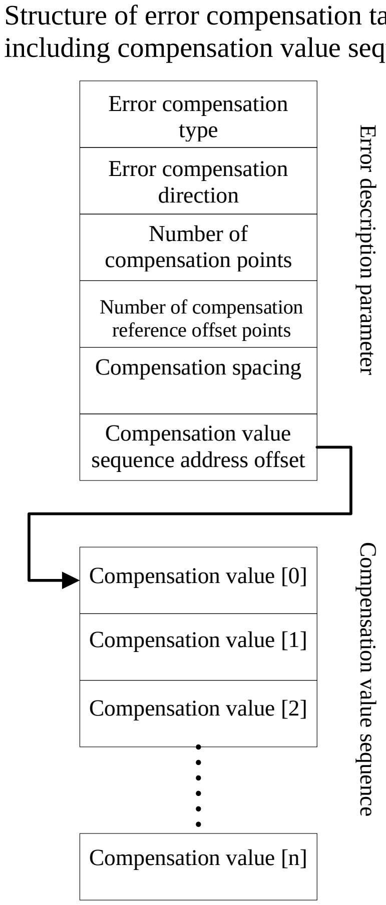



Figure 1-9 Error compensation table containing compensation value sequence

Linear displacement error and straightness error compensation parameters include the two parts error description parameter and compensation value sequence. The storage address of the two kinds of parameters in memory is not continuous, and users need to set the offset of compensation value sequence address to associate the current error compensation table with the compensation value sequence. Meaning of parameters in the table is shown in Table 2-1:

Table 1-1 Meaning of parameters in error compensation table

| Parameter | Meaning |
|---|---|
| Error compensation type | 0: There is no compensation;<br>1: X_Y; 2: X_Z;<br>3: Y_X; ; 4: Y_Z;<br>5: Z_X; 6: Z_Y;<br>7: X_X; 8: Y_Y ; 9: Z_Z; |
| Compensation direction | 1: Unidirectional compensation;<br>2: Bidirectional compensation;<br>3: Unidirectional modulus compensation (used for compensation of rotary axis, expandable);<br>4: Bidirectional modulus compensation (used for compensation of rotary axis, expandable). |
| Number of compensation points | Value range: 0-2147483647<br>Description: The number of compensation points determines the size of compensation |

| | |
|---|---|
| | value sequence (for bidirectional compensation, the number of compensation values stored in compensation value sequence will be twice the number of compensation points).<br>Note: Because the CNC system allocates a limited memory to error compensation table (determined by the system parameter *Maximum number of spatial error compensation values VOL_COMP_VALUE_MAX_NUM*), the memory capacity limit must be considered while selecting number of compensation points, and error will be reported if the number of compensation values exceeds the limit. |
| Number of compensation reference offset points | Unit: Number of offset points.<br>Value range: -2147483648 to 2147483647<br>Description: The parameter is used to determine compensation travel of current motion axis.<br>Compensation travel in negative direction = - (Number of offset points * Compensation point spacing)<br>Compensation travel in positive direction = (Number of compensation points-Number offset points-1) * Compensation point spacing<br>For example, when the compensation point spacing is 60mm, the number of compensation points is 20, and the number of offset points of reference point is 3, the compensation travel in negative direction is -180mm and the compensation travel in positive direction is 960mm. Thus, sequence of compensation points: -180, -120, -60, 0, 60, 120,…,900 and 960. |
| Compensation point spacing | Unit: For moiton axis, unit is μm.<br>Value range: 0-2147483647 |
| Compensation value sequence | Unit: For motion axis, unit is μm.<br>Value range: 2147483648 to 2147483647<br>Description: Compensation value sequence should be arranged according to the order of corresponding compensation point coordinates (from small to large). In case of bidirectional compensation, input positive compensation data and negative compensation data successively.<br>For example, when the number of compensation points is set as 10 and bilateral compensation is adopted, the first 10 in compensation value sequence are positive and the last 10 are negative compensation values.<br>On the condition that modulus compensation function is not enabled, when the axis position of machine tool is out of the compensation travel range, take the compensation value at boundary point as the compensation value of current position. |

(2) Perpendicularity error compensation table

The perpendicularity error compensation table needs not to build compensation value sequence, and the structure is shown in Table 1-2

Table 1-2 Perpendicularity error compensation table

| **Parameter** | **Value range** |
|---|---|
| Error compensation type | 0 to 6 |
| Perpendicularity error | -2147483648 to 2147483647(0.001°) |

Perpendicularity error compensation table describes the perpendicularity relations among different axes. Meaning of parameters in the table is shown in Table 1-3.

Table 1-3 Meaning of parameters of perpendicularity error compensation

| Parameter | Meaning |
|---|---|
| Error compensation type | Compensation type is determined by motion axis (error generating axis) and compensation axis. The meaning of value is shown below:<br>0: There is no perpendicularity error compensation;<br>1: X-Y; 2: X-Z; 3: Y-X; 4: Y-Z; 5: Z-X; 6: Z-Y.<br>(For the above, the first item represents motion axis, and the second item represents compensation axis) |
| Perpendicularity error | Used to calculate perpendicularity error compensation value |

## B. Processing flow of spatial geometric error compensation module

The processing flow of the CNC system to compensate the geometric error of the machine tool is shown in Figure 1-10

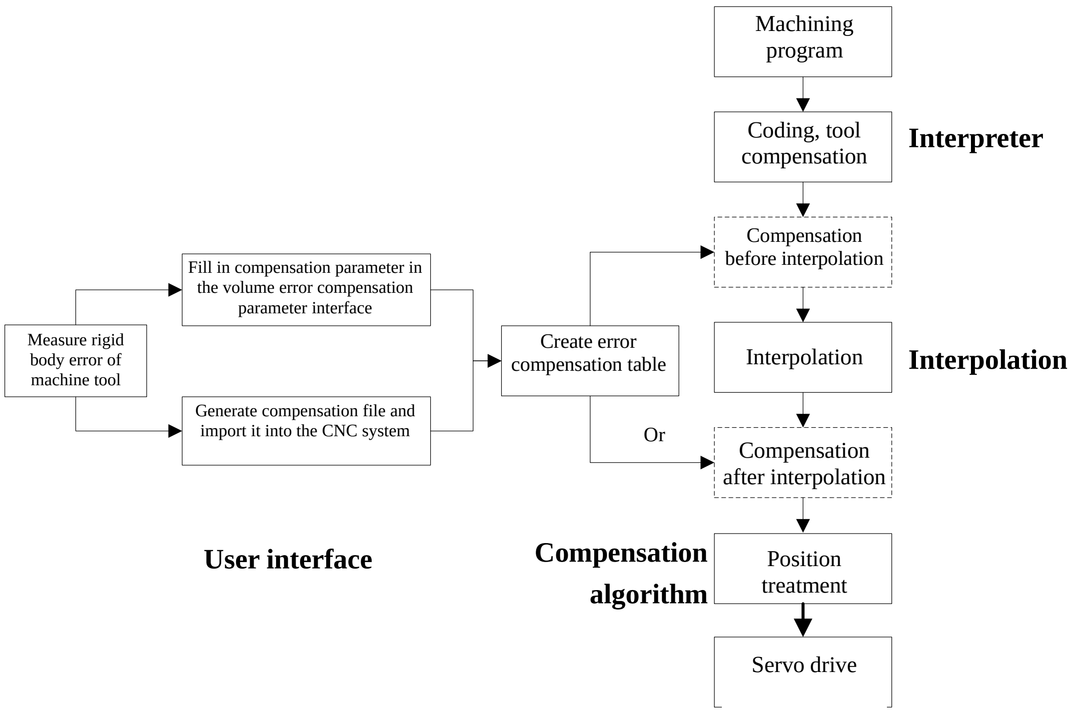



Figure 1-10 Processing flow of spatial error compensation module

It can be seen from Fig. 2-5 that error compensation module consists of the following three parts:

(1) Input of compensation parameters of CNC system. As a user interface of error compensation module, compensation parameters of the CNC system can be inputted in two ways. The first is to directly fill error compensation parameters in corresponding parameter items in the spatial error compensation interface. The second is to generate compensation files in the format specified by the CNC system and import them into the CNC system.

(2) Establishment of error compensation table. Error compensation table describes compensation type, compensation value sequence, and other information of errors in detail. The compensation value of

corresponding position is obtained with compensation algorithm through querying error compensation table.
(3) Compensation algorithm. The compensation algorithm determines the compensation value by querying the error compensation table, and performs compensation before interpolation (or compensation after interpolation).
Manual processing that must be contained in the working process of error compensation module includes: (1) Input of spatial error compensation parameters of CNC system; (2) Correct configuration of three axes for spatial error compensation (X, Y, Z axis perpendicular to each other).
HNC8 series completes the compensation interface of 21 geometric errors of machine tool, through which 21 geometric errors (three linear displacement errors, six straightness errors, three perpendicularity errors, and nine angle errors) of machine tool can be quickly compensated. Compensation of these 21 errors can satisfy the accuracy requirements of most CNC machine tools.
The software operation interface is an interaction platform between user and machine tool software. After module and function analysis of software, a reasonable operation interface is designed. The software interface design follows the principle of standardization, reasonability, aesthetics, and usability. According to software compensation function, the designed HMI is shown below:

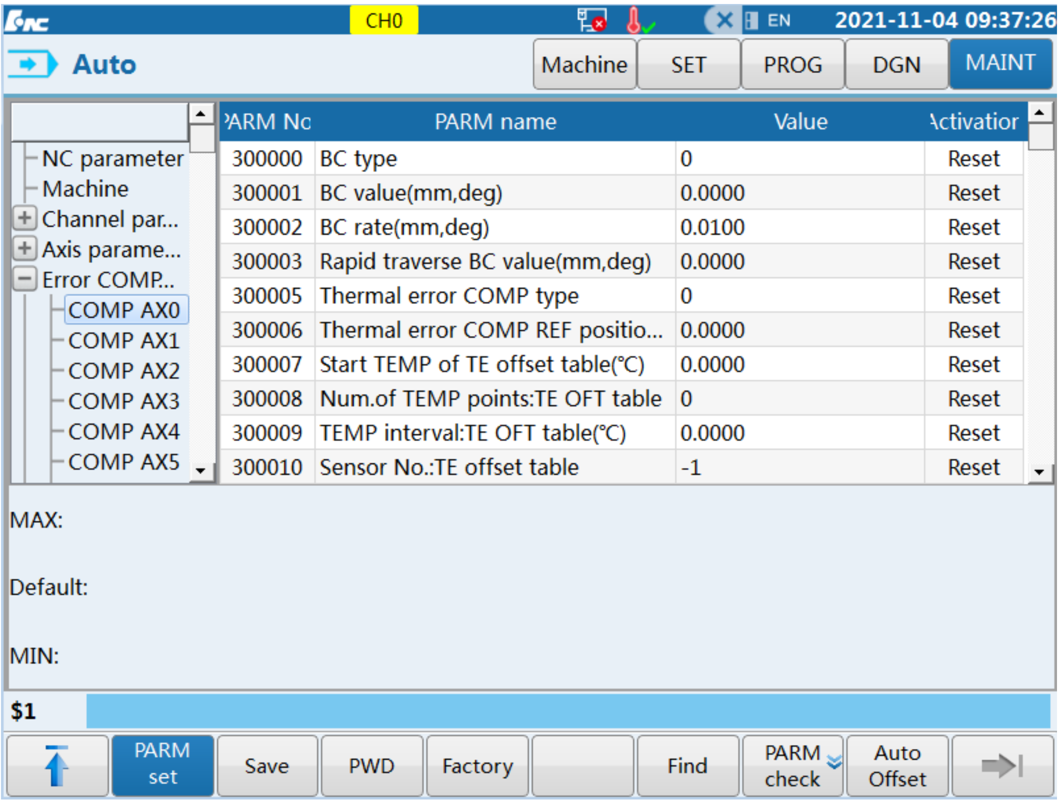


Figure 1-11 21 spatial errors--backlash compensation function interface

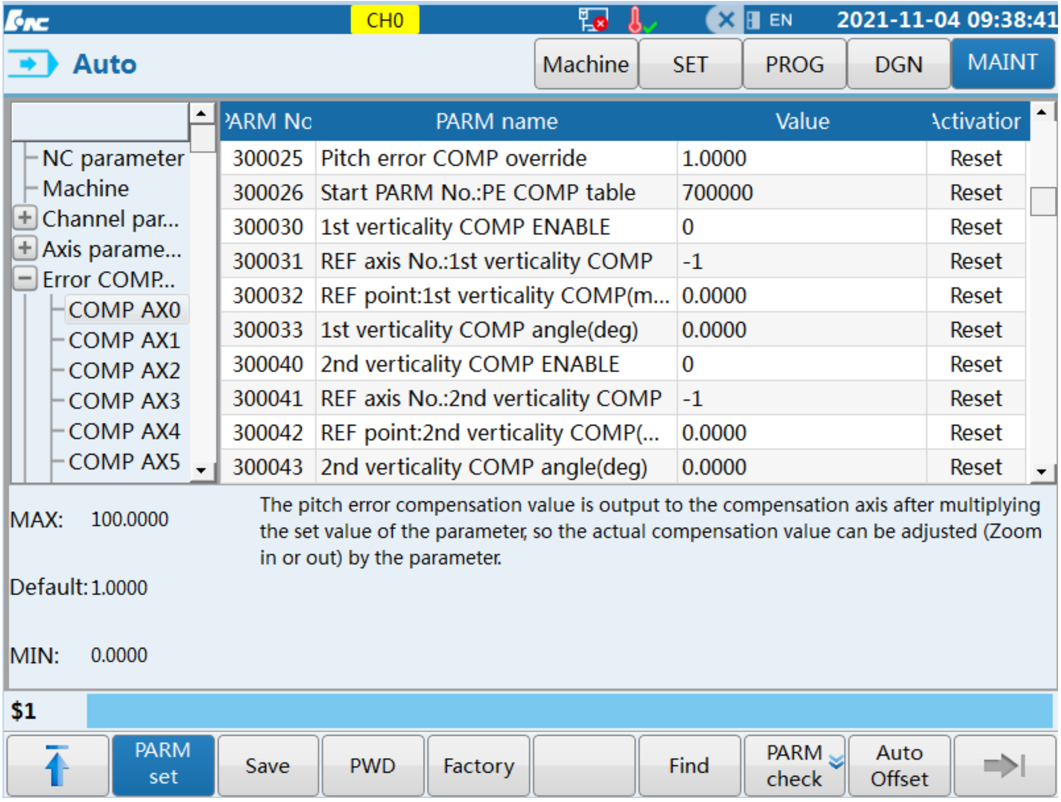



Figure 1-12 21 spatial errors--pitch error compensation function interface

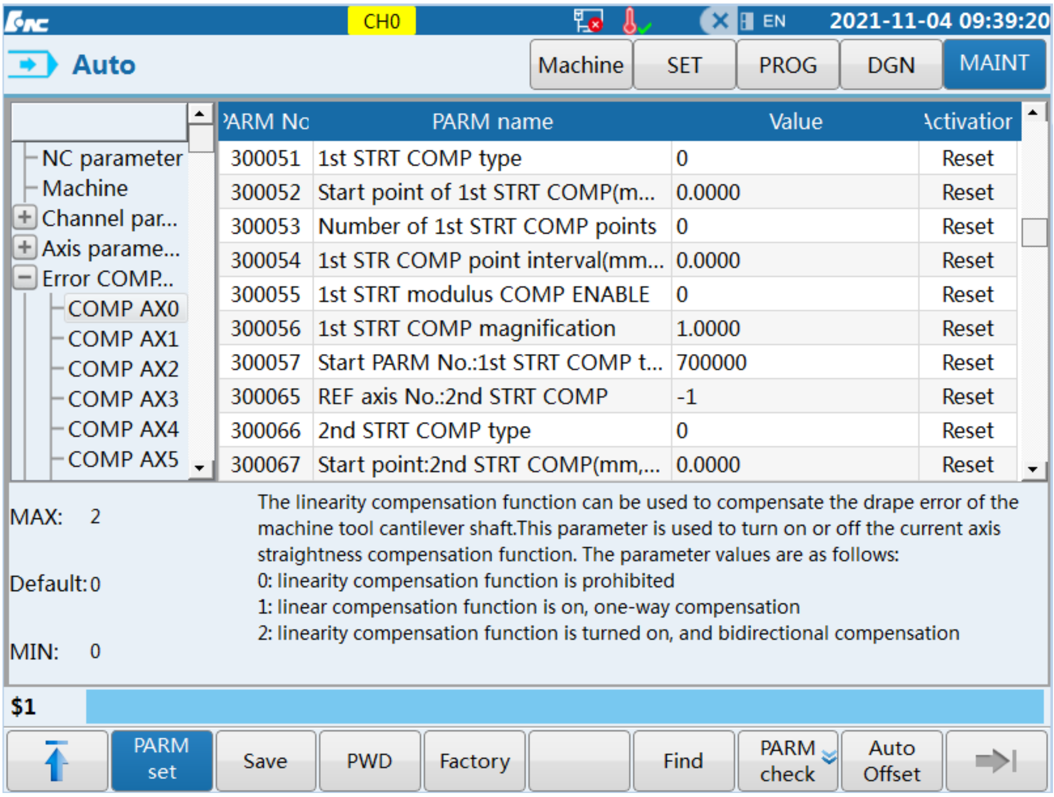



Figure 1-13 21 spatial errors--straightness error compensation function interface

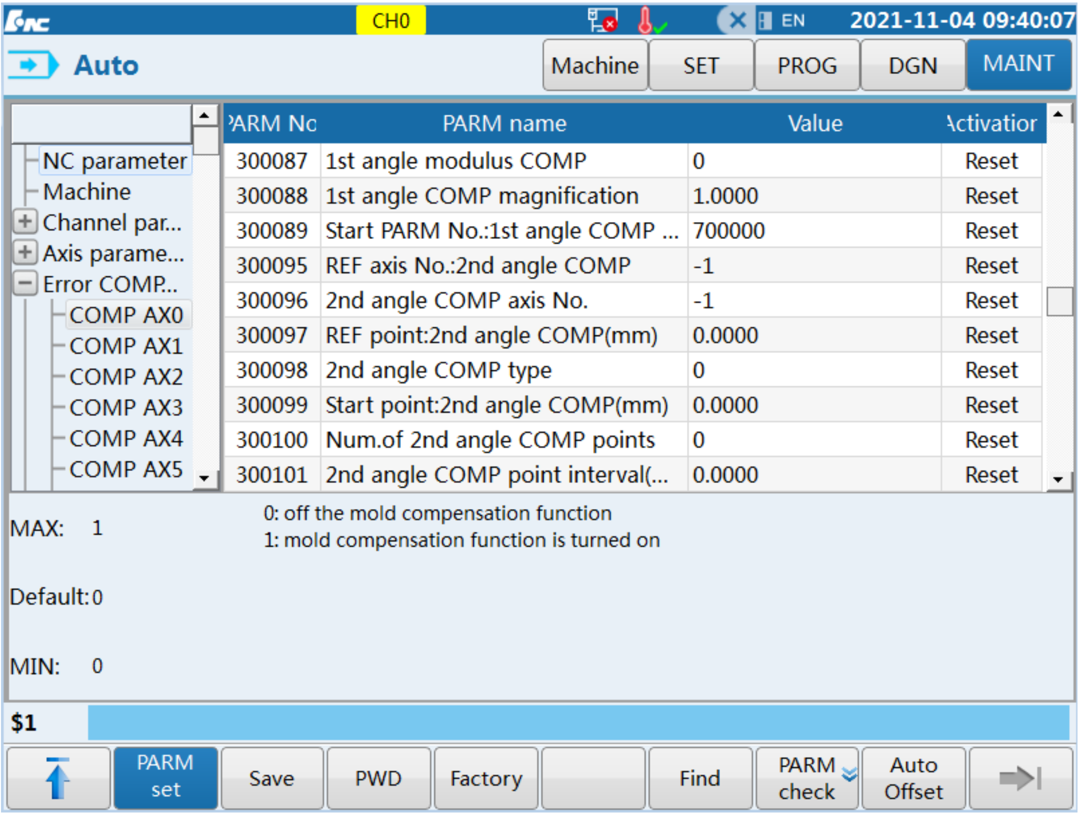



Figure 1-14 21 spatial errors--angle error compensation function interface

## 1.3. Full Closed-loop Optimization

### 1.3.1. Introduction to Full Closed-loop

Linear axis or rotary axis of machine tool often contains multi-stage mechanical transmission; while assembly quality of mechanical transmission part, wear and thermal deformation caused by the use directly affect workpiece machining quality and accuracy. Thus, for some machine tools, grating ruler or circular grating will be installed on the final load side for full-closed loop control in order to ensure machining accuracy.

Fig. 1-15 and Fig. 1-16 are the control block diagram of semi-closed loop and full-closed loop of drive. Under semi-closed loop control, feedback of motor position and speed is from the motor encoder;

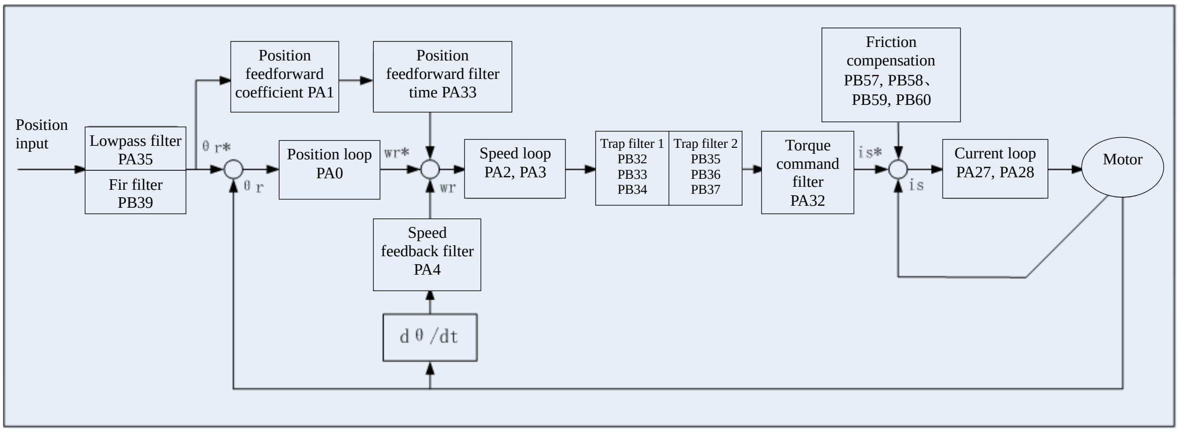



Figure 1-15 Semi-closed loop control block diagram

Under full-closed loop control, speed feedback is from the motor encoder and position feedback is given by feedback part on the load side.

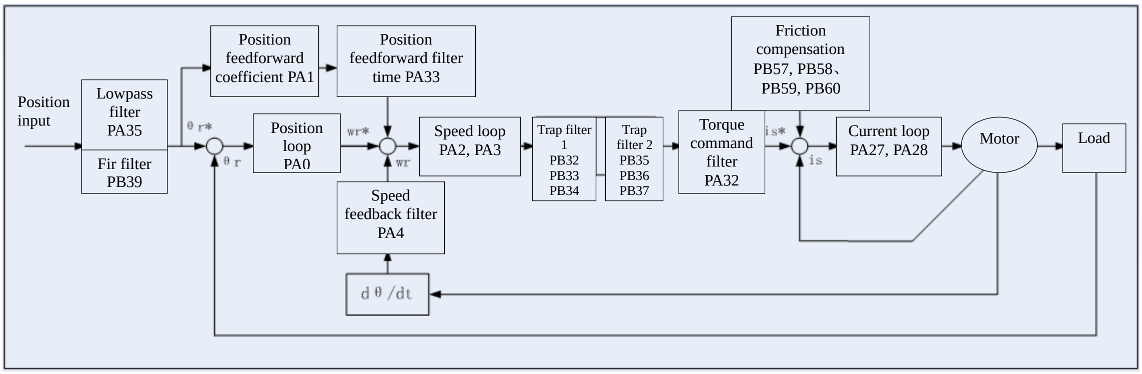



Figure 1-16 Full-closed loop control block diagram

### 1.3.2. Related Parameters

### A. Full-closed loop parameters

Full-closed loop feedback of drive supports several signal types, which are set through control parameters STB13, STB12, and STB11, as shown in Table 1-4:

Table 1-4 Full-closed loop feedback type

| STB13 | STB12 | STB11 | Type | Remark |
|---|---|---|---|---|
| 0 | 0 | 0 | TTL squre wave | |
| 1 | 1 | 0 | 1Vpp sine cosine | e.g.: ERA4000 series, etc. |
| 0 | 1 | 0 | Yuheng absolute linear grating ruler | JC09/JC12 absolute grating ruler |
| 1 | 0 | 1 | Endat2.2 absolute circular grating | RCN1280/5380/8380 |
| 1 | 0 | 0 | Endat2.2 absolute linear grating | LC183/483/483/485 |
| 0 | 0 | 1 | Fagor SSI angle encoder 27bit | HA/DA 27bit/32768 |
| 0 | 1 | 1 | Fagor SSI angle encoder 23bit | HA/DA 23bit/32768 |
| 1 | 1 | 1 | Fagor SSI linear grating ruler | LA /SA/GA/L2A/S2A/G2A |

Main control parameters of full-closed loop are shown in Table 1-5

Table 1-5 Full-closed loop control parameters

| Parameter | Meaning |
|---|---|
| STB13, STB12, STB11 | Full-closed loop type setting |
| STB14 | Full-closed loop enable |
| PA10 | Inverted full-closed loop feedback setting (PA10 is set as 512 to invert full-closed loop feedback |
| PB46 (>myriabit) | Full-closed loop feedback pulse count corresponding to a motor revolution=PB46*10000+PB47 |
| PB47 (>myriabit) | |
| PB54 | Synchronous error detection range |
| PB55 | Number of bits shifted to the right of encoder2 |
| PA30 | Dual loop feedback compensation coefficient |
| PA39 | Full-closed loop damping control gain |

## B. Commissioning Steps

(1) First, correctly set full-closed loop feedback types STB11-STB13, and set PB46-PB47 (full-closed loop feedback pulse count corresponding to a motor revolution) based on number of encoder pulses, full-closed loop feedback resolution, and mechanical transmission ratio. Set STB14 as 0 to run the motor in semi-closed loop. Cut off the power after saving.

(2) Power on again, control the motor to run incrementally for one revolution, and observe related displayed items on the drive side:

- Motor encoder feedback pulse count (total pulse count=DP-PFH*10000+DP-PFL):
  DP-PFL (motor encoder feedback pulse count, below myriabit)
  DP-PFH (motor encoder feedback pulse count, above myriabit)
- Full-closed loop feedback pulse count (total pulse count=DP-FPH*10000+DP-FPL):
  DP-FPL (full-closed loop feedback pulse count, below myriabit)
  DP-FPH (full-closed loop feedback pulse count, above myriabit)

If motor encoder feedback pulse count changes in the opposite direction to full-closed loop feedback pulse count, set PA10 as 512 to invert the encoder feedback. Then, set STB14 as 1 to enable full-closed loop. According to the full-closed loop feedback pulse count corresponding to a thread pitch, reset <*Numerator of electronic gear ratio*> and <*Denominator of electronic gear ratio*> in coordinate axis parameters of CNC system. If the drive PA10 is set as 512, set <*Inverted encoder feedback sign*> in device interface parameters, then cut off the power after save.

(3) Power on again and run the motor in full-closed loop. Gradually adjust speed and position gain parameters in the following ways.

Set both position loop proportional gain (parameter PA0) and speed loop proportional gain (parameter PA2) as 200. Control the worktable to run back and forth at the speed of F1000mm/min, and gradually increase the value of speed loop proportional gain during operation until vibration occurs. At that time, the current value is the critical speed loop gain. Then, set the value of speed loop gain as 80% of critical speed loop gain. and gradually increase the value of position loop gain until vibration occurs. At that point, the current value is the critical speed loop gain. Set the value of speed loop gain as 80% of critical speed loop gain as well.

## C. Full Closed Loop Vibration Suppression Measures

Due to the elasticity of the transmission shaft and the influence of non-linear factors such as clearance and friction, the mechanical vibration may easily occur when full closed loop control is used, which can be eliminated using the following methods.

### Adjust gain

The most direct way is to reduce position loop proportional gain (parameter PA0) and properly increase acceleration/deceleration time constant in system axis parameter. However, if position loop gain is reduced or acceleration/deceleration time constant is increased excessively, dynamic response performance of the system will decline, a big position tracking error will be caused, and dynamic machining requirements can hardly be met. The following measures can be taken for vibration suppression when gain adjustment cannot meet the requirements.

### Use notch filter

Notch filter can be used to restrict resonance of mechanical part. If mechanical resonance tested by SSTT or diagnostic tools of the system is constant vibration of a constant frequency (vibration frequency does not change with speed) and vibration frequency is greater than 400Hz, notch filter can be used to restrict mechanical resonance. If vibration frequency is less than 400Hz or intermittent vibration occurs, notch filter should not be used. For commissioning of notch filter and test for vibration frequency, refer to instructions on commissioning of SSTT notch filter.

### Dual-loop feedback control function

The principle of dual-loop feedback function is that positional deviation of axis is under semi-closed loop control during operation, and full-closed loop control during shutdown. It is used to improve dynamic performance of full-closed loop operation and avoid vibration. This function is suitable for the situation where the mechanical transmission clearance is large, the semi-closed loop can run stably, but the full closed loop has poor characteristics, and vibration is prone to occur after the position and speed gains are increased.

Parameter PA30 Dual-loop feedback compensation, unit: 0.1ms

PA30 is 0 by default. During commissioning, increase PA30 by 30 each time and increase PA2 speed proportional gain and PA0 position proportional gain in steps of 50. Repeat the above steps and limit the maximum set value of PA30 to 300.

### Damping control function

The damping control function is a function that reduces the vibration on the load side by feeding back the speed difference between the motor end and the load end of the machine tool to the torque command. This function is equivalent to increasing the damping coefficient of the system. If the part between the motor and mechanical load has poor rigidity or elastic deformation, a big speed difference may easily occur between the motor end and the load end and consequently it is hard to control speed on the load end steadily. In this case, damping control can be adopted.

Set PB72 as 1 to enable damping control function. PA39 refers to damping control gain and the value is 0 by default. During commissioning, gradually increase the value of PA39 by steps of 100 and observe whether vibration on the load end can be restricted during operation.

## 1.4. Synchronization Control Matching Optimization

### 1.4.1. Initial Threshold Setting

Basic settings of synchronous axis include: System parameter, PLC setup, etc., which are described in detail in the later chapter Function of Gantry Synchronous Axis. Here only threshold settings of synchronous axis are stressed.

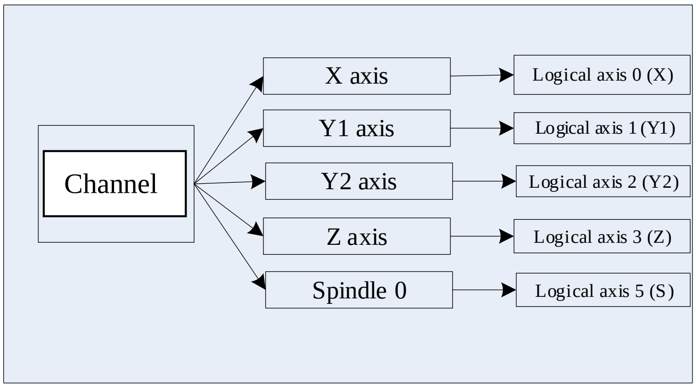



Figure 1-17 Axis configuration

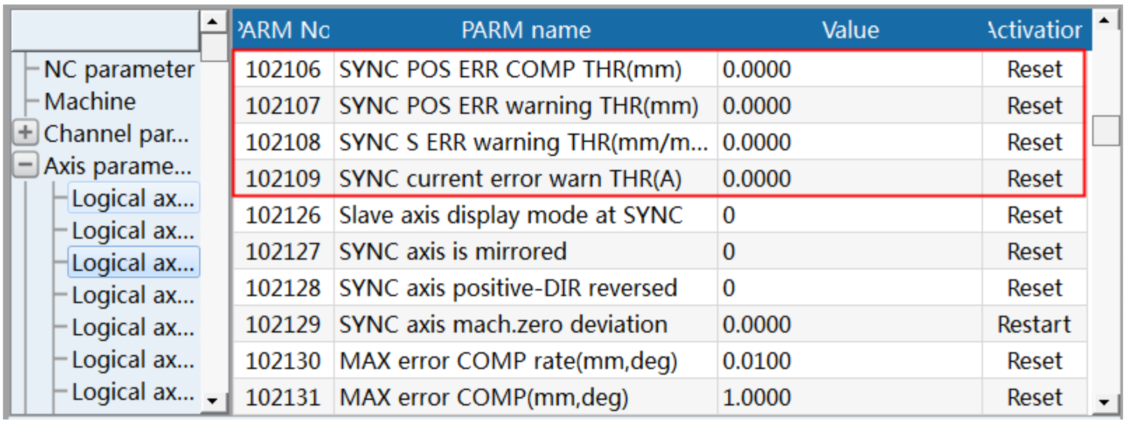
Y1 is a master axis and Y2 is a slave axis. Threshold parameter of slave axis is shown below

Table 1-6 Synchronous axis threshold parameter list

| Parameter | | Notes to parameters | Recommended value |
|---|---|---|---|
| 102106 | Synchronization position error compensation threshold | Maximum allowable synchronization error compensation value | Alternate parameter, no need to be set |
| 102107 | Synchronization position error alarm threshold | An alarm is given when synchronization position error exceeds the value | |
| 102108 | Synchronization speed error alarm threshold | An alarm is given when synchronization speed error exceeds the value | Alternate parameter, no need to be set |
| 102109 | Synchronization current error alarm threshold | An alarm is given when synchronization current error exceeds the value | |

As shown above, logical axis 2 is a slave axis. Relevant thresholds are set as follows

Upon initialization, the following synchronization thresholds are set as 0 and detection is not enabled.



| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |
| 102126 | Slave axis display mode at SYNC | 0 | Reset |
| 102127 | SYNC axis is mirrored | 0 | Reset |
| 102128 | SYNC axis positive-DIR reversed | 0 | Reset |
| 102129 | SYNC axis mach.zero deviation | 0.0000 | Restart |
| 102130 | MAX error COMP rate(mm,deg) | 0.0100 | Reset |
| 102131 | MAX error COMP(mm,deg) | 1.0000 | Reset |



Figure 1-18 System threshold setups

Save parameters, power off and restart the system.

### 1.4.2. Position Calibration of Synchronous Axis

In some case, the position of synchronous axis should be calibrated. Handwheel can be used for adjustment of synchronous axis after synchronization is released.

The system releases emergency stop and switches to handwheel mode. Meanwhile, release synchronization function of synchronous axis in PLC. Adjust the position of synchronous axis and then set the zero point of coordinates.

**PLC setups are as follows**

(1) Add slave axis enable signal

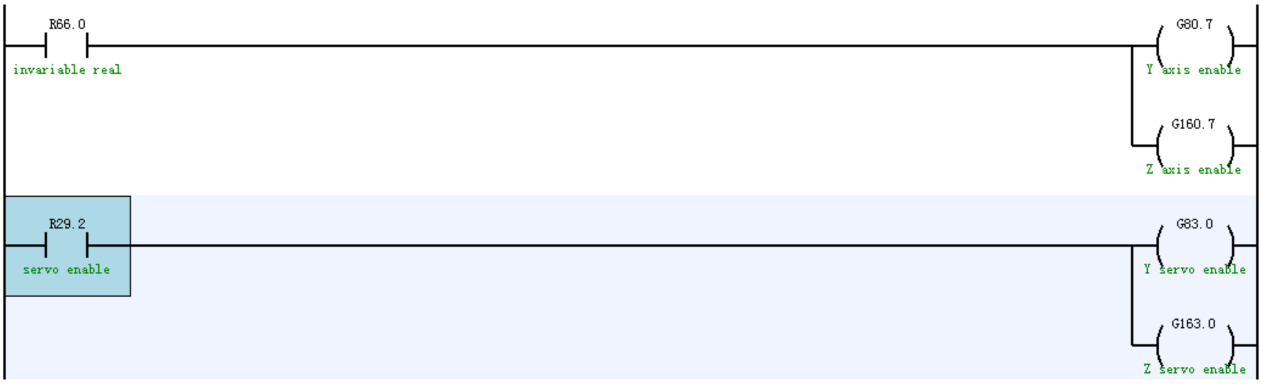



Figure 1-19 Axis enable PLC setups

(2) Add external reset flag G2960.3 upon reset.

**Note:** When external reset flag G2960.3 is not added and the system interface appears an alarm *Slave Axis Tracking Error Too Large*, users cannot clear the message by reset but by adding this flag into PLC.

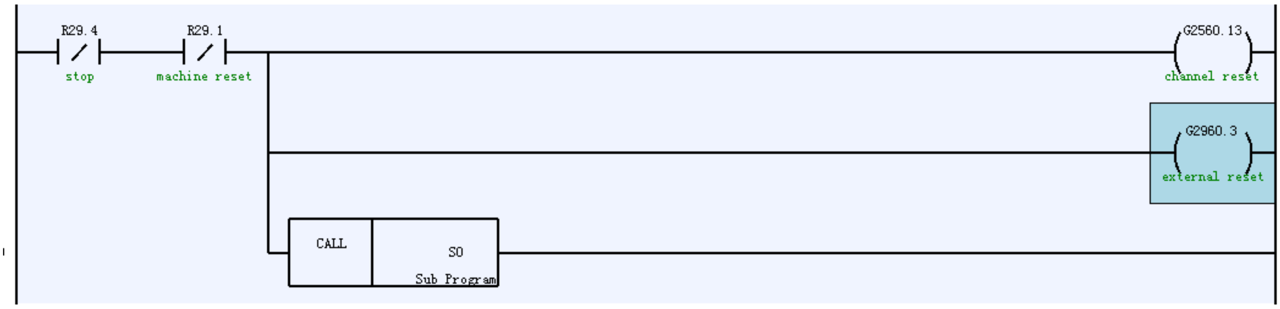



Figure 1-20 External reset signal

(3) Add synchronization release of slave axis to PLC.

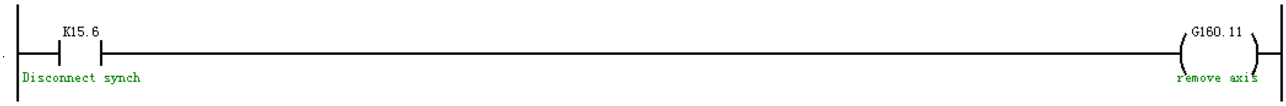



Figure 1-21 Slave axis is desynchronized

### 1.4.3. Synchronous Axis Threshold Setups

(1) Enable automatic adjustment function of synchronous axis

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 0 | Reset |

(2) Set compensation threshold and alarm threshold of synchronous axis to complete configuration of synchronous axis.

| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

**Pay attention to the following:**

①. Presently, threshold of synchronization position error and threshold of synchronous speed error alarm are alternate parameters and need not be set temporarily.

②. threshold of synchronization position error and threshold of synchronization current error alarm should be set according to actual situation.

③. SSTT can be used to sample the position and current of Y and Y1

Calculate maximum value of position and error current of Y and Y1 through error synchronization function of SSTT, and take 1.2-1.5 times of the maximum value and fill it in the system

| | | |
|---|---|---|
| synchronization error | 2 | After tract the two channels(output = ch1 - ch2) |

**Note:**

①. After enable the automatic adjustment of flexible synchronization, if synchronization position error value is less than synchronization position error threshold, the position of slave axis motor will be adjusted automatically when emergency stop is released, in order that machine actual coordinates of master axis and slave axis are consistent. If synchronization position error value is greater than synchronization position error threshold, the system will give an alarm *Synchronization out of Tolerance*, and the position of slave axis motor will not be adjusted automatically either.

②. When automatic adjustment of flexible synchronization is disabled, synchronization position error occurs, and emergency stop is released, the position of synchronous axis will not be adjusted automatically.

## 1.5. Five-axis RTCP Calibration

### 1.5.1. 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.

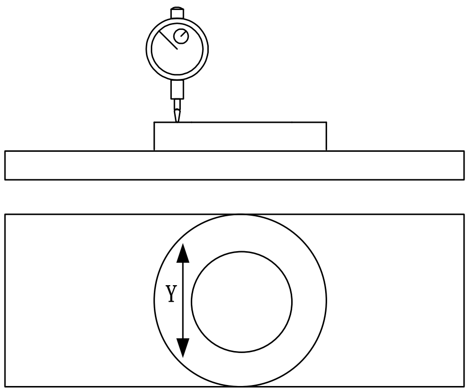



Figure 1-22 A axis accuracy test

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.

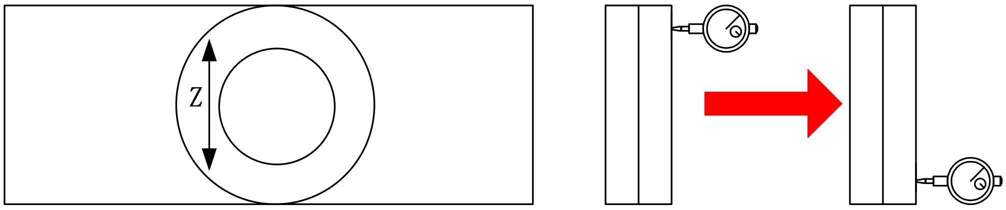



Figure 1-23 A axis accuracy test

- **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.

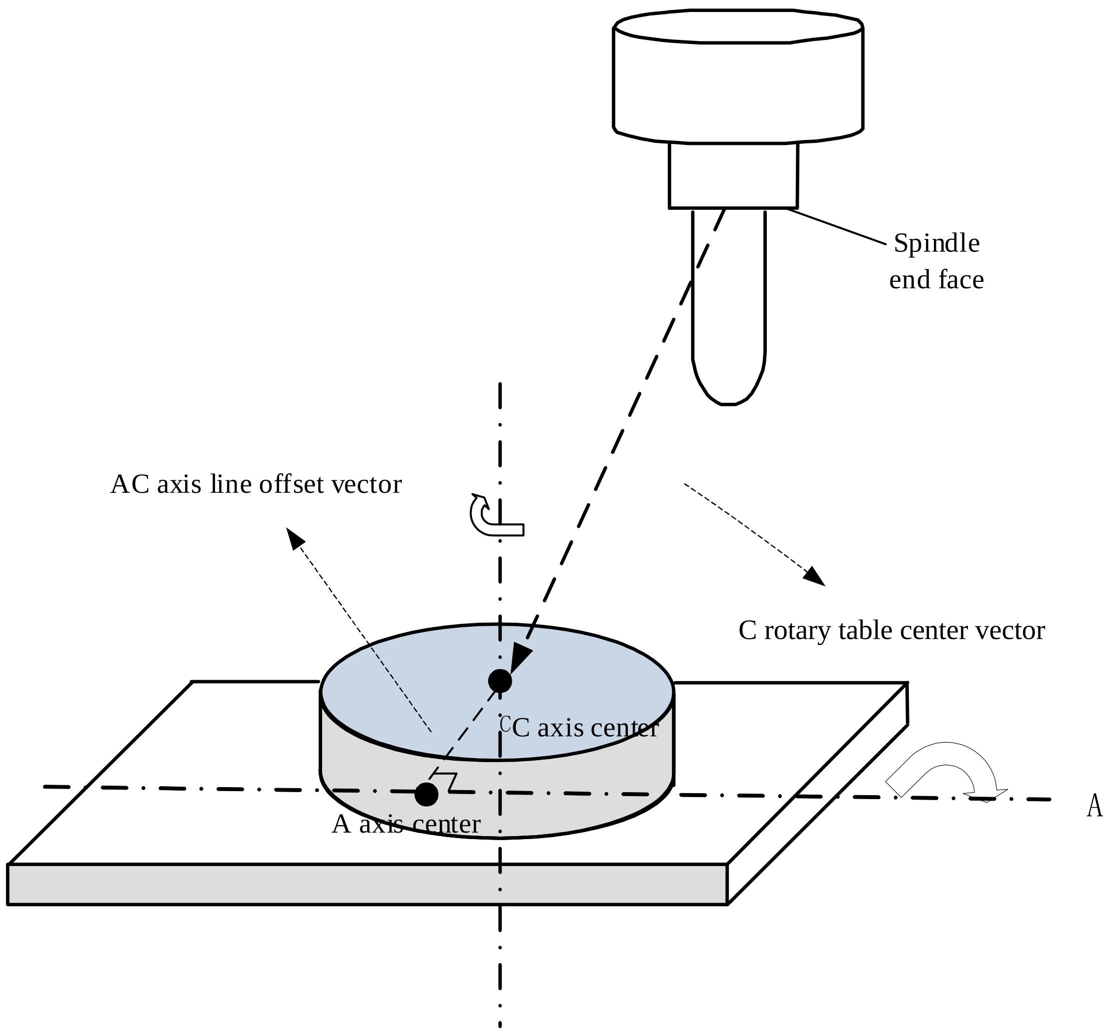



Figure 1-24 RTCP parameter diagram

**(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.

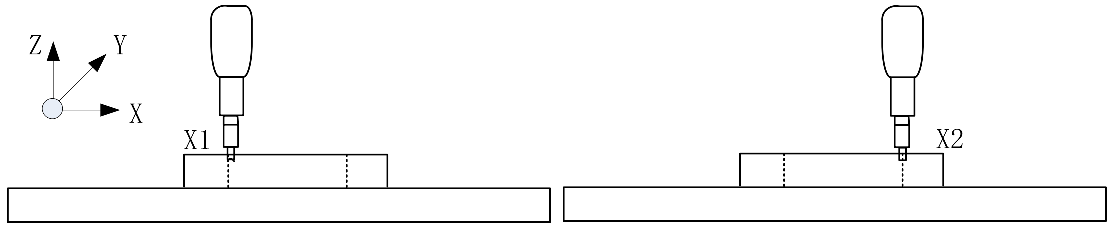



Figure 1-25 Rotation center in X direction of C axis

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.

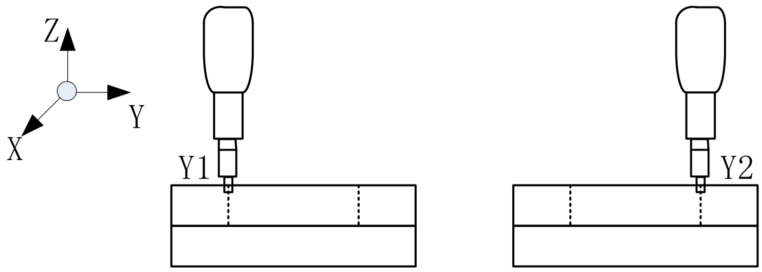



Figure 1-26 Rotation center in Y direction of C axis

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.

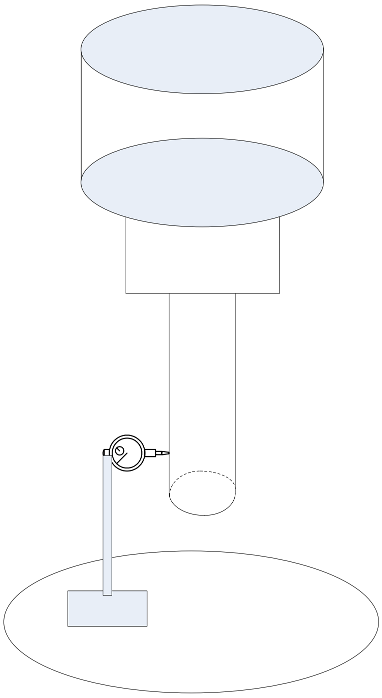


Figure 1-27 Test of rotation center of C axis

**(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.

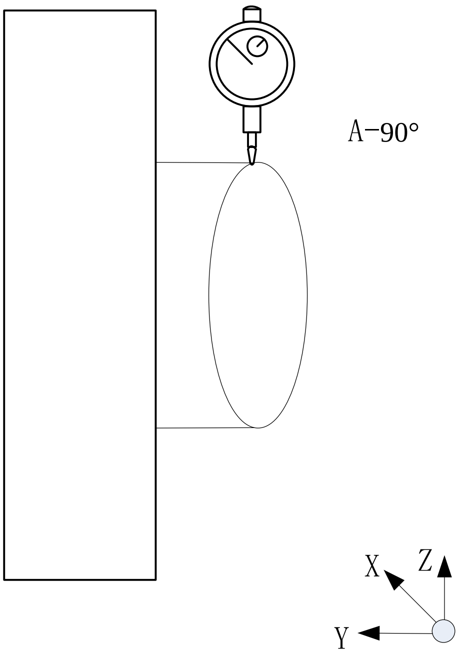



Figure 1-28 Offset vector of AC axis when A axis can rotate for ±90°

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.

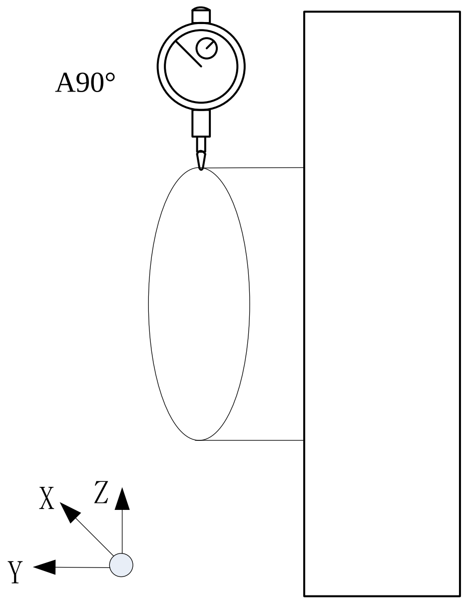



Figure 1-29 Offset vector of AC axis when A axis can rotate ±90°

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.

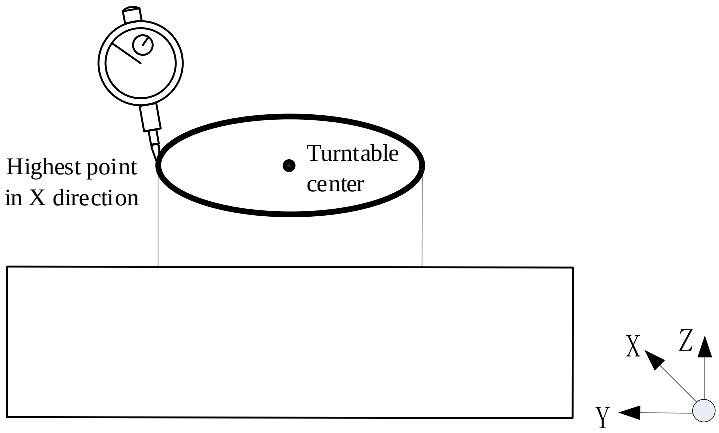



Figure 1-30 Offset vector of AC axis when A axis cannot rotate ±90°

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.

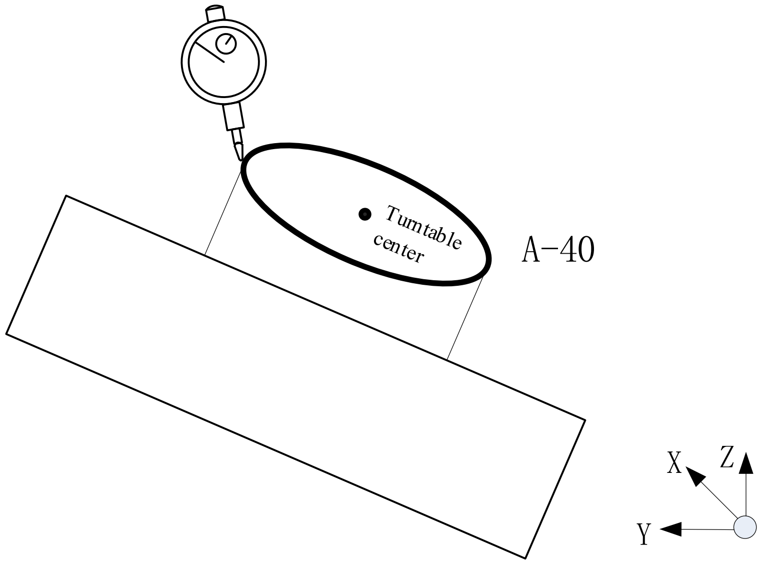



Figure 1-31 Offset vector of AC axis when A axis cannot rotate ±90°

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.

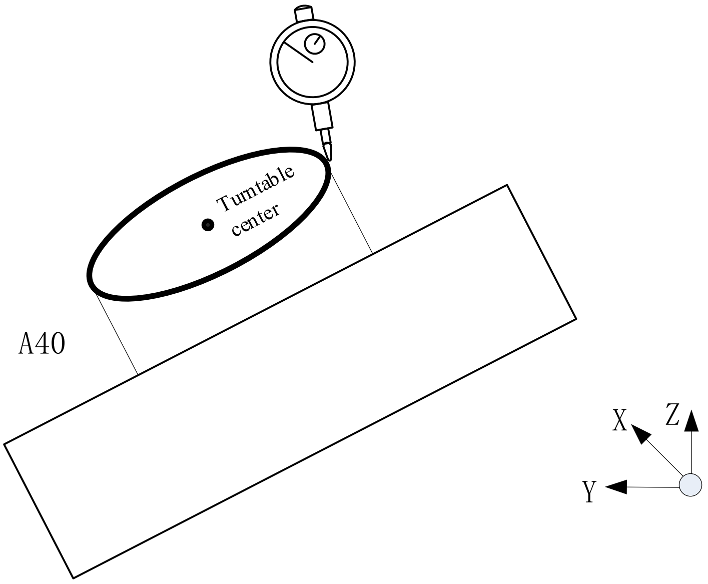



Figure 1-32 Offset vector of AC axis when A axis cannot rotate for ±90°

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.

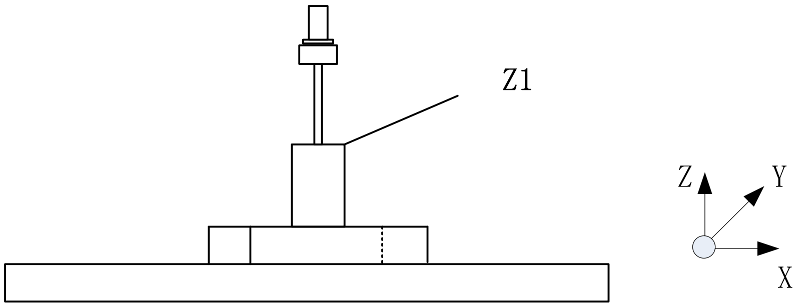



Figure 1-33 Offset vector in Z direction of C axis center

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.

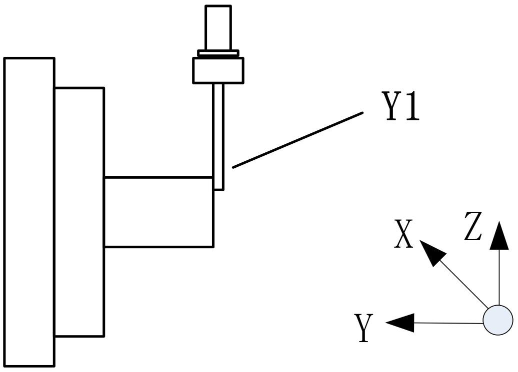



Figure 1-34 Offset vector in Z direction of C axis center

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.

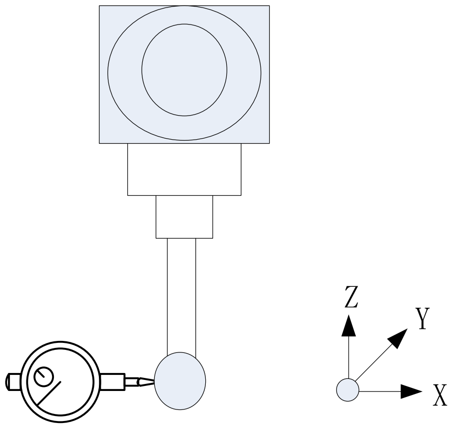



Figure 1-35 RTCP accuracy test of C axis

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.

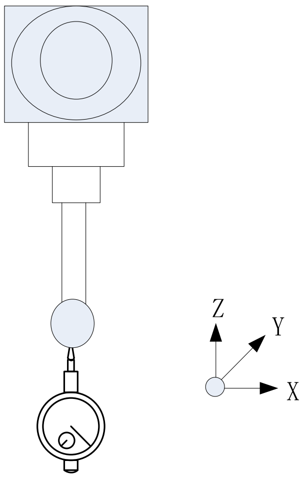



Figure 1-36 RTCP accuracy test of A axis

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

### 1.5.2. 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.

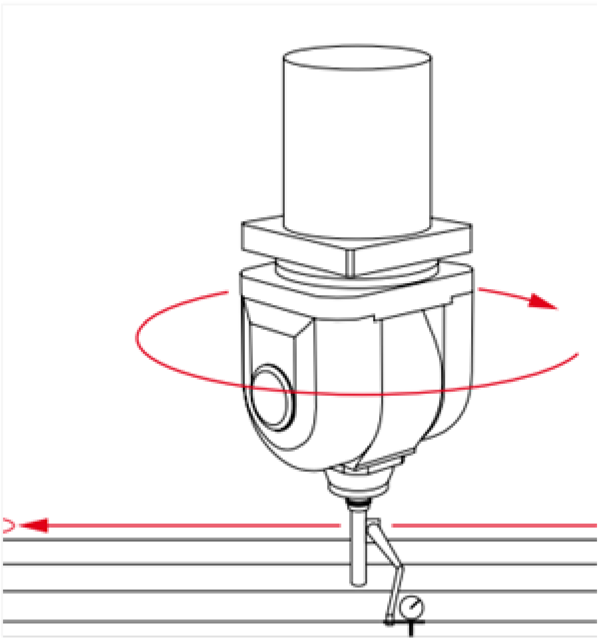


Figure 1-37 Accuracy test of C axis

◆ **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.

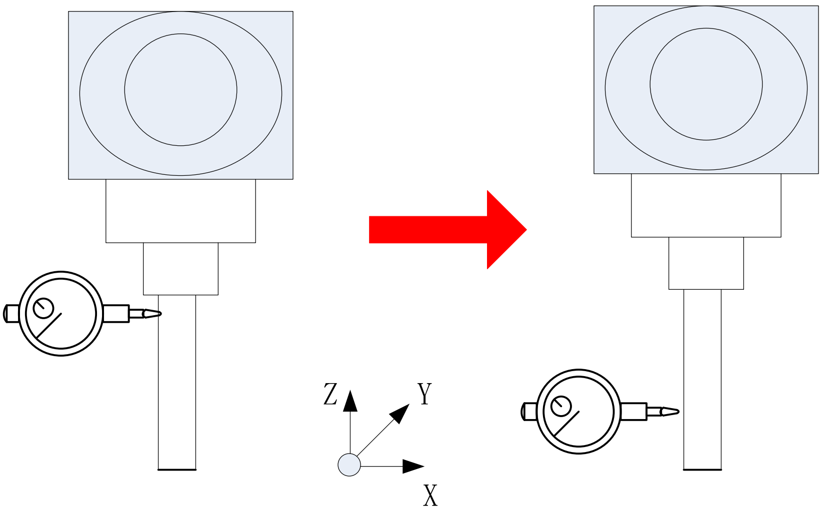



Figure 1-38 Zero position test of B axis

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.

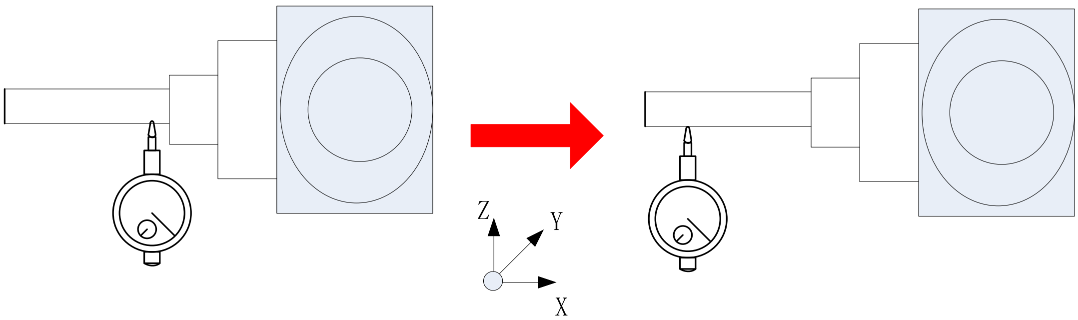



Figure 1-39 90° positioning accuracy test of B axis

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.

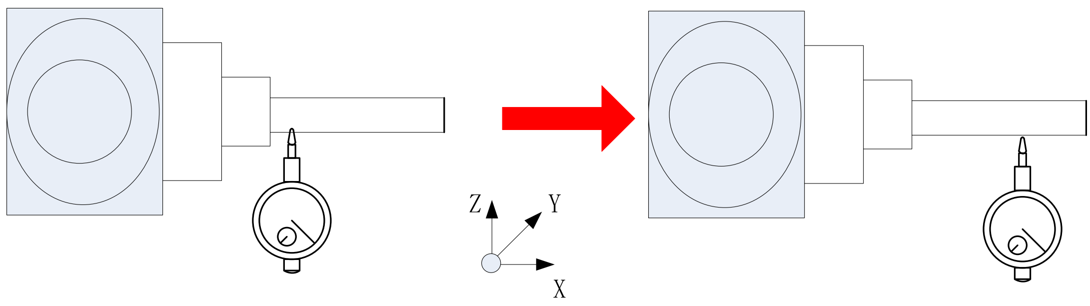



Figure 1-40 90° positioning accuracy test of B axis

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.

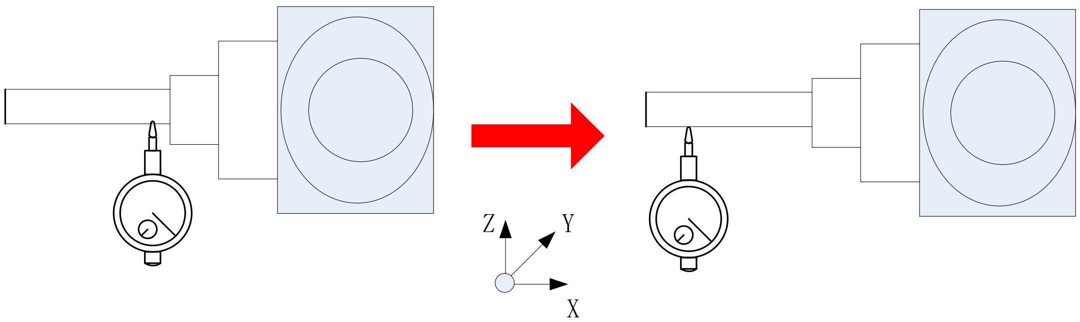



Figure 1-41 Positioning accuracy test of C axis

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.

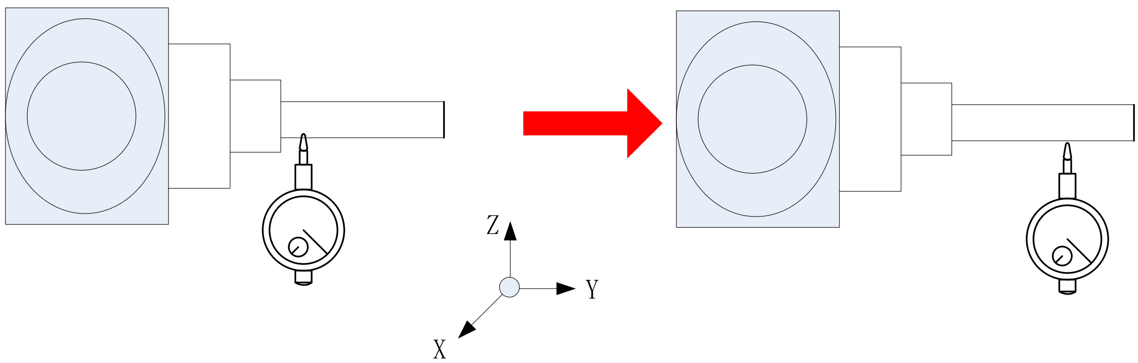



Figure 1-42 Positioning accuracy test of C axis

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.

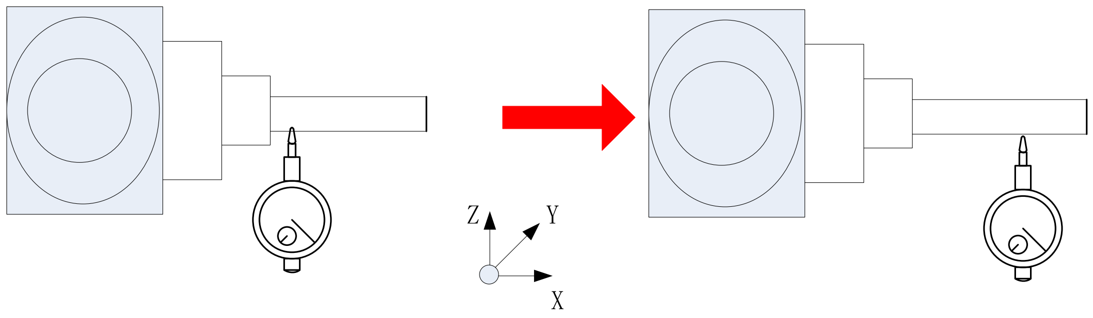



Figure 1-43 Positioning accuracy test of C axis

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.

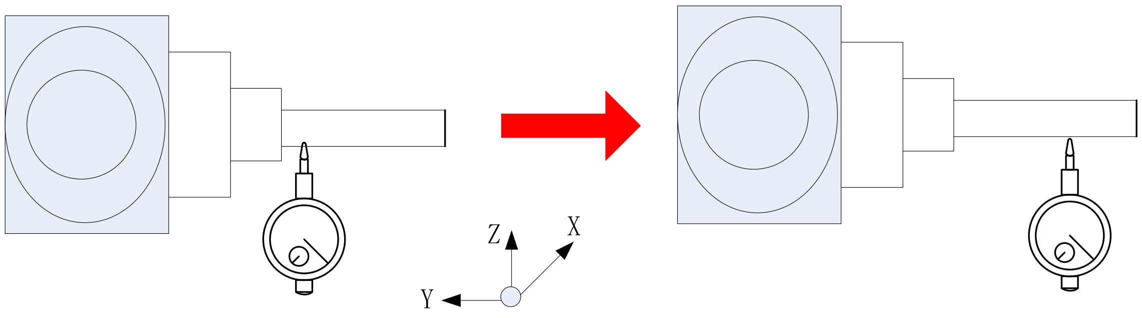



Figure 1-44 Positional accuracy test of C axis

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.

**2. 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.

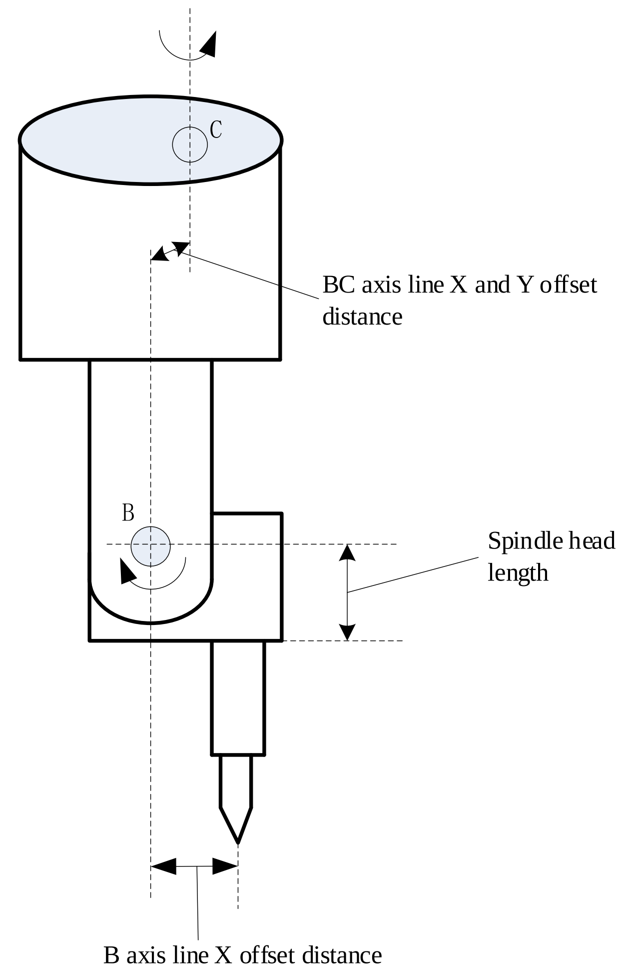


Figure 1-45 RTCP parameter calibration diagram

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).

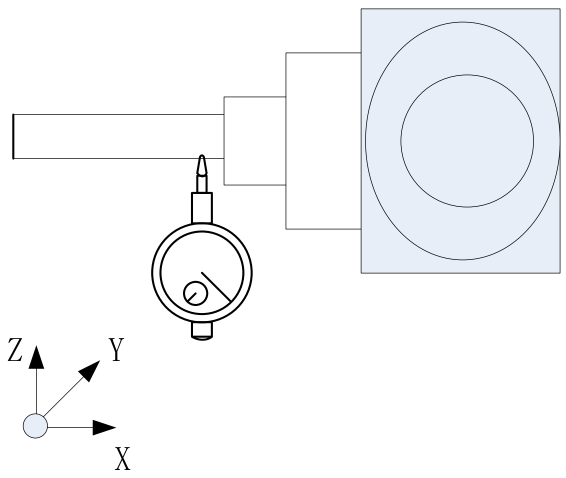


Figure 1-46 Offset distance in X direction of B axis

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.

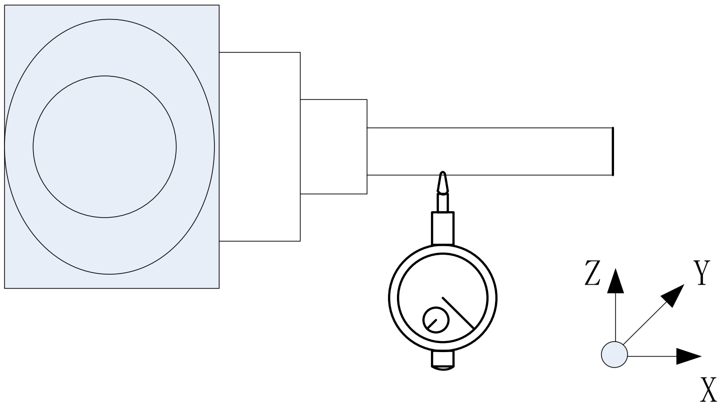



Figure 1-47 Offset distance in X direction of B axis

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)

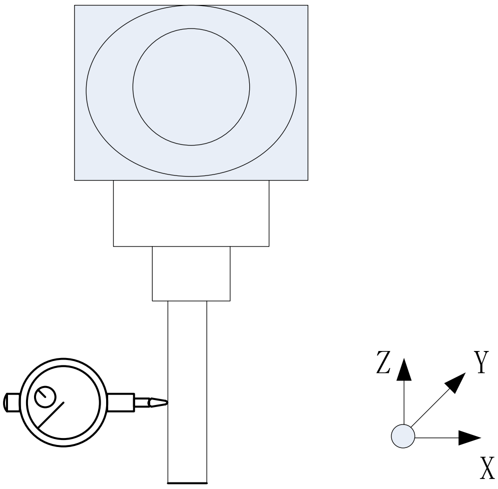



Figure 1-48 Offset distance in X direction of BC axes

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)

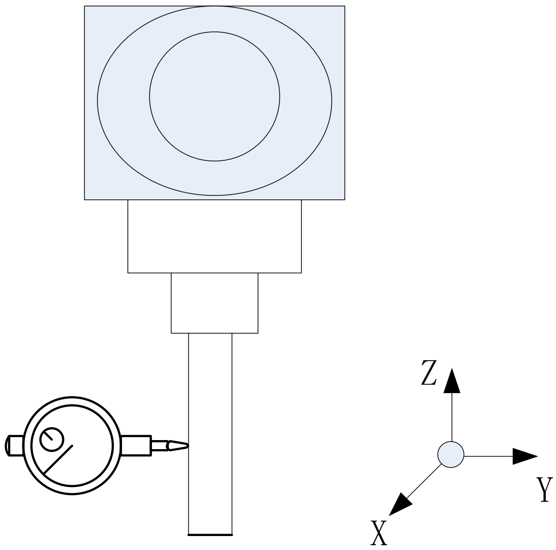



Figure 1-49 Offset distance in Y direction of BC axes

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.

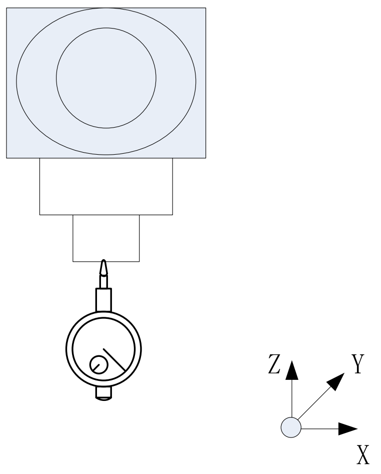



Figure 1-50 Spindle head length measurement

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.

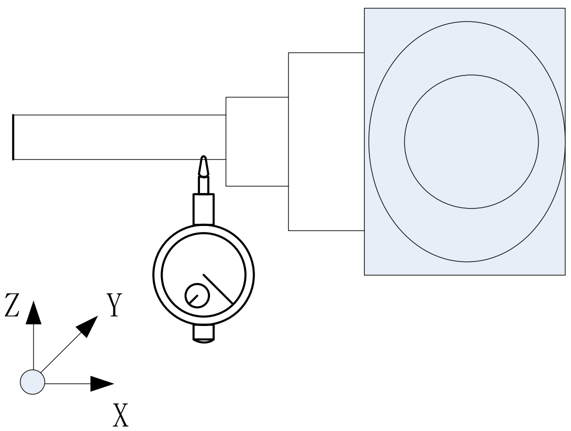



Figure 1-51 Spindle head length test

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.

**3. 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.

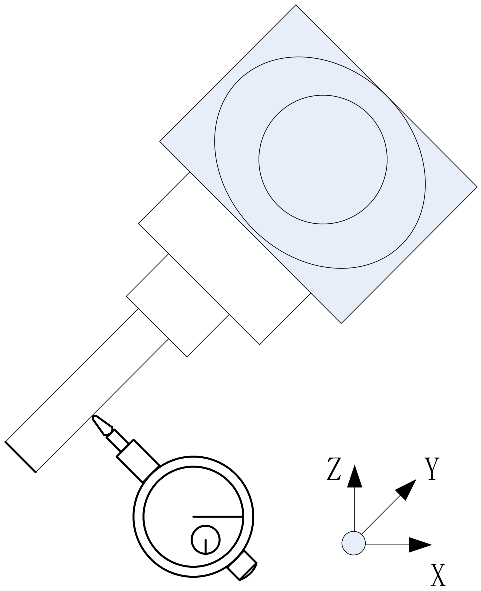



Figure 1-52 Positioning accuracy test of B axis

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.

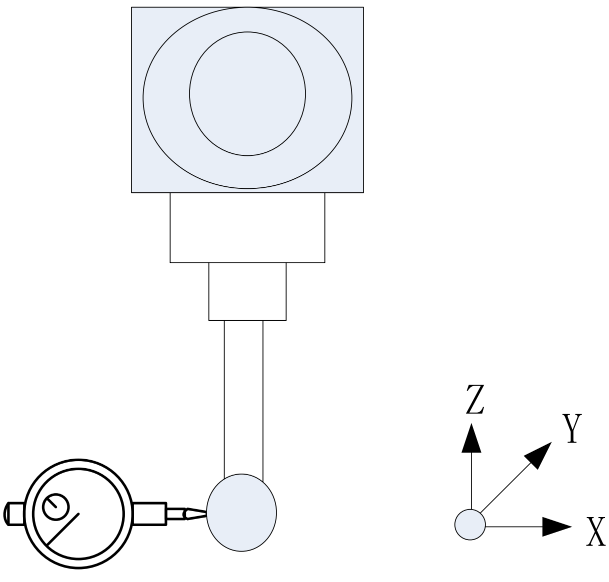



Figure 1-53 RTCP accuracy test of C axis

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

```
G54
F500
```

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

e) Observe 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.

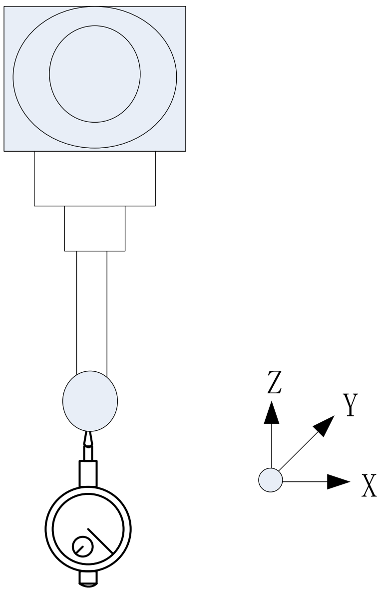



Figure 1-54 RTCP accuracy test of B axis

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

### 1.5.3. Calibration of B Swivel Head & C Rotary Table Mixed Structure

**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

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

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

Move Z axis up and down.

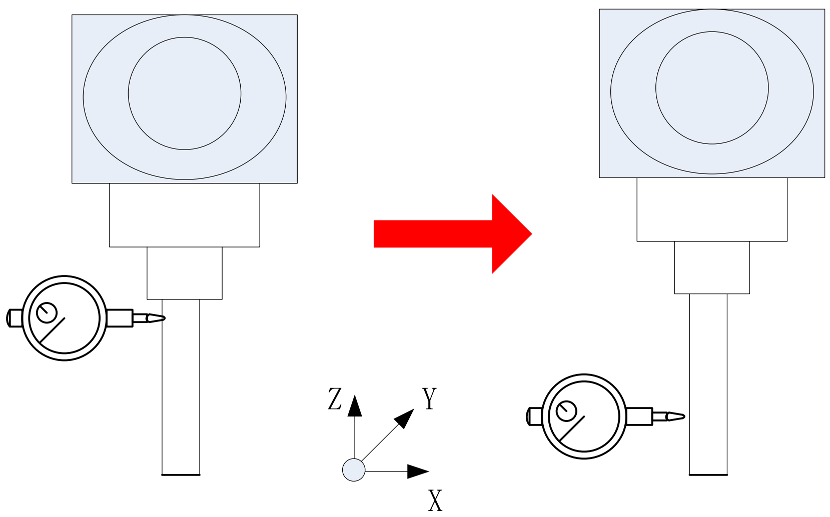



Figure 1-55 Zero test of B axis

c) 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.

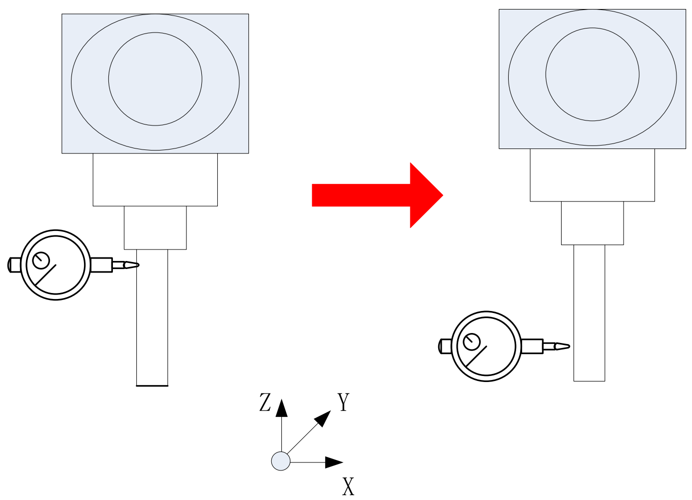



Figure 1-56 Positioning accuracy test of B axis

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.

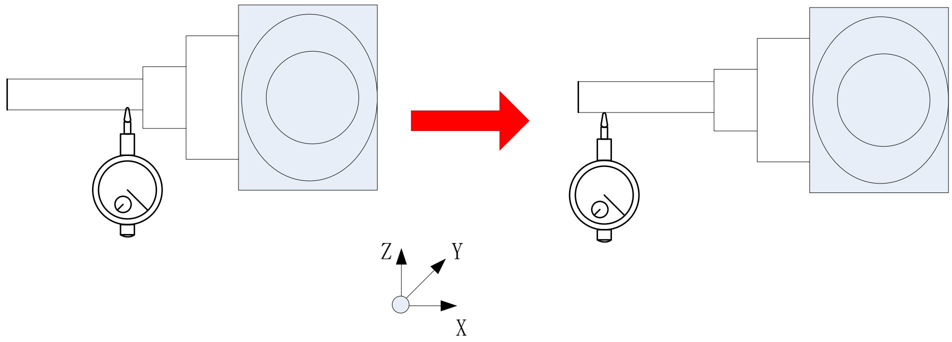



Figure 1-57 Positioning accuracy test of B axis

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.

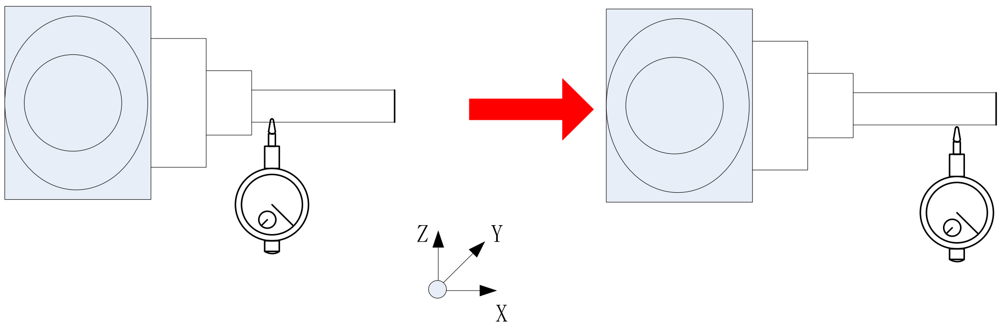



Figure 1-58 Positioning accuracy test of B axis

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.

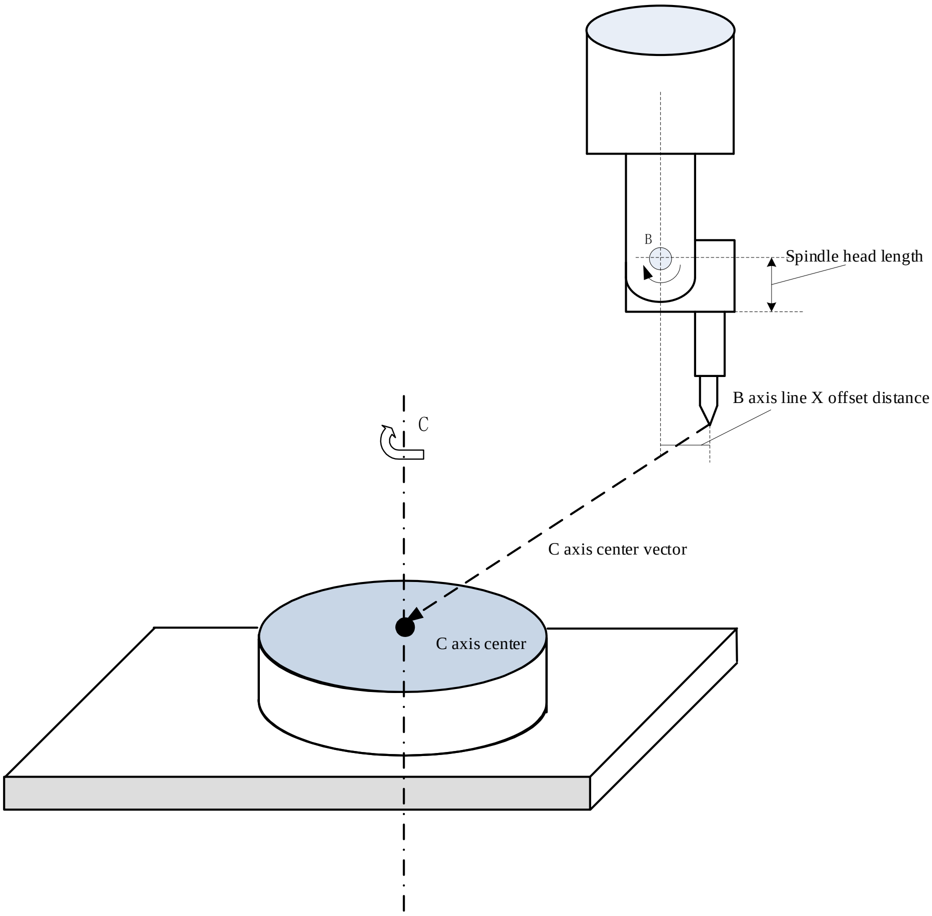



Figure 1-59 RTCP parameter diagram of machine tool

**(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).

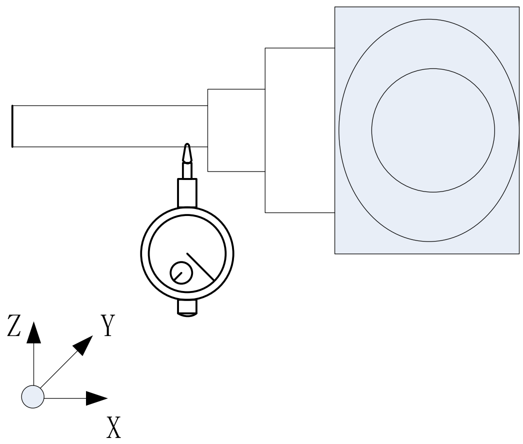



Figure 1-60 Offset distance in X direction of B axis

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.

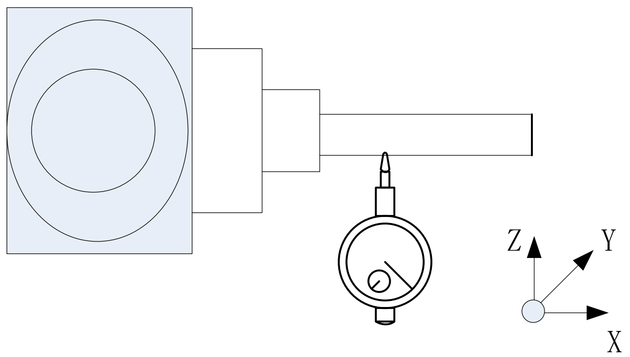



Figure 1-61 Offset distance in X direction of B axis

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.

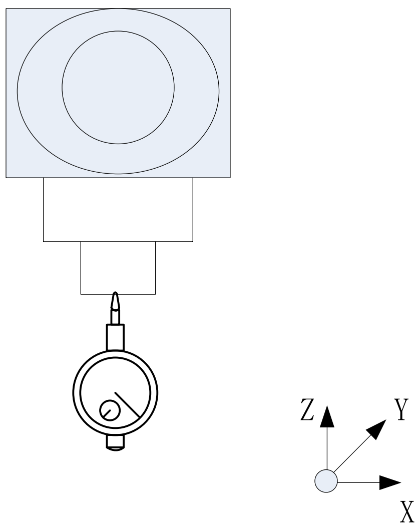



Figure 1-62 Spindle head length measurement

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.

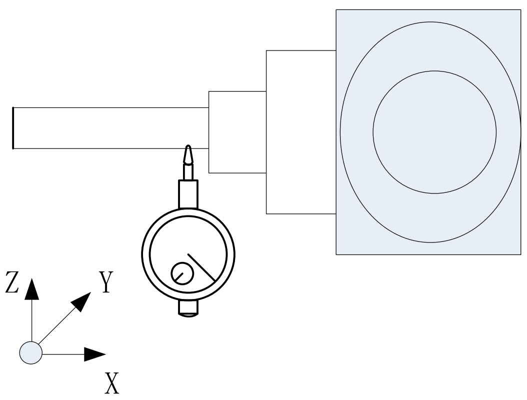



Figure 1-63 Spindle head length measurement

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.

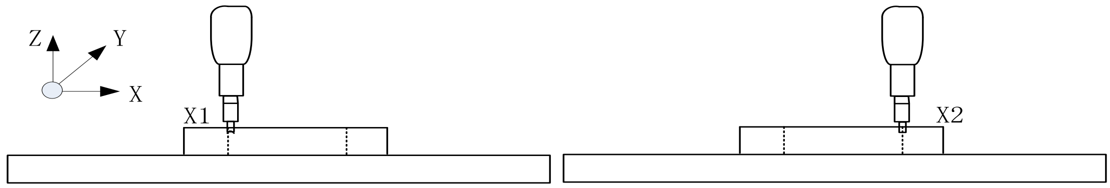



Figure 1-64 Offset distance in X direction of C axis center

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.

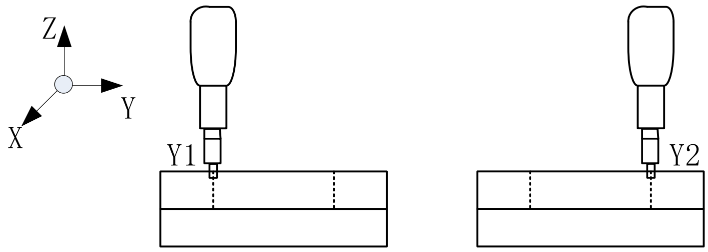



Figure 1-65 Offset distance in Y direction of C axis center

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.

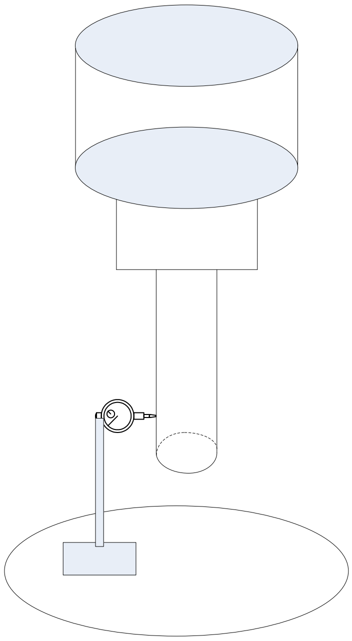


Figure 1-66 C axis center position test

◆ **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.

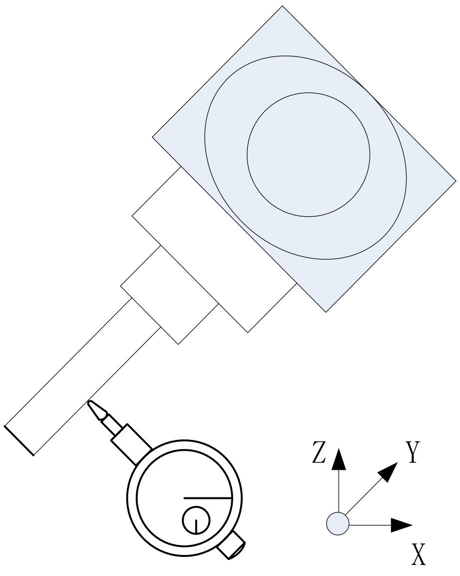



Figure 1-67 Positioning accuracy test of B axis

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.

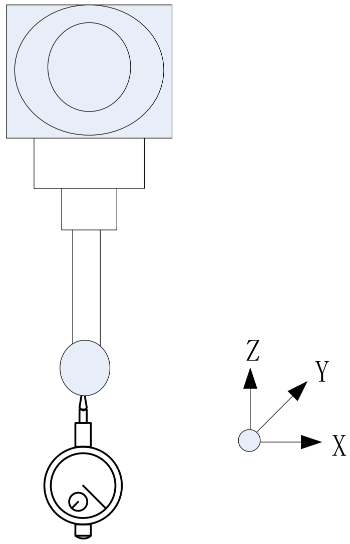



Figure 1-68 RTCP accuracy test of B axis

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.

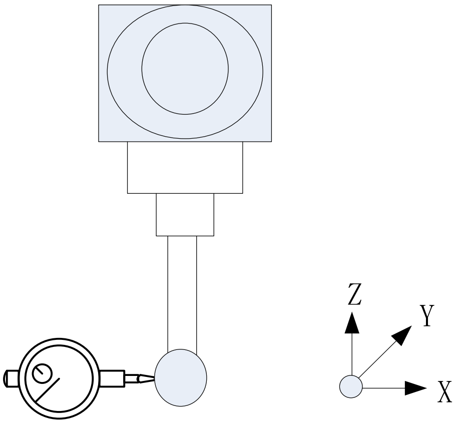



Figure 1-69 RTCP accuracy test of C axis

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

# 2. Function Commission

## 2.1. RTCP Function of Five-axis Machine Tool

For traditional three-axis CNC machine tools, the direction of tools and axes remains unchanged during machining and the interpolation motion of machine tool can be performed only along three linear axes. When the part is processed, shortages of three-axis CNC machine tool are highlighted. Compared with three-axis machine tool, simultaneous-five-axis machine tool has another two degrees of rotation freedom and the tool actions can be flexibly changed, which keeps the best cutting condition of tool and effectively avoids machining interference. Thus, Simultaneous-five-axis CNC machining has significant advantages while machining complex free-form surfaces.

Real five-axis machine tool has the RTCP function. Automatic conversion can be performed according to swing of spindle and machine coordinates of rotary table. During programming, only coordinates of the workpiece need to be considered, not the kinematic chain structure of the five-axis machine tool. Whether it is a real five-axis, it does not depend on whether the motion of five axes can be simultaneous, the main key is that the system has the real RTCP five-axis algorithm. In RTCP mode, the tool path of 5-coordinate programming can be planned directly in workpiece the coordinate system, so programming is simpler and more efficient. CNC system without RTCP has to rely on CAM programming and postprocessing. The path of tool nose is planned in advance; the center positions of the fourth and fifth axes should be inputted in the postprocessing module, the generated CNC program is the machine coordinate point, and the motion path of tool nose in actual machining approaches the programming path. As a result, accuracy cannot be guaranteed. Besides, the program must be reset for the same part if machine tool or tool is replaced, which extremely inconvenient in practical use.

### 2.1.1. Tool Center Point Control (RTCP)

During five-axis machining, the path of the tool center changes due to addition of rotary axis and error of machine tool structure. RTCP mode is enabled through corresponding G code program. The system fixes the tool center point as the control point and ensures the tool center point moves along the specified path through real-time tool length compensation. Users just need to carry out five-axis programming under the workpiece coordinate system with no need to consider structural error of machine tool, which greatly simplifies CAM programming and improves machining accuracy.

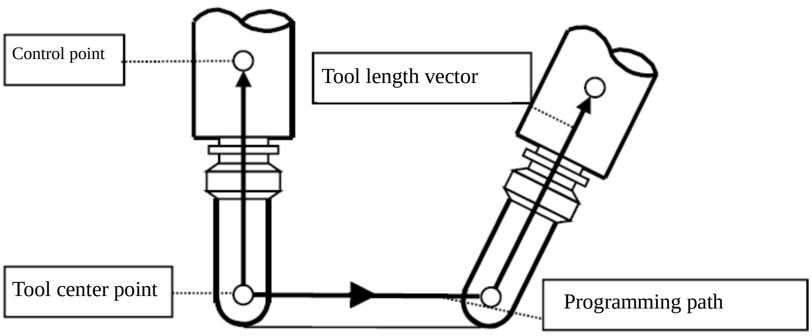
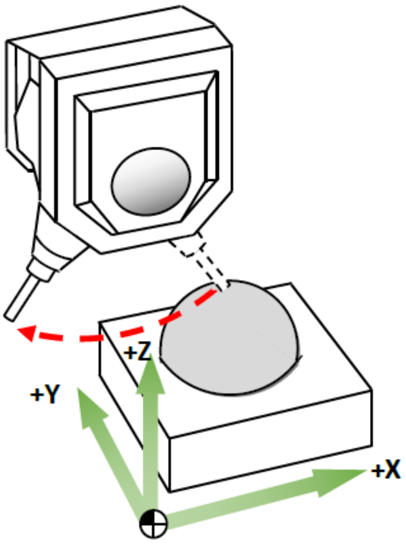
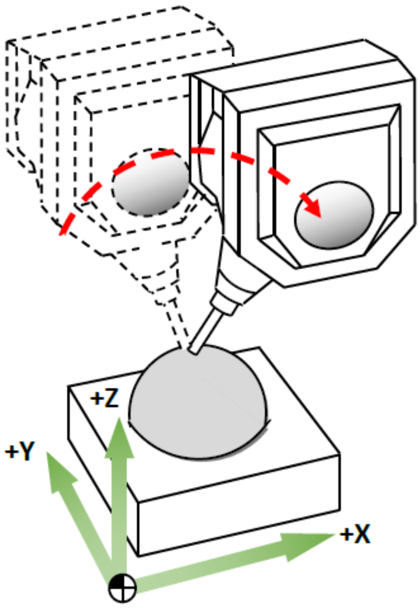



Figure 2-1 Tool center point programming




When RTCP function is not used, the tool rotates around the rotary axis center.




When RTCP function is used, the tool nose will stop at the fixed point; when the rotary axis moves, the system will perform the compensation of linear axis automatically.

Figure 2-2 RTCP diagram

**RTCP Programming Format**

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

G43.4: Rotary axis angle programming

G43.5: Tool vector programming

G49: Cancel RTCP function

Where, G43.4(G43.5) is to enable RTCP function; H specifies tool length compensation number to offset the tool center point toward the control point by one tool length compensation along the tool axis; G49 is to cancel RTCP function.

## 2.2. 3+2 Orientation Machining Function

3+2 orientation machining function is often used for five-axis machining. This function is used to build a trait coordinate system (TCS) on a slope and programming is performed in this coordinate system. Whereas TCS adapts to the slope, programming on a slope is as simple as that on a flat surface. As shown below

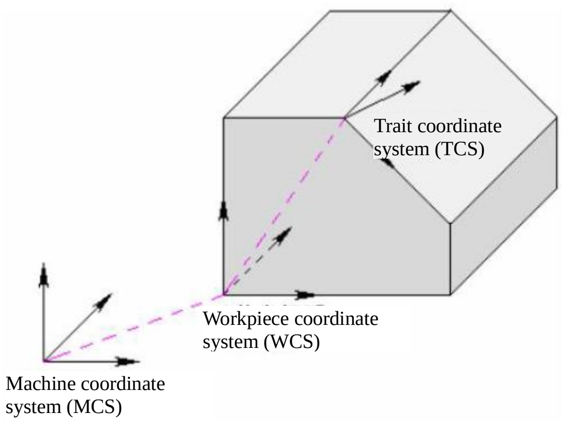



Figure 2-3 TCS diagram

TCS of workpiece can be designated in three ways:

1. Input data of TCS in the CNC interface and select a set of data with G68.1 command to build TCS;
2. Build TCS with G68.2 command through Eulerian angles. G69 is to cancel currently established TCS;
3. Using G68.3 command directly in the program to build TCS through spatial angle. G69 is to cancel currently established TCS.

Note:
1) Should enable function of TCS with G43.4/G43.5 before using TCS.
2) After TCS is built, all programming coordinates are coordinate values under TCS.

**1. Build TCS Through Three Points (G68.1)**

TCS can be built through specifying the following three points:

P1: Zero of TCS

P2: Any point in the positive direction of X axis of TCS

P3: Any point in the first and second quadrants of XY plane of TCS

(Coordinates of the above points are those in the workpiece coordinate system.)

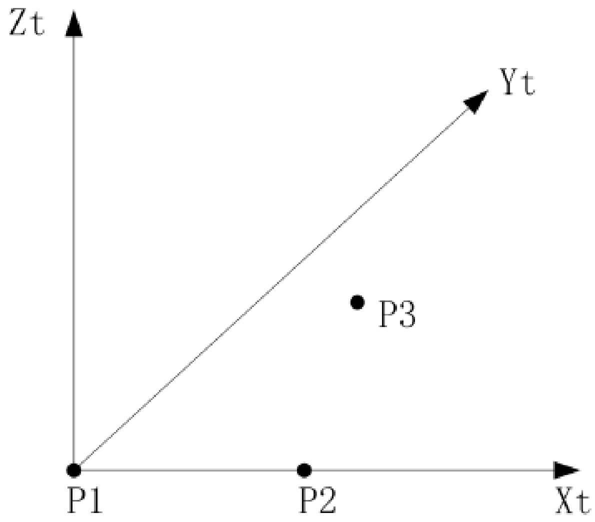



Figure 2-4 TCS diagram

P1, P2 and P3 can be inputted in the CNC interface. System supports 20 groups of TCS, and a set of parameters is selected with G68.1 to build TCS in the program

Programming format:

G68.1 Q_

Q: Select parameters to build TCS. The value ranges from1 to 20.

2. Build TCS Through Eulerian Angles (G68.2)

Eulerian angle is the rotation angle of coordinate axes around the rotation coordinate system, which is defined as follows:

1) Angle of precession (EULPR): Angle of rotating around Z axis.
2) Angle of nutation (EULNU): Angle of rotating around X axis changed by angle of precession.
3) Angle of rotation (RULROT): Angle of rotating around Z axis changed by angle of nutation.

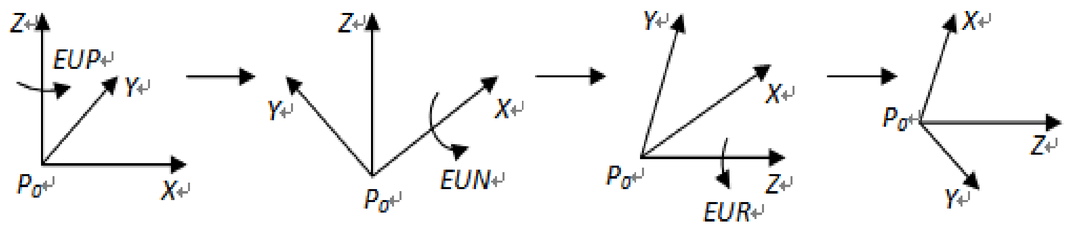



Figure 2-5 G68.2 reverse order diagram

**Programming format:**
G68.2 X_Y_Z_I_J_K ; Build TCS

**Command description:**

| | |
|---|---|
| X | X value under the workpiece coordinate system, and the origin of X axis in TCS |
| Y | Y value under the workpiece coordinate system, and the origin of Y axis in TCS |
| Z | Z value under the workpiece coordinate system, and the origin of Z axis in TCS |
| I | Angle of precession. Angle at which axis C rotates around Z axis in workpiece coordinate system |
| J | Angle of nutation. Angle at which axis A rotates around X axis in workpiece coordinate system after angle of precession changes. |
| K | Angle of rotation. Angle at which axis C rotates around Z axis in workpiece coordinate system after angle of nutation changes. |

**Detailed description:**
According to the command of G68.2 Xx Yy Zz Ia Jb Kc, CNC executes TCS as below:
(1) Regard Xx Yy Zz specified by G68.2 as the origin of TCS.
(2) The offset TCS rotates a° around Z axis.
(3) Then, rotate b° around X axis of the coordinate system.
(4) Finally, rotate c° around Z axis of the coordinate system.
(5) The coordinate system becomes a coordinate system built on a slope, namely TCS.

The rotation angle of the aforesaid coordinate system is based on the rotation relative to the positive direction of the rotation center axis (the anticlockwise direction of the rotation center axis is the positive direction).

**Application example**
NC program block format example

| | |
|---|---|
| G43.4 H1 | Enable tool center point control |
| G68.2X30. Y-40. Z0.0 I90. J45. K-90. | Build TCS using Eulerian angles |
| G53.2(G53.1) | The tool axis swings to the direction parallel with Z axis of TCS |
| G01 X0 Y-15 F2000 | Under TCS, X axis moves to 0mm and Y axis moves to -15mm |

**3. Build TCS Through Spatial Angle (G68.3)**
Like G68.2, G68.3 is also used to build TCS. However, IJK of G68.3 are angles of rotating around X axis, Y axis, and Z axis in the spatial coordinate system.

Command format:

| | |
|---|---|
| G68.3X_Y_Z_I_J_K_ | ; build TCS |
| G69 | ; cancel TCS |

| | |
|---|---|
| X | X value under the workpiece coordinate system, and the origin of X axis of TCS |
| Y | Y value under the workpiece coordinate system, and the origin of Y axis of TCS |
| Z | Z value under the workpiece coordinate system, and the origin of Z axis of TCS |
| I | Angle at which axis A rotates around X axis in workpiece coordinate system |
| J | Angle at which axis B rotates around Y axis in workpiece coordinate system |
| K | Angle at which axis C rotates around Z axis in workpiece coordinate system |

**4. Tool Axis Direction Control**

For slope machining, axis direction of tool is defined by G43.4 (or G43.5), and tool axis direction control command should be executed. The CNC system can control tool axis direction with G53.1 or G53.2 to swing the tool axis to the direction parallel with Z axis of TCS (namely vertical to XY plane of TCS). After TCS is built by G68.1/G68.2/G68.3, users can specify G53.1/G53.2 to swing tool axis to the direction parallel with Z axis of TCS.

**Programming format:**

G53.1/G53.2

**Command description:**

When G53.1 command is executed, only the rotary axis moves, and the linear axis has no compensating motion; When G53.2 command is executed, rotary axis moves, and the linear axis has compensating motion to keep the relative position unchanged between tool nose and workpiece.

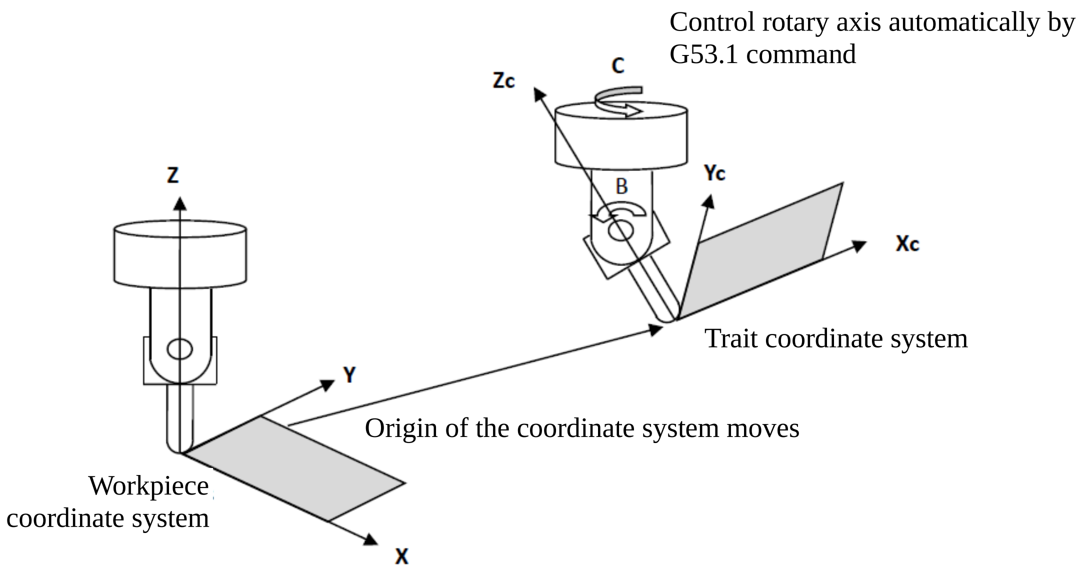



Figure 2-6 G53.1 tool axis direction control

When G53.1 is used, A and C axes move, and linear axes X, Y and Z axes do not move. Make the direction of tool axis be Z axis of TCS.

When G53.2 is used, X, Y, Z, A, and C axes move simultaneously, that is, tool nose follows the fixed position. Make the direction of tool axis be Z axis of TCS.

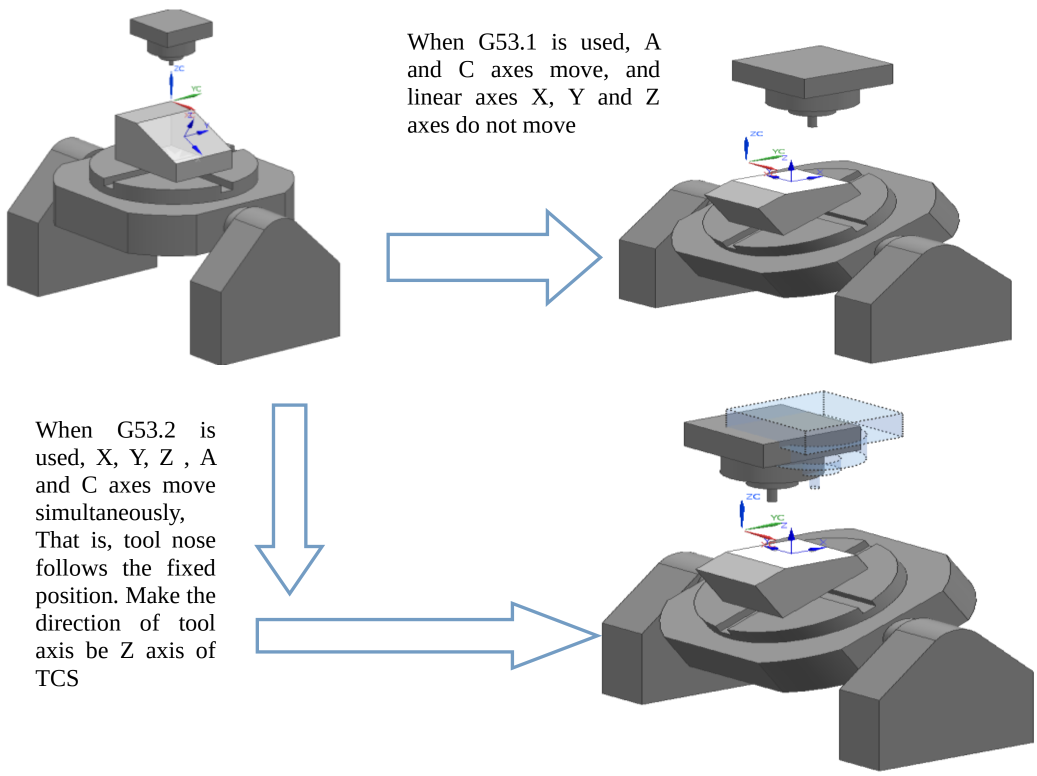



Figure 2-7 Difference between commands G53.1 and G53.2 in AC dual rotary table machine tool

NC program block format example

| | |
|---|---|
| G43.4 H1 | Enable tool center point control |
| G68.2X30. Y-40. Z0.0 I90. J45. K-90. | G68.2 command builds TCS |
| (G68.1 Q1) | (G68.1 command builds TCS) |
| G53.1 | The tool axis swings to the direction parallel with Z axis of TCS, and A and C axes moves simultaneously. |
| (G53.2) | (The tool axis swings to the direction parallel with Z axis of TCS, and X, Y, Z, A, and C axes moves simultaneously) |
| G01 X0 Y-15 F2000 | |

Note:

1) Tool axis direction control must be used together with TCS and placed in the line following TCS, that is: command G68.1/G68.2/G68.3 builds TCS, and the following line must be G53.1 or G53.2 so that the tool axis is vertical to the slope. And it must be in a separate line; otherwise, rotation angle is not executed but linear axis moves only. The current moving coordinate value is not the specified value, or an alarm is given.

2) When the program executes the G68.1/G68.2 line during slope machining, movement is not produced but TCS is built, and when the program executes G53.1/G53.2, movement is produced and the movement speed inherits the F value. If there is only G00 in front of the program line, the parameter 040030 "Default feedrate of channel" will be executed.

3) If G53.2 is used and the machine position is not appropriate, an alarm "Axis overtravel" will be issued when the line is executed, or the interference between tool and workpiece or worktable may occur.

4) G68.2 X*Y*Z*I*J*K* P1/2, based on G68.2 command, if there are multiple solutions of motion direction for rotary axis to reach the specified inclined plane, P parameter can be used to designate positive and negative solutions to make a choice between CW and CCW rotation.

Namely, tool orientation angle of TCS has positive and negative values. System offers three choices to users:

Define the first rotation axis close to the tool as trait axis and use it as the criterion

If P=0, the trait axis rotates along the shortest path

If P=1, the trait axis rotates in the positive direction

If P=2, the trait axis rotates in the negative direction

## 2.3. Automatic Calibration

In order to overcome accuracy problems and using limitations of manual measurement, maximize automation of measurement, and realize accurate control of rotary tool center point, the system offers automatic calibration of key geometric dimensions of five-axis machine tool. During calibration, trigger probe and datum sphere are used, and structural parameters of five-axis machine tool is automatically calculated and generated by measuring macro program and fitting collected data point to improve measurement accuracy and efficiency. Automatic calibration is applicable to detecting A, B and C models defined by ISO/CD 10791-6. It is convenient to use and does not require professional operators to implement RTCP parameter calibration. Renishaw strain probe and datum sphere with magnetic stand are used for measurement, as shown below.

Figure 2-8 Trigger probe and datum sphere

Automatic parameter measurement module based on HNC-8 high-end CNC five-axis machine tool greatly improves measurement accuracy and measurement efficiency. Automatic measurement process is shown below.

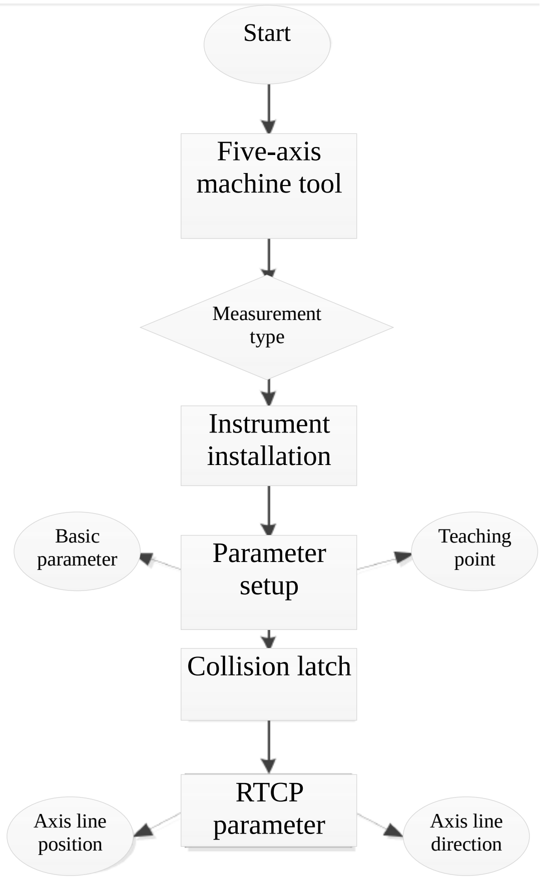



Figure 2-9 Automatic measurement process

Automatic measurement process of RTCP parameter is shown below.

**1. Parameter Setup**

Set measurement parameters including 10 basic parameters such as 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 as well as teaching points of master axis and slave axis. The software interface is shown in Fig. 2-10.

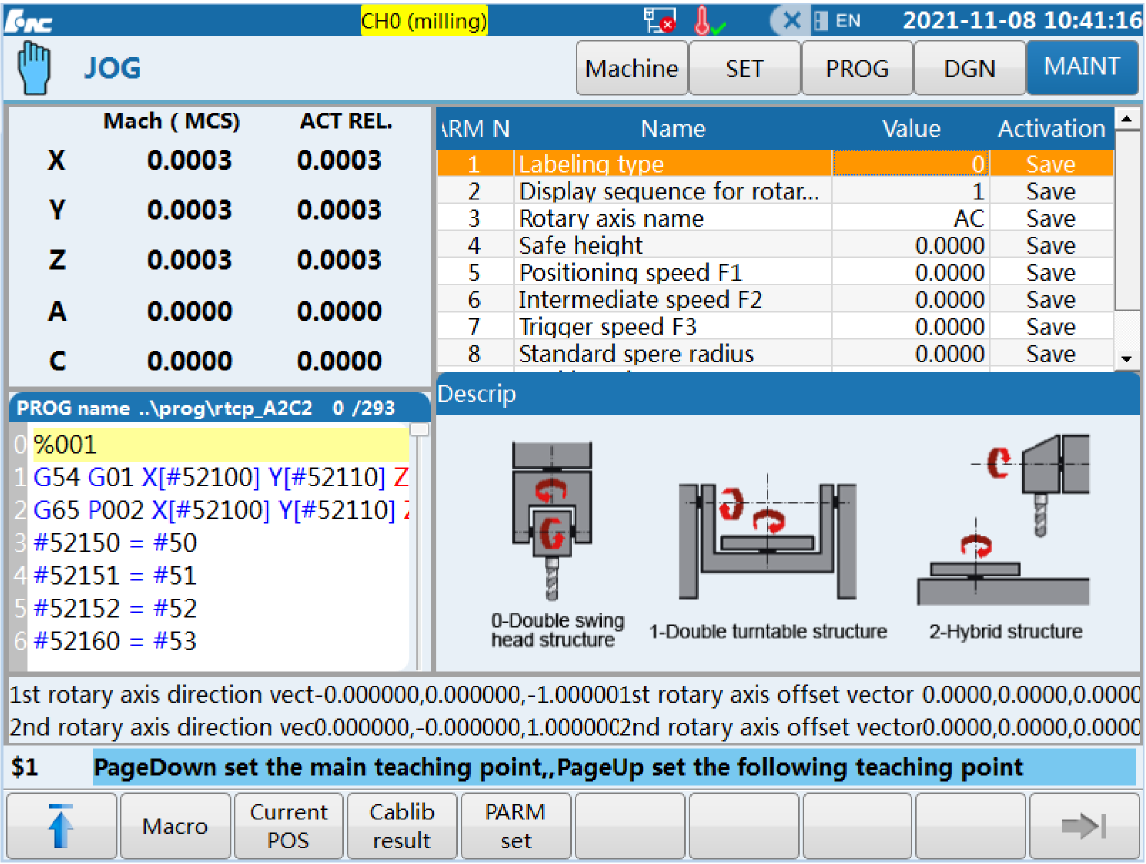



Figure 2-10 Automatic calibration interface

In basic parameters, "measurement type", "rotary axis display sequence", and "rotary axis name" are parameters for measuring attributes of five-axis machine tool; "safety height", "positioning speed", "intermediate speed" and "trigger speed" are parameters for controlling the collision of trigger probe with datum sphere; "datum sphere radius", "tool length", and "tool radius" are parameters for attributes of measurement gage.

**2. Acquisition of Teaching Points**

Teaching point: Teaching point is used to identify position of datum sphere on the circumference in order to roughly and automatically locate datum sphere after cycle start and ensure effective collision. To acquire teaching point, feed manually and take the highest point of datum sphere (the highest point by naked eye, not exactly).

Teaching point of different machine tools is acquired in different ways. When the probe is installed on the spindle, the datum sphere is on the worktable, and the probe just collides with the top of datum sphere, coordinates of machine tool at this moment is the teaching point, as shown in Fig. 2-11. When the probe is installed on the worktable, the datum sphere is installed on the spindle, and the probe just collides with the bottom of machine tool, coordinates of machine tool at this moment is the teaching point, as shown in Fig. 3-5. With teaching point as the reference point, if radius of datum sphere is known, the probe can be controlled to collide with the datum sphere at multiple points and the coordinates can be latched without interference in order to ascertain machine coordinates of datum sphere center.

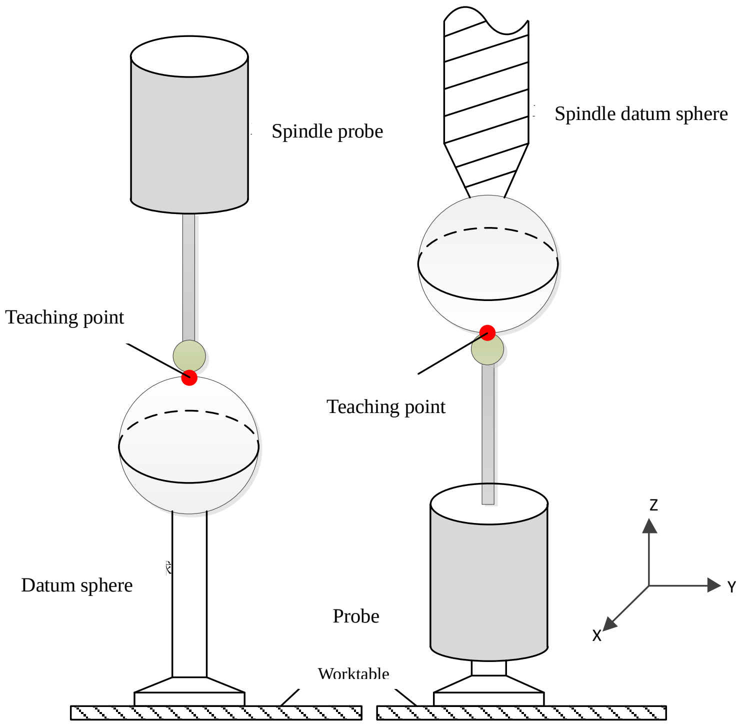



Figure 2-11 Teaching point of spindle probe (left) and teaching point of spindle datum sphere (right)

For dual swivel head five-axis machine tool that datum sphere is fixed on the spindle and rotates with swivel head, multiple teaching points can be obtained at different swing angles, as shown in Fig. 2-13 (number of teaching points is not restricted in different positions of datum sphere). For dual rotary table five-axis machine tool that datum sphere is fixed on the rotary table and rotates with the rotary table, multiple teaching points can be obtained at different angles (number of teaching points is not restricted in different positions of datum sphere). For machine tool with mixed structure swivel heads, select teaching point as shown in Fig. 2-13.

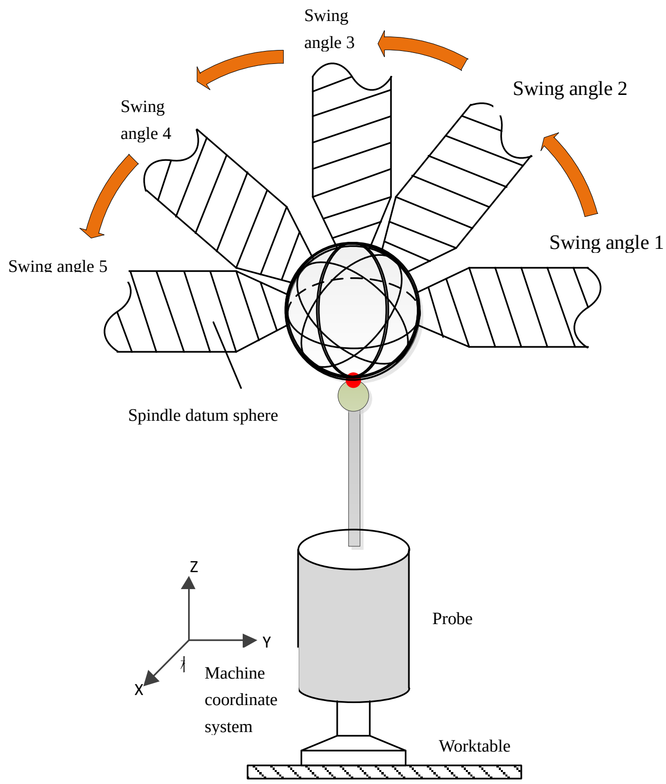



Figure 2-12 Multiple teaching points for dual swing head

Figure 2-13 Multiple teaching points for dual rotary table

Acquisition of teaching point (with C rotary table as an example): Place datum sphere on the rotary table using magnetic stand, feed the tool manually, estimate the highest point of sphere and record the teaching point 1. Rotate the rotary table an angle and acquire teaching point 2 in the same way, and so on, until 8 teaching points are acquired.

**3. Acquisition of Collision Point**

The measurement macroprogram controls collision between probe and datum sphere based on the set basic parameters and multiple teaching points of master axis and slave axis and latches coordinates X, Y and Z of collision points in the machine coordinate system. The probe collides with datum sphere at each teaching point.

The highest point (not strictly the highest point) and three points on the equator are obtained. Save these four points by macro-variable and calculate coordinates of sphere center on the HNC-8 secondary development platform. Coordinates of other 7 exact sphere centers at other teaching points are also obtained in the same way. Note: For easier programming, three points on the equator are distributed at a 90° angle, and the collision points of datum sphere are shown in Fig. 2-14.

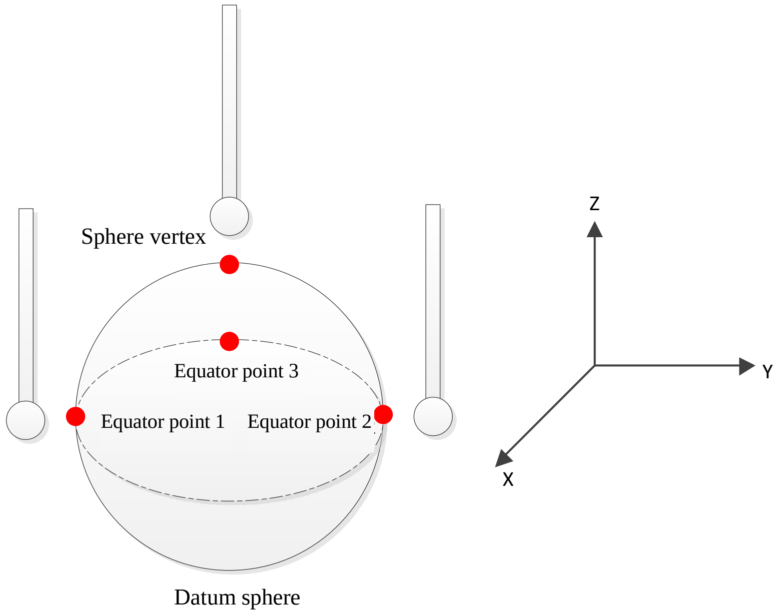



Figure 2-14 Diagram of collision points of datum sphere

**4. RTCP Parameter Calculation**

Coordinates of datum sphere center corresponding to each teaching point is calculated based on coordinates of latched collision point. For coordinates of each datum sphere center, fit axis direction and spatial position of master rotary axis and slave rotary axis using the least square processing, and finally figure out RTCP parameters including direction vectors of master axis and slave axis and offset vectors of master axis and slave axis.

Specific implementation is shown below:

**A. Installation of Renishaw probe**

In HNC-8, M40 and M41 commands are developed to turn on and off the probe. Fig. 2-15 is the electrical 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 skip command of the CNC system is activated by signal on the rising edge. PLC ladder of G31 command activation is shown in Fig. 2-16, 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 Fig. 2-17.

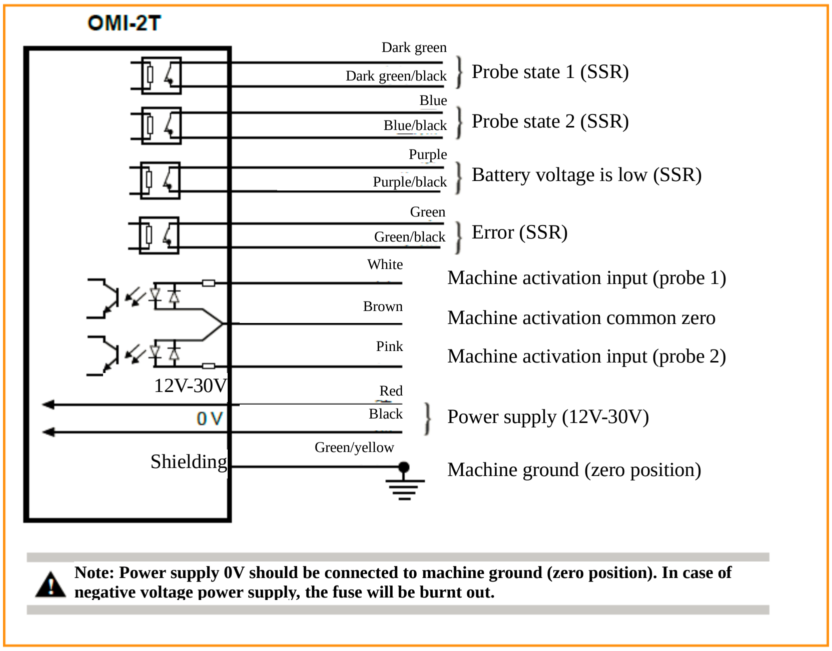
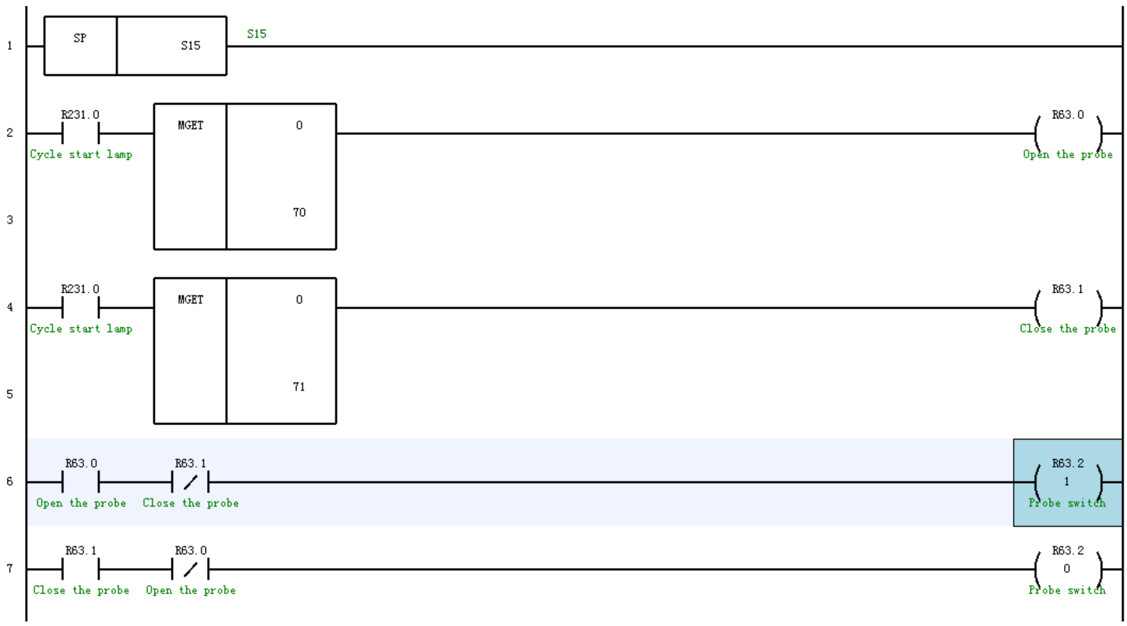



Figure 2-15 is the circuit diagram of signal receiver of probe

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

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


Figure 2-16 Activation of G31 block skip function



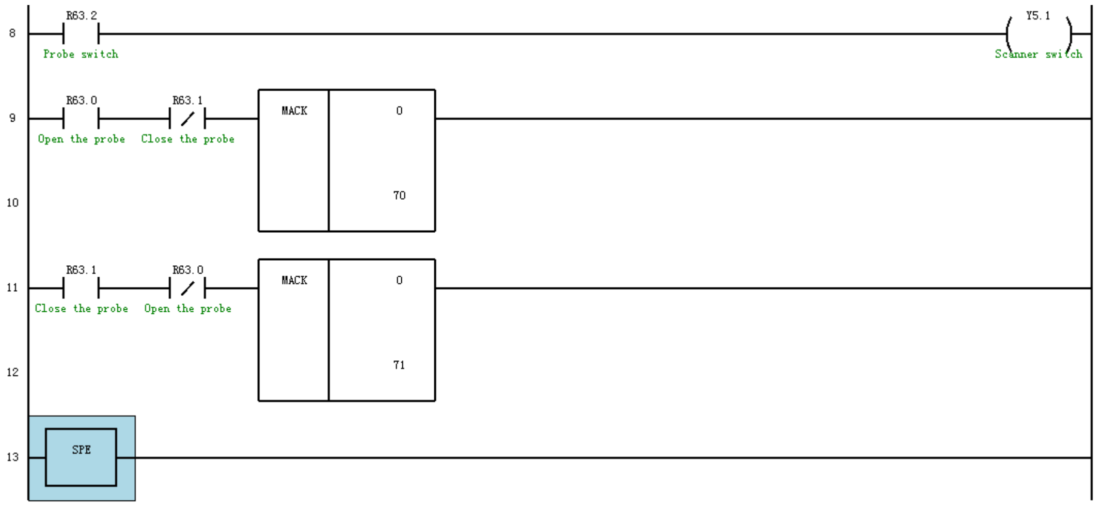



Figure 2-17 Probe turning on and off M70 and M71

**B. Click on Set- Machine calibration to enter the automatic machine calibration interface**

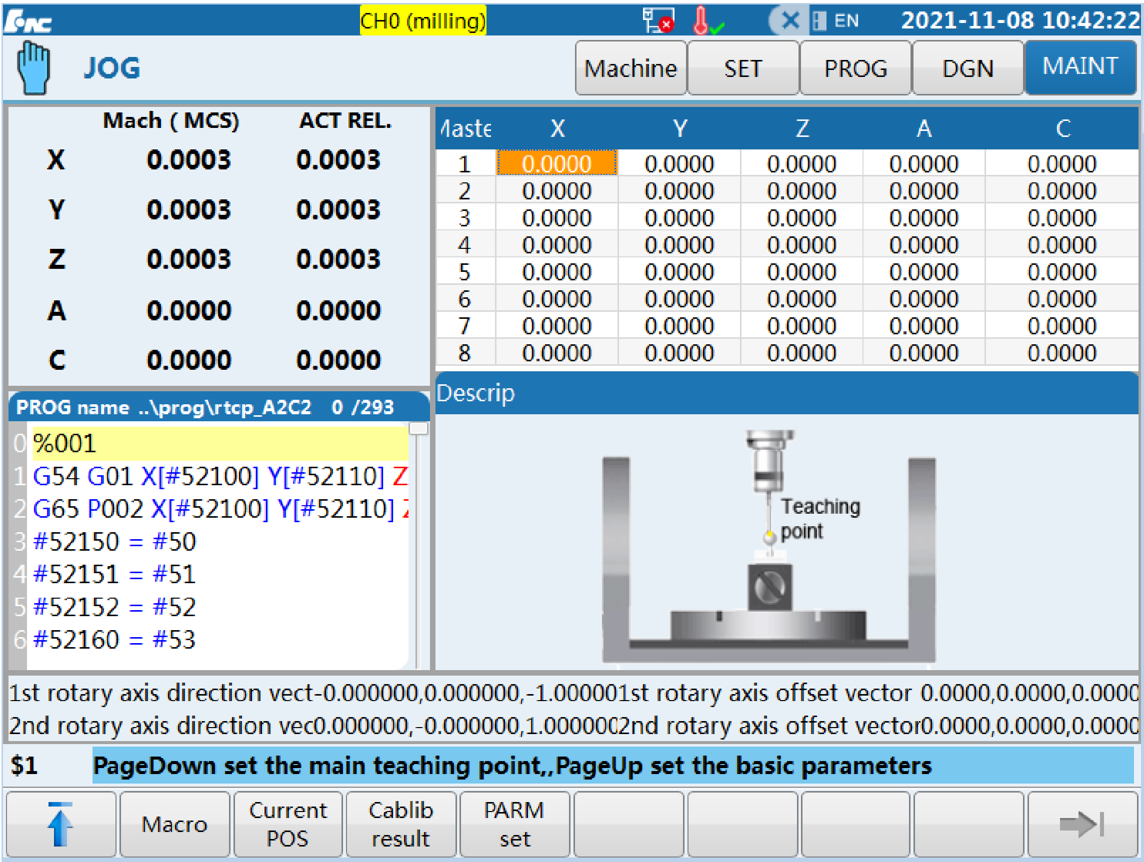



Figure 2-18 Automatic machine calibration interface

The software interface includes machine coordinate area, parameter setting 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:

(1) Machine coordinate area: To display coordinates of current machine tool and relative coordinates;

(2) Parameter setting 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.
(3) Macroprogram area: To display measurement macroprogram;
(4) RTCP parameter area: To display measurement results of RTCP parameters.

Function description:
(1) Current position: To acquire teaching points of master axis and slave axis, 5 coordinates of machine tool, and set them into corresponding parameter table in parameter setting area;
(2) Macroprogram: The measurement software loads corresponding measurement macroprogram of "Measurement type" in basic parameters, and displays it in macroprogram area;
(3) Measurement result: Macroprogram controls collision between probe and datum sphere and latches machine coordinates of collision point. System fits and calculates RTCP parameters according to coordinates of collision point and displays them in RTCP parameter area;
(4) Parameter import: Set measurement results of RTCP parameters in the CNC system to take effect.

**C. Fill corresponding parameters in the calibration parameter column based on actual need**

| Parameter | Notes to parameters |
| --- | --- |
| Calibration type | 0: Dual swivel head structure<br>1: Dual rotary table<br>2: 1 swivel head and 1 rotary table |
| Display sequence of rotary axis | 0: The second rotary axis is displayed in front<br>1: The first rotary 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 |

**D. Click on PageDown to collect teaching points of the first rotary axis**

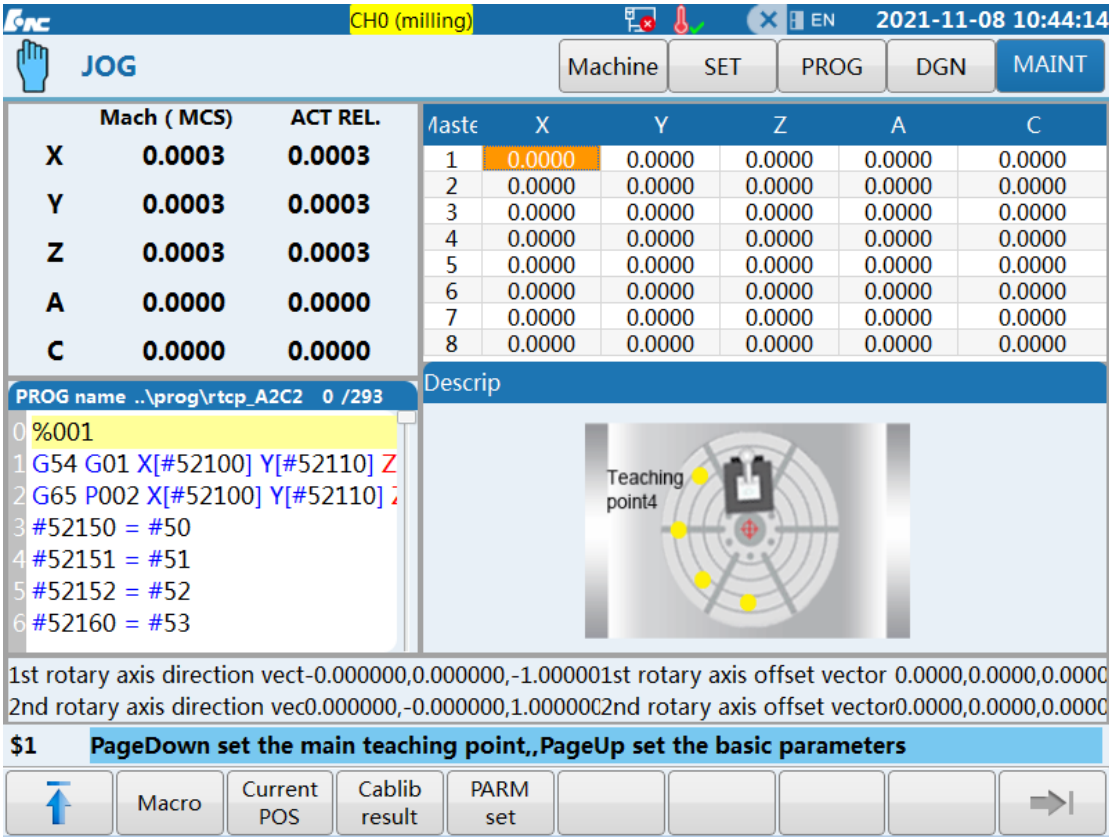



Figure 2-19 Collection of teaching point of the first rotary axis

**E. Click on PageDown to acquire teaching point of the second rotary axis**

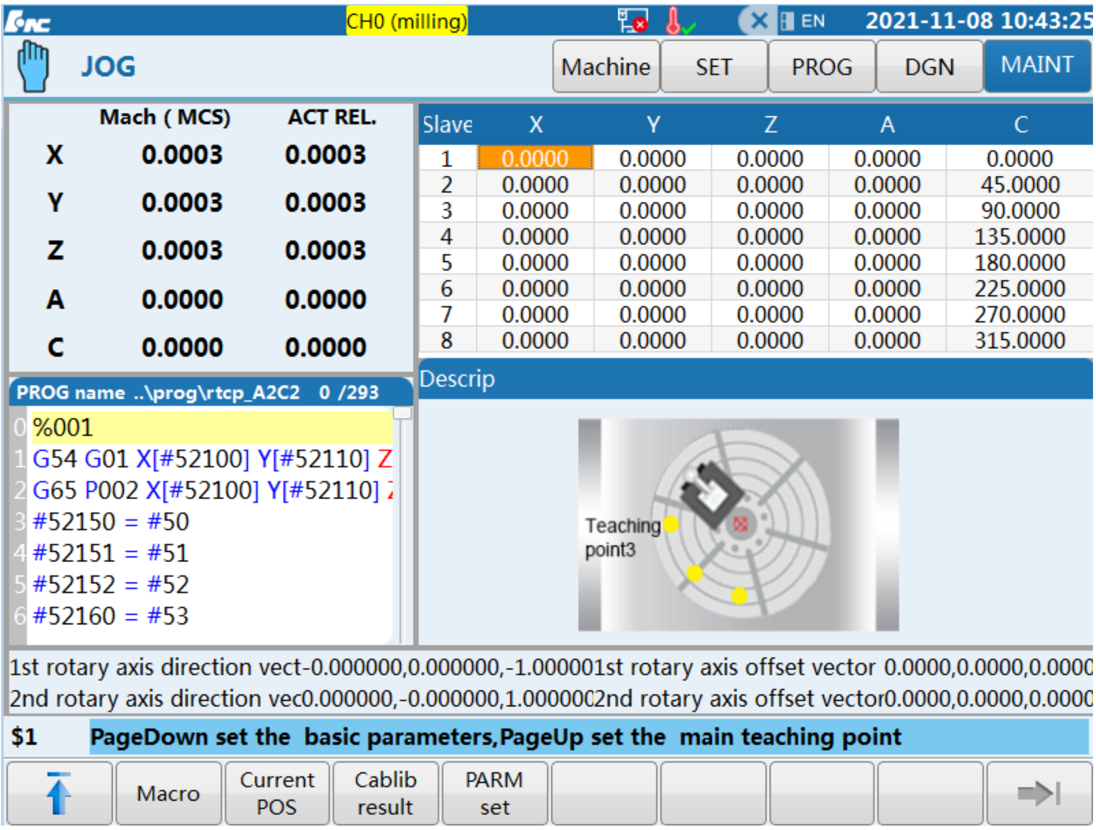



Figure 2-20 Acquisition of teaching point of the second rotation axis

**F. Click on Macroprogram to generate measurement program**

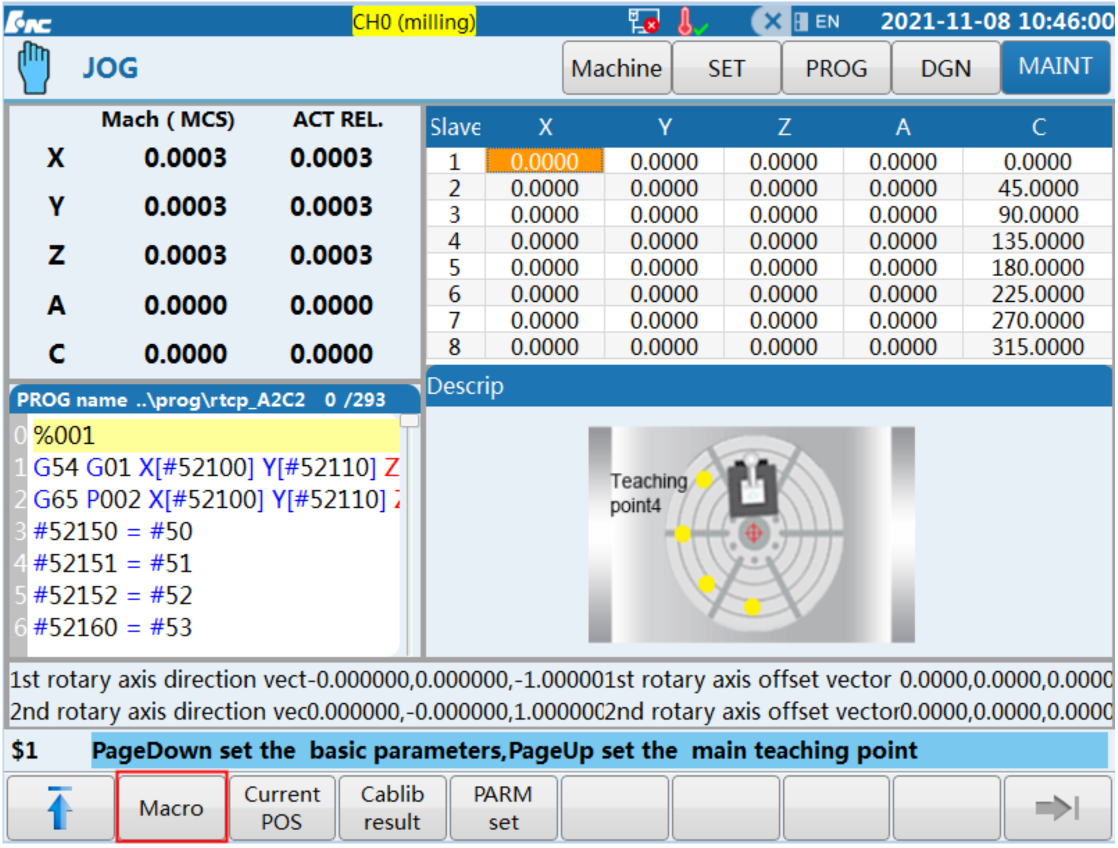



Figure 2-21 Generation of measurement program

**G. Press Cycle start and the system starts measurement automatically**

**H. After measurement, click on Parameter setting to write calibration results in the system**

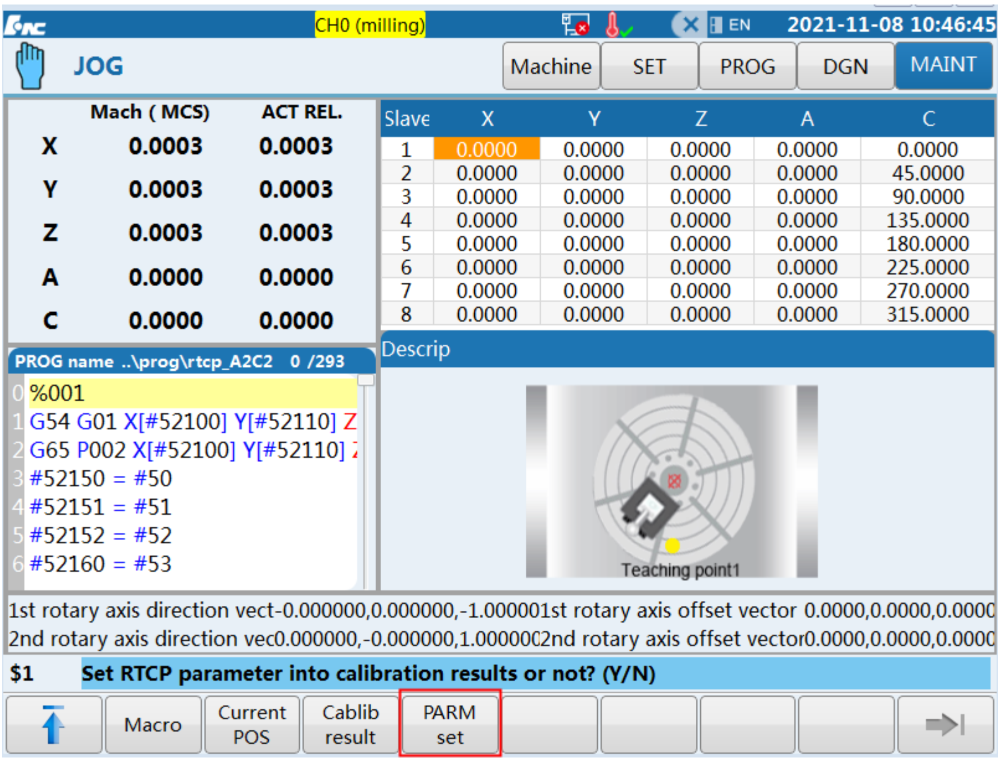



Figure 2-22 Writing of calibration result

Automatic measurement methods of structure parameters for simultaneous-five-axis machine tool are tested and verified. The measured structure includes dual swivel head, dual rotary table, and hybrid type. RTCP positioning accuracy test is conducted for measurement results, which fully verify correctness and accuracy of automatic measurement methods.

During test, Renishaw Ø25 mm datum sphere is used for worktable, and its roundness tolerance is within 0.1um and diameter tolerance is ±1μmm, which meet the requirements for measurement accuracy and the impact on measurement results is negligible. The spindle adopts Ø25 mm datum sphere with integrated shank. The dial indicator points at the equator of datum sphere on the XY plane of the machine coordinate system. Rotate the spindle, and the indicator reads within 4um. The datum sphere also needs good concentricity of spindle to meet the requirements for measurement accuracy, and the impact on measurement results can be negligible.

While testing accuracy of RTCP parameter result, install datum sphere on the spindle of five-axis machine tool, have the dial indicator contact the the equator or bottom of datum sphere, and activate RTCP function. Perform the full travel rotation of slave axis, pause at different angles, and record the indicator readings; then perform the full travel rotation of master axis, pause at different angles, and record the indicator readings. Measurement accuracy of RTCP parameter is directly reflected in the indicator readings. If reading for both master and slave axes can be within 20um, it is considered that measurement results of RTCP parameters reach the high accuracy requirements.

### 2.3.1. Automatic Calibration of AC Dual Rotary Table Structure

#### 2.3.1.1. Calibration of machine tool

AC dual rotary table machine tool


Figure 2-23 Calibration of machine tool

#### 2.3.1.2. Calibration tool

Renishaw OMP-600 probe

Datum sphere: Renishaw diameter 25mm

Figure 2-24 Calibration tool

### 2.3.1.3. Calibration process

**A. Installation of Renishaw probe**

In HNC-8, M40 and M41 commands are developed to turn on and off the probe. Fig. 2-25 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 Fig. 2-26, 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 Fig. 2-27.

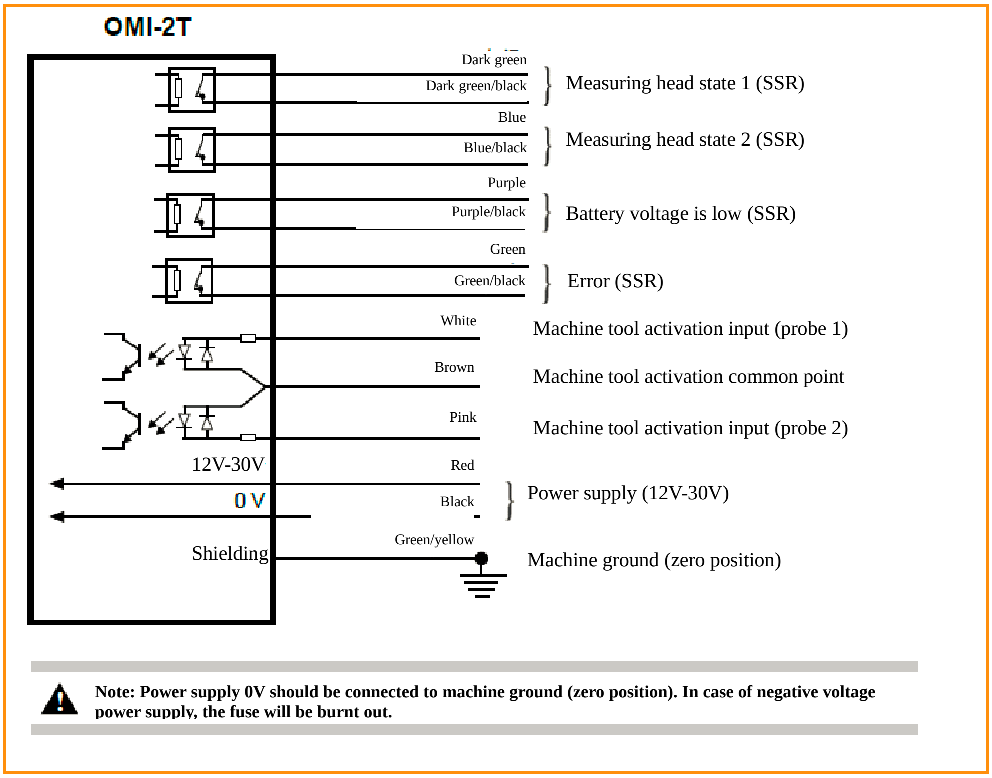



Figure 2-25 Electronical wiring of probe signal receiver

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 2-26 Activation of G31 block skip command

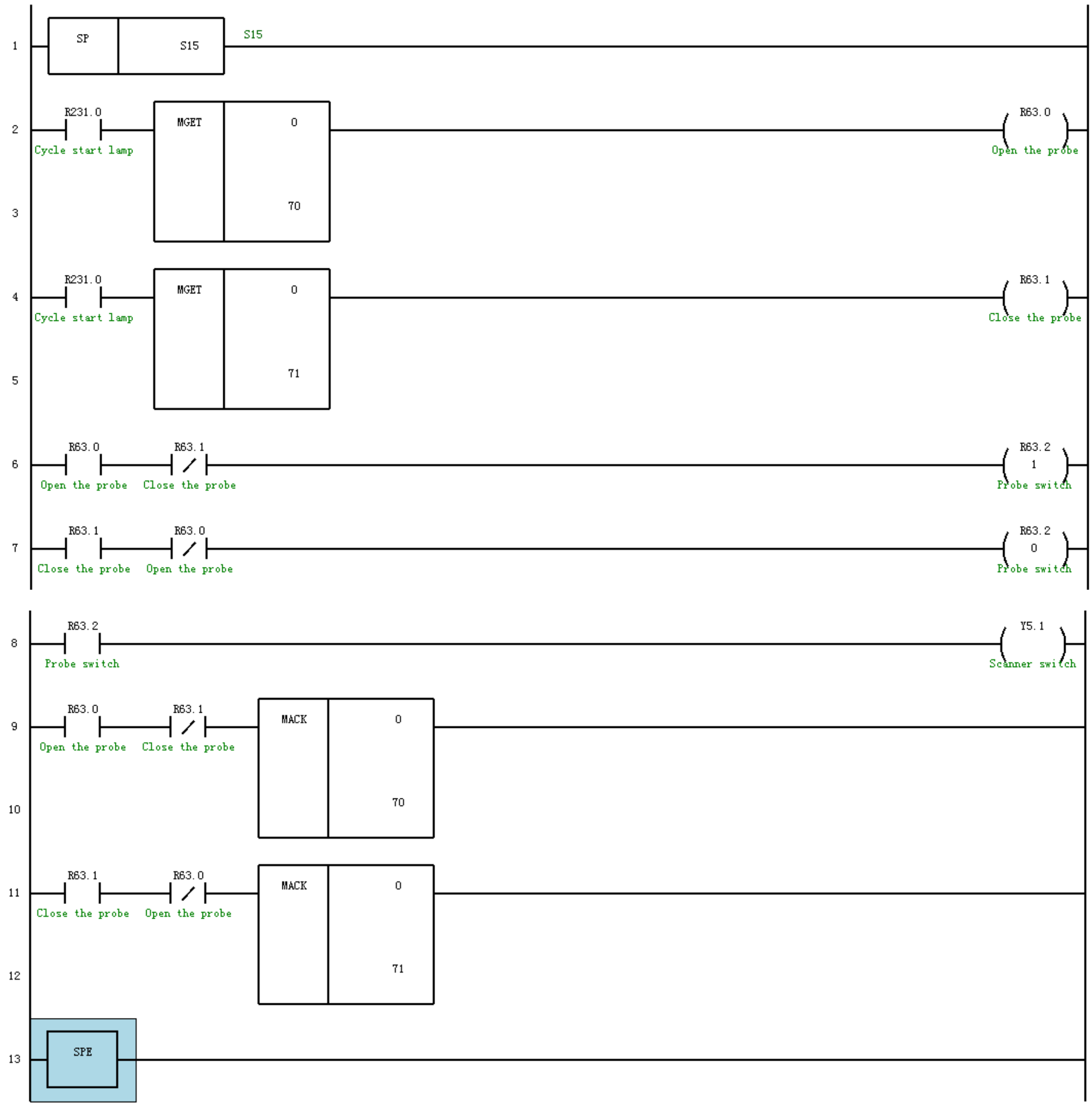



Figure 2-27 Turning on and off of M70 and M71 probes

**B. 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 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

**C. Spindle concentricity calibration**

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

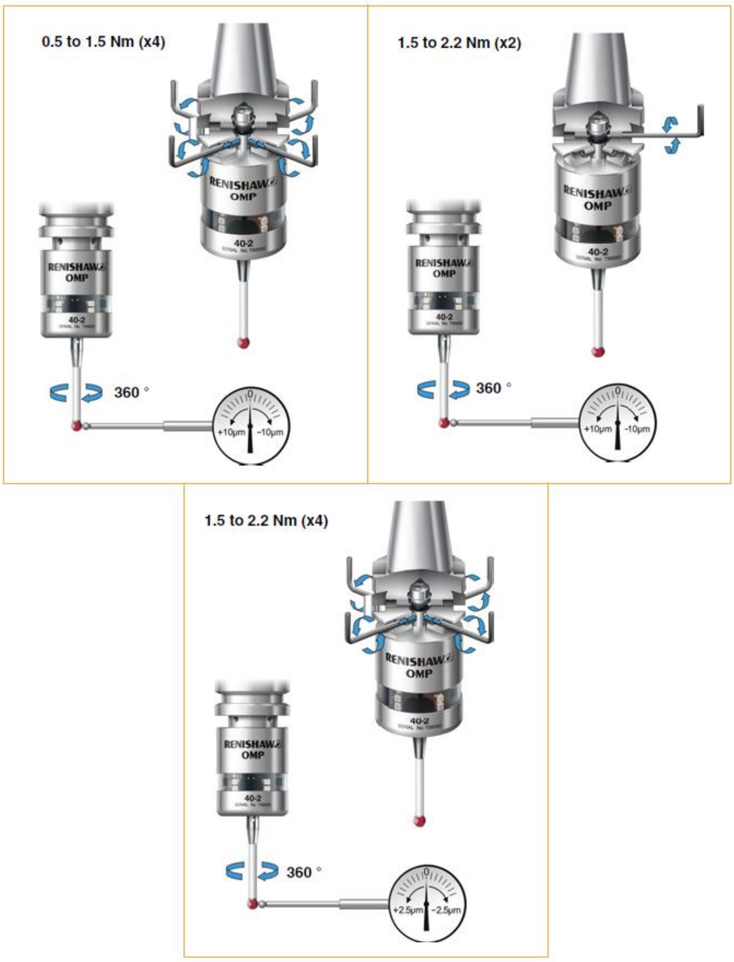



Figure 2-28Fig. 6-7 Probe calibration

**D. Automatic Calibration Test**

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

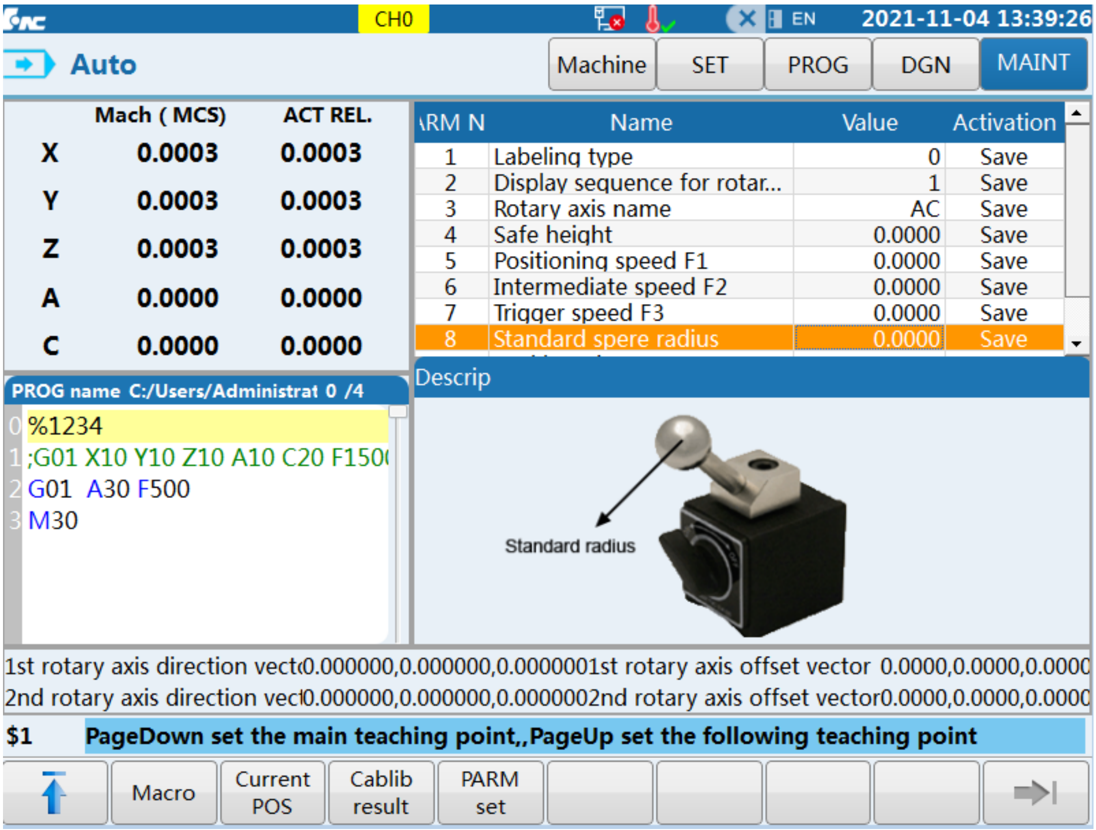



Figure 2-29 Automatic machine calibration interface

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:

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

2) 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.

3) Macroprogram area: To display measurement macroprogram;

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

Function description:

1) 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";

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

3) 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";

4) 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<br>1: Dual rotary table<br>2: 1 swivel head and 1 rotary table |
| Display sequence of rotary axis | 0: The second rotation axis is displayed in front<br>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 acquire 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.

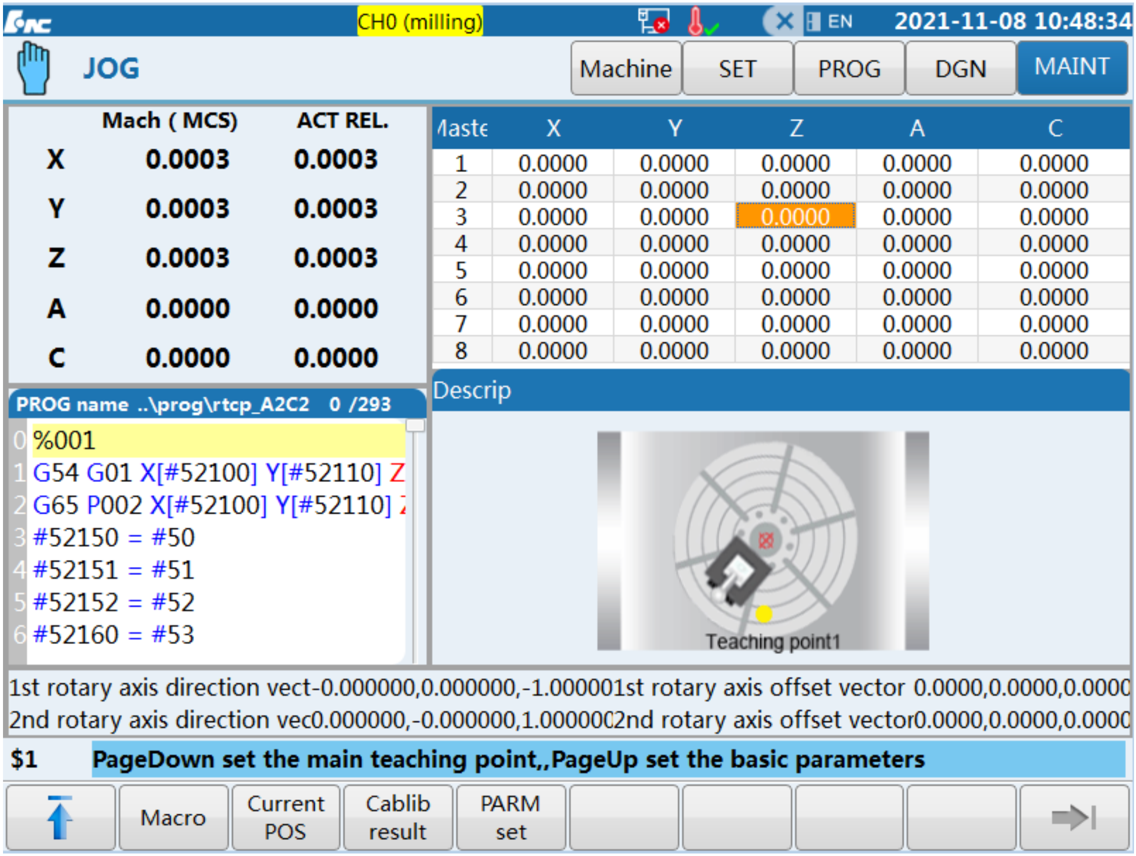



Figure 2-30 Acquisition of teaching point of the first rotation axis

(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.

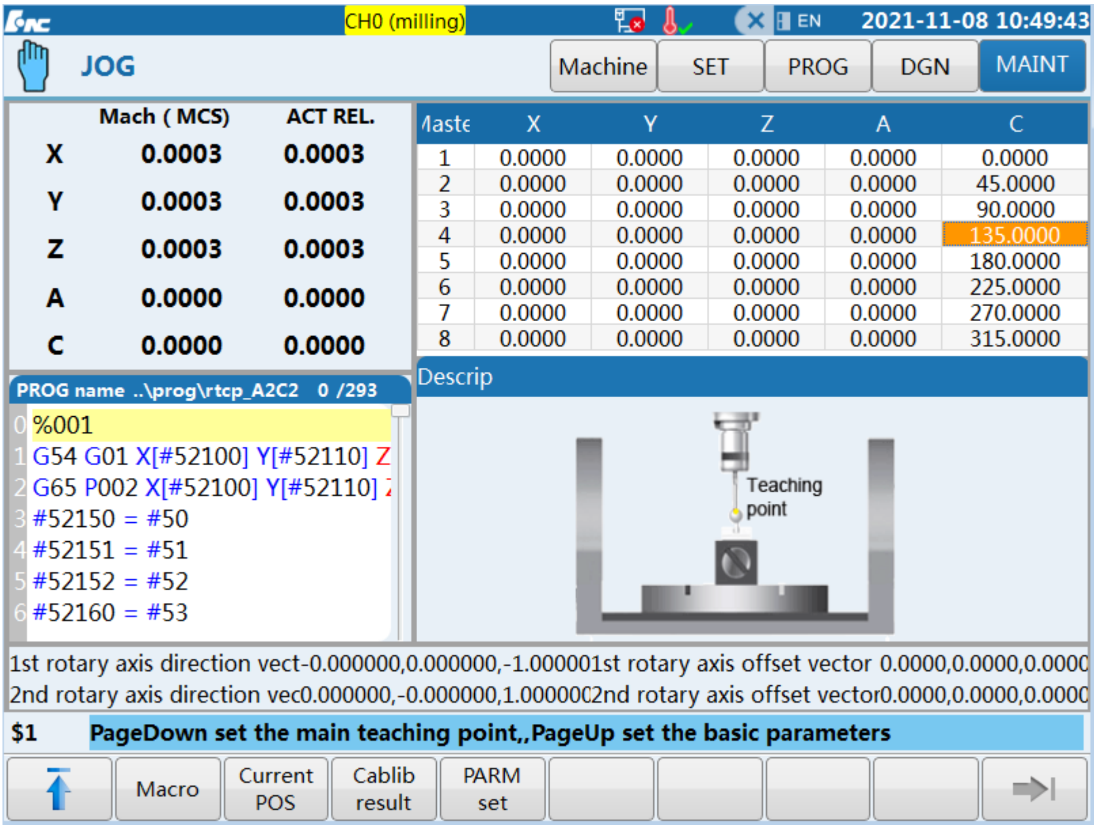



Figure 2-31 Acquisition of teaching point of the second rotary axis

(5) Click on Macroprogram to generate measurement program

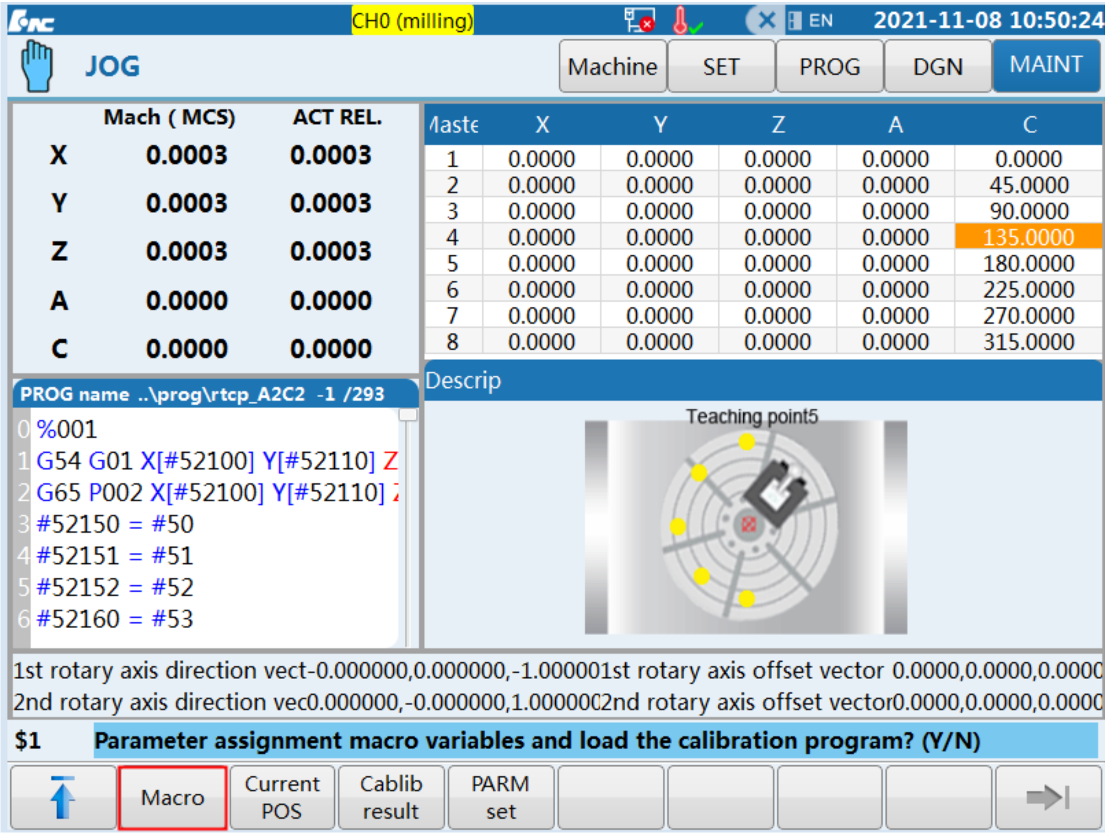



Figure 2-32 Generation of measurement program

(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

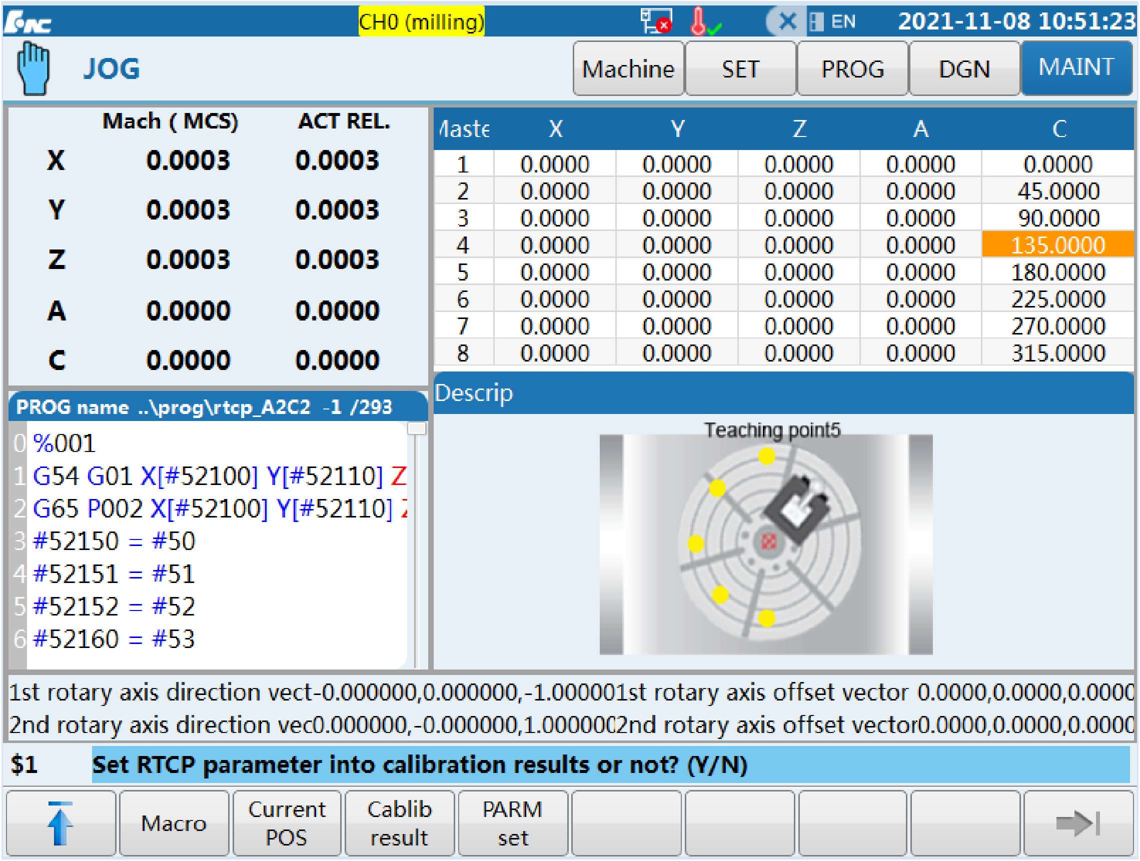



Figure 2-33 Writing of calibration result

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:

1) 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.

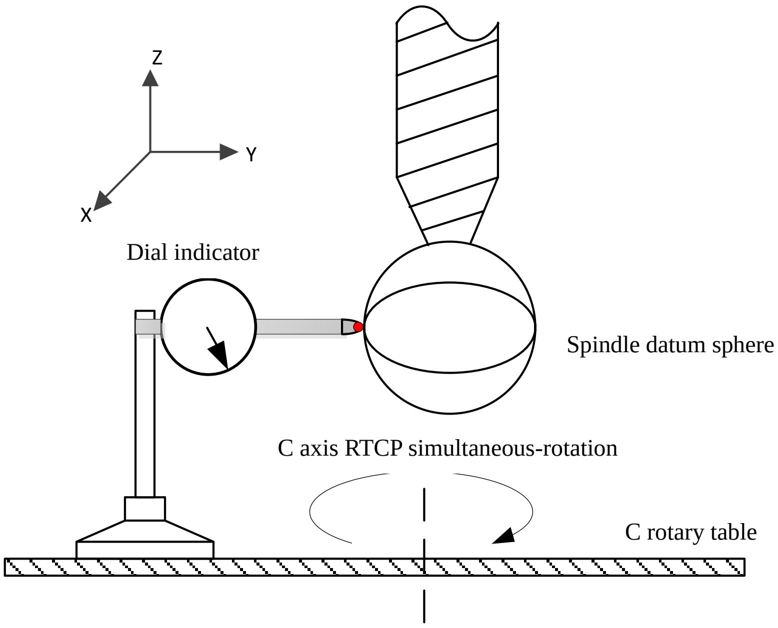



Figure 2-34 RTCP accuracy test of C axis forAC dual rotary table

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 |

2) 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.

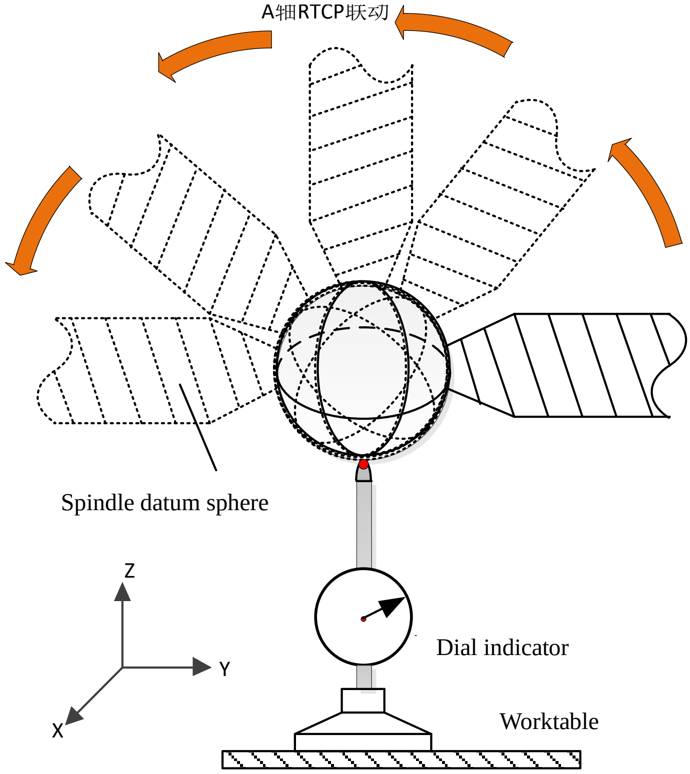



Figure 2-35 RTCP accuracy test of A axis for AC dual rotary table

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 |

## 2.4. PID Function

PID control (proportional-integral-derivative control) has been widely used in industrial control because of the following advantages:

1) Mathematic model of control object needs not known. Actually, accurate mathematic models of most industrial objects cannot be acquired. For this type of systems, PID control can give a satisfactory result. According to Japan's statistics, PID and modified PID account for about 90% of control loops.

2) PID controller has a typical structure, programming is simple, and parameter adjustment is convenient.
3) Strong flexibility and adaptability. Various control mode for PID control variation and improvement can be adopted based on specific situation of control object, such as PI, PD, PID with dead zone, integral separation PID, and variable speed integral PID. With the development of intelligent control technology, parameter self-tuning of PID controller can be realized through combining PID control with fuzzy control, neural network control and other modern control modes in order to ensure durability of PID controller.

**PID control with PLC**

As shown in the figure, it is the diagram of the system structure that adopts PLC to implement PID control on analog quantity.

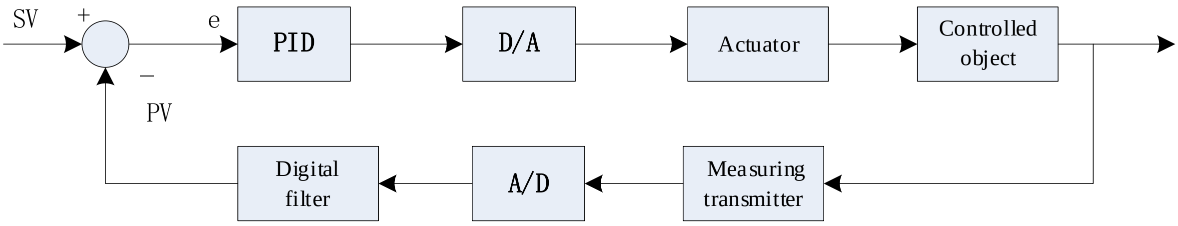



Figure 2-36 System structure diagram of PID control of analog quantity using PLC

Use of PID function instruction: Offer function instruction for PID control in PLC standard module, namely PID instruction of PLC. Actually, it is a subprogram used for PID control, and should be used cooperatively with A/D and D/A modules, which can achieve the effect similar to using PID process control module.

**PLC_PID instruction**

Module name of PID instruction is PID. Operands [S1], [S2], [S3], and [S4] are shown in the figure. [S1] and [S2] are used to store the given value of target value set and currently-measured sampling feedback value act. B registers from ［S3］to [S3]＋14 are used to store value of control parameter, and operation result out is stored in ［S4］. Variables corresponding to each B register will be described below.

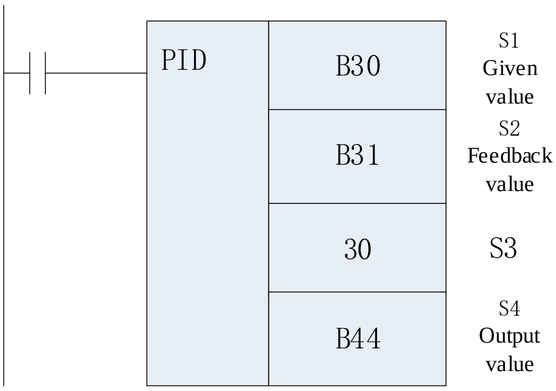



Figure 2-37 PID module diagram

### 2.4.1. PID Module Parameters

Operands [S1], [S2], and [S4] are B data registers, and operand [S3] is a constant.
[S1]: Target value.
[S2]: Sampling value.
[S3]: Occupy 14 B registers from [S3] (no more than total number of B registers).
[S4]: PID output result value.
Before PID operation is turned on, users should set the value of parameters to be set corresponding to [S3] - [S3]+13 of B register. If the data register with power failure holding function is adopted, it is unnecessary to write repeatedly. Before PID module is reused each time, reset [S3］+11 and［S3］+12 of B register. PID control

parameters and setups are shown below:

Table 2-1 Parameter comparison table

| B register | Parameter | Meaning and setting range |
|---|---|---|
| [S3] | Target value set | Desired value |
| [S3] +1 | Current sampling value act_n | Actual sampling value of this period |
| [S3] +2 | Previous sampling value act_l | Sampling value of the previous period (used for first-order filter) |
| [S3] +3 | Filter constant L | Input filter constant: 0-99(%) (no filter action at the time of 0) |
| [S3] +4 | Sampling period Ts | Sampling period (Ts): 1-32767ms (1ms by default) |
| [S3] +5 | Proportional gain kp | Proportional gain (Kp): 1-32767 (%) |
| [S3] +6 | Integral time Ti | Integral time (Ti): 0-32767 (no integral action at the time of 0) Integral time |
| [S3] +7 | Derivative time Td | (Td): 0-32767 (no differential action at the time of 0) |
| [S3] +8 | Upper limit of output out_max | Set based on actual situation (no upper limit at the time of 0, 32000 corresponds to 10V or 20mA) |
| [S3] +9 | Lower limit of output out_min | Set based on actual situation (no lower limit at the time of 0, -32000 corresponds to -10V or -20mA) |
| [S3] +10 | PID action period Tc | PID action period (ms) (PLC1 period is 1ms by default) |
| [S3] +11 | Err_next | Sampling error of previous period (need to be cleared) |
| [S3] +12 | Err_last | Sampling error of the period before the previous period (need to be cleared) |
| [S3] +13 | Timekeeping Ntime | Start counting immediately after module is turned on |
| [S3] +14 | PID output out_s | PID control output value. |
| [S3] +15 | PID inverted output flag | 0 by default, modify it as 1 when inverted output of PID module is needed partially |

**PID module parameter setting**

1) Select 14 B registers to be used by PID module (It is assumed that registers B30 to B44 will be selected for PID module).

2) Assign values to the target value B30 of the PID module and the current sampling value B31 through the instruction similar to MOV in the PLC. These two values are used as the input of the PID module (the previous sampling value does not need to be set).

3) Set the value of B registers corresponding to seven parameters of PID module in the state display menu (B33-B40). The setting rules are shown in the Table.

4) Add PID module to functional module of PLC, and set four PID parameters from top to bottom as follows: B30, B31, 30 and, B44. Serial number of the first B register corresponds to the third parameter [S3] of PID module, so [S3] is set as 30.

5) When PID module is turned on, the value of B44 is the output of PID module.

Note: 1) PID instructions can be reused simultaneously, but B data register element number used for operation cannot be repeated.

2) Setting of control parameter and PID operation data must be entered within the standard range; otherwise, unknown error will occur.

**PID principle**

PID instruction adopts incremental PID algorithm, and control algorithm uses feedback first-order inertial digital filter, so that this instruction has a better control than common PID algorithm.

The most important thing in PLD control is the discrete incremental PID algorithm. PID operation formula is shown below:

$$\triangle lnc = kp*((err-err\_next) + (ts/ti)*(err) + (td/ts)*(err-2*err\_next+err\_last)) ;$$

$$Out+ = \triangle lnc$$

Where: △lnc is the difference between the PID output of the current and the previous sampling, out is the current PID output; e, e_next, and e_last are the difference between current sampling value and desired value, sampling value of previous period and desired value, and sampling value of the period before last period and desired value, respectively. Kp is proportional gain, Ts is sampling period, and Ti and Td are integral time and derivative time.

**PID Parameter Tuning**

4 major parameters Kp, TI, TD and, TS of PID controller need to be tuned. The control will be affected if any of them is selected inappropriately. During parameter tuning, users should grasp the relationship between PID parameter and dynamic and static performance of the system. Among the three controls P (proportional), I (integral) and D (derivative), proportion is consistent with error signal in time. In case of error, proportion adjustment that is proportional to the error can be produced promptly. The larger proportional coefficient Kp is, the better proportional control, and the higher stability precision of the system is. However, for most systems, output oscillation will be increased and stability will be reduced if Kp is too large.

Integral control is related to the current error and historical errors. As long as error is not 0, output of controller will constantly change due to the integral action until error disappears. The integral no longer changes when the system is at steady state. Thus, integral can eliminate steady-state error and improve control accuracy. However, integral acts slowly and may have a bad impact on dynamic stability of system. When integral time constant TI increases, the integral effect weakens, and dynamic performance (stability) of system may improve, but the elimination of steady-state error slows down.

Differential part shows a good adjustment in advance based on the pace of error change. Differential part reflects trend of the system changes and is more timely than proportional control, so differential has the characteristics of prediction. When differential time constant TD increases, overshoot decreases and dynamic performance improves, but it may decrease high-frequency interference suppression.

While selecting sampling period TS, it must be much smaller than pure lag time or rise time of system step response. TS should be as small as possible so that the sampling value can timely reflect change of analog. However, computational workload of CPU will increase if TS is too small, and the difference between two adjacent sampling values rarely changes. Thus, TS should not be too small and generally is set as 1ms.

**PID Parameter Adjustment**

PID is the abbreviation of proportional, integral and differential control. The difficulty of PID control is not to adjust parameters of PID to achieve the control according to the using on site. The key of parameter tuning is to correctly understand physical meaning of parameters. While tuning parameters of PID controller, parameter of controller can be adjusted by experimental methods according to the qualitative relationship between parameters of controller and dynamic performance and steady-state performance of system.

In order to reduce parameters to be tuned, pure proportional P controller can be adopted first. For the sake safety of system, conservative parameters should be set in the beginning of commissioning, for instance, proportional coefficient should not be too large to avoid exceptions such as system instability or excessive overshoot. If a signal is given, information of system performance can be obtained based on output waveform of controlled variable such as overshoot and adjusting time. PID parameters should be adjusted repeatedly based on the relationship between PID parameters and system performance.

If overshoot of step response is too large and it is stable after multiple oscillations or not stable at all, the proportional coefficient should be reduced; if overshoot of step response doesn’t occur, controlled volume rises too slowly, and transition time is too long, adjust parameters in the opposite direction. Then, add integral link for better control performance. Based on the above KP, set a large integral time constant and gradually reduce integral time based on system response until oscillation occurs. If error is eliminated slowly, properly reduce integral time and intensify the integral effect. Repeatedly adjust proportional coefficient and integral time until system is stable. PI control can satisfy general control needs. If overshoot is still large, add differential control and increase differential time gradually from 0. Repeatedly adjust parameters of proportional, integral, and differential parts.

In general, the adjustment of PID parameters is a comprehensive and interactive process. It is very important and essential to try again and again.

### 2.4.2. Example of PLC Control (Spindle Position Control of PID)

The following is the use of PID in PLC for spindle position control, the input of PID module is B30 and B31 and, the output is B44. (For parameter adjustment of relevant B registers, refer to Diagnosis -> State display)

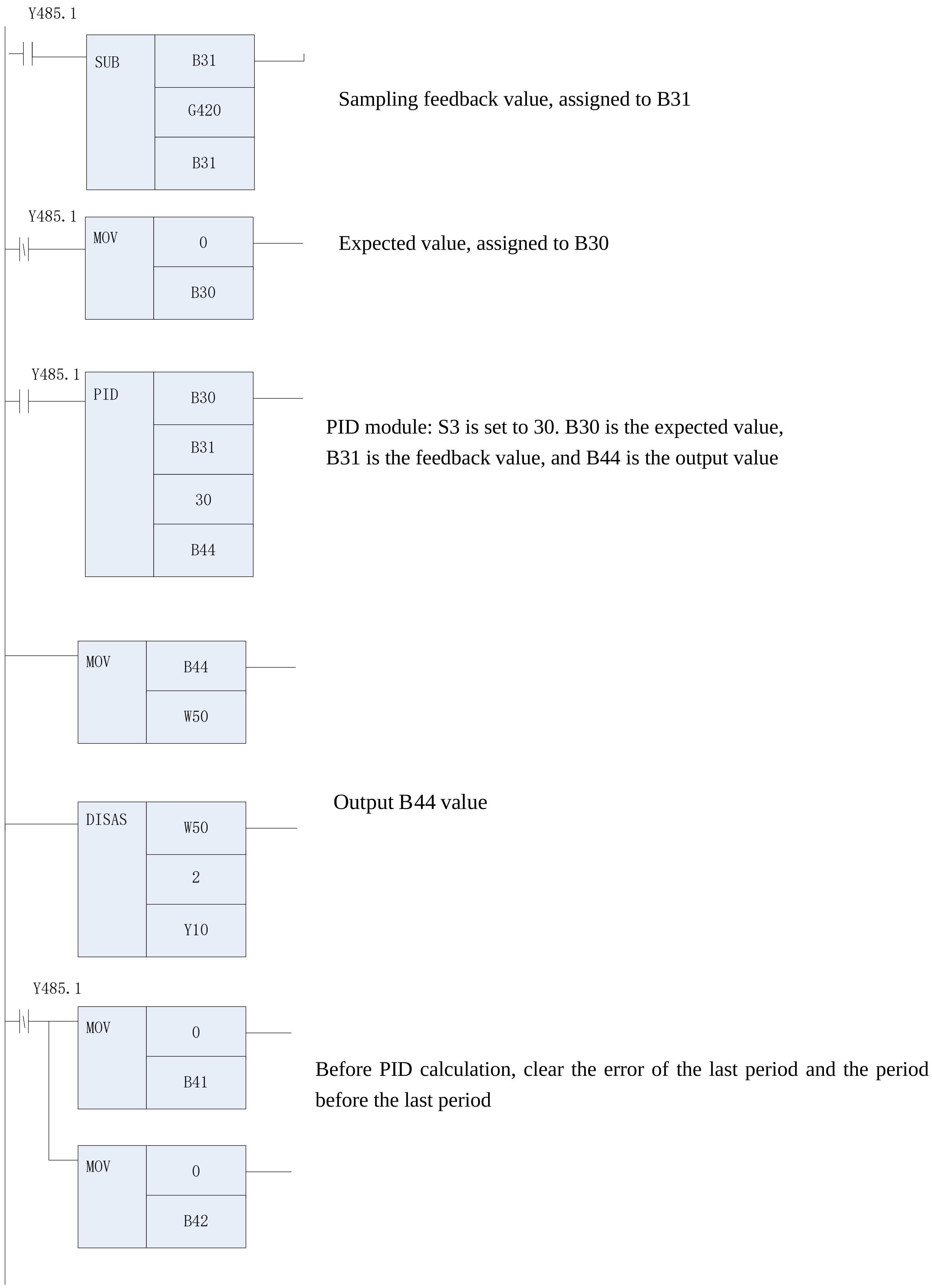



Figure 2-38 PID control example

## 2.5. Five-axis Rigid Tapping

In some applications of multiaxis, addresses of A, B and, C axes are occupied by actual axis, and rigid tapping needs to be conducted through U, V, or W programming. Type of tapping axis should be defined by J.
Set "Coordinate axis number" of tapping axis as -2 in channel parameter

**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 |

With U axis tapping as an example, set 040007 U coordinate axis number as -2

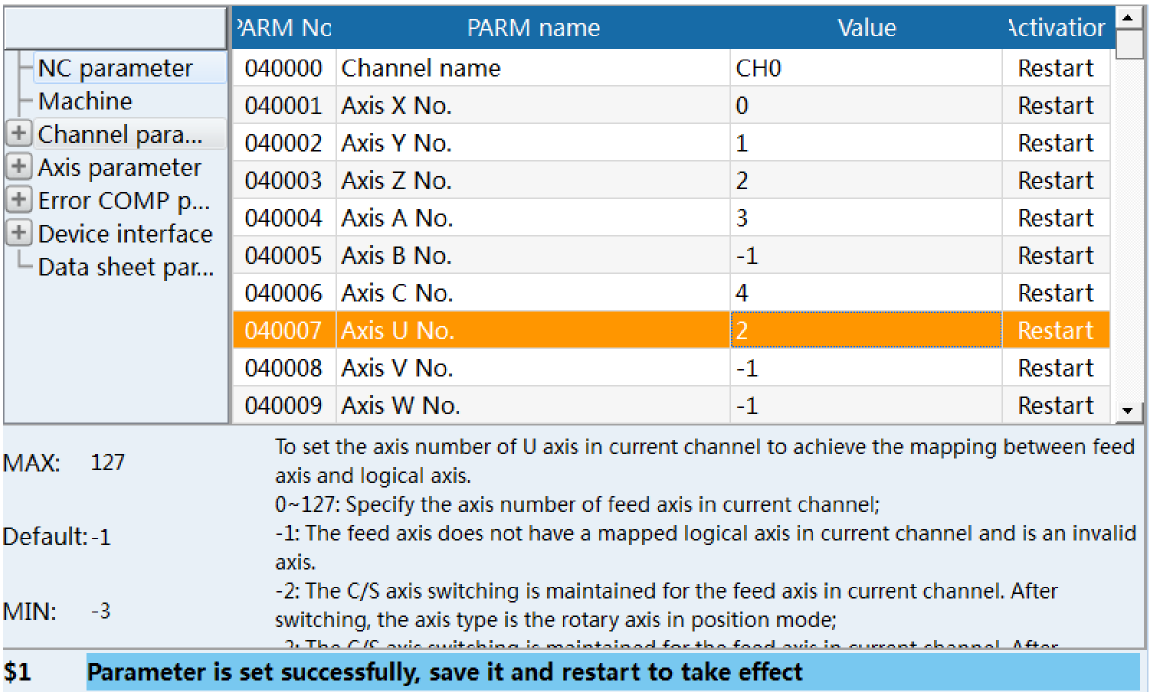



Figure 2-39 Parameter setting

## 2.6. Lathe Milling Combo Function

### 2.6.1. Lathe/Milling Conversion

Relevant parameters of the system should be modified for the lathe milling combo function, and the specific parameter are set as shown below.

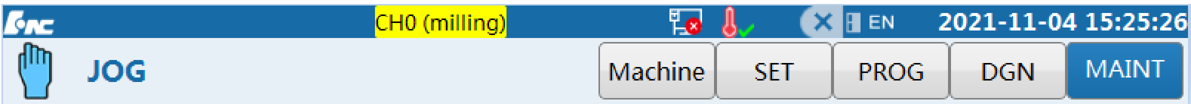
| Parameter type | Parameter | | Value | Description |
|---|---|---|---|---|
| NC parameter | 000065 | Lathe tool diameter display enable | 1 | X axis diameter display |
| Machine user parameters | 010001 | Cutting type of channel 0 | 2 | Lathe milling combo system |
| Channel parameter (channel 0) | 040032 | Diameter programming enable | 1 | Enable X axis diameter programming mode |
| Channel parameter (channel 0) | 040094 | Machine type of power off | 0 | After machine tool is powered on, it is selected as the working mode before power-off automatically |
| Channel parameter (channel 0) | 040101 | Number of tools on spindle | 2 | -- |

#### 2.6.1.1. Lathe Milling Switching Programming Command

LATHE: Lathe mode command
MILL: Mill mode command

#### 2.6.1.2. Visual Interface for Lathe/Milling Mode Switching

The upper tool bar displays that the current channel is in lathe or milling




#### 2.6.1.3. Graphics Simulation for Lathe Milling Combo

Click on "Path display" in the "Machining" menu to enter the graphics simulation interface

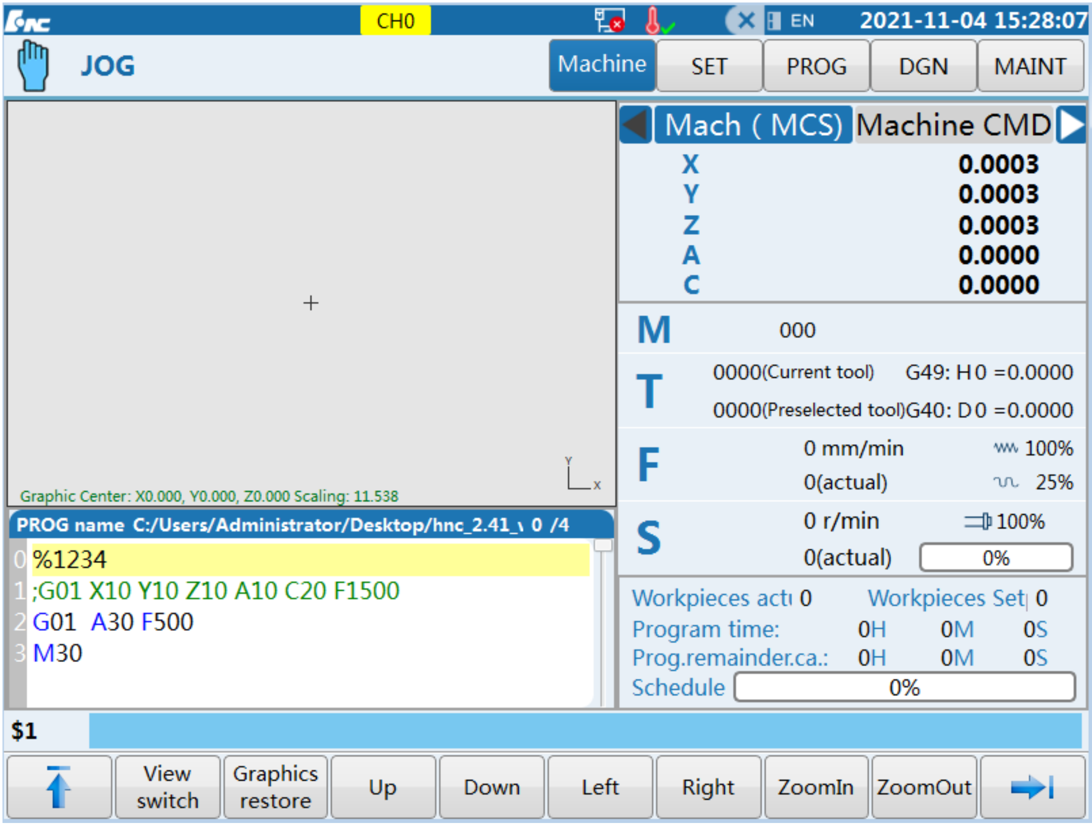



Figure 2-40 Path display

### 2.6.1.4. Selection of Machine Tool Working

Channel parameter "040094 machine type of power off"

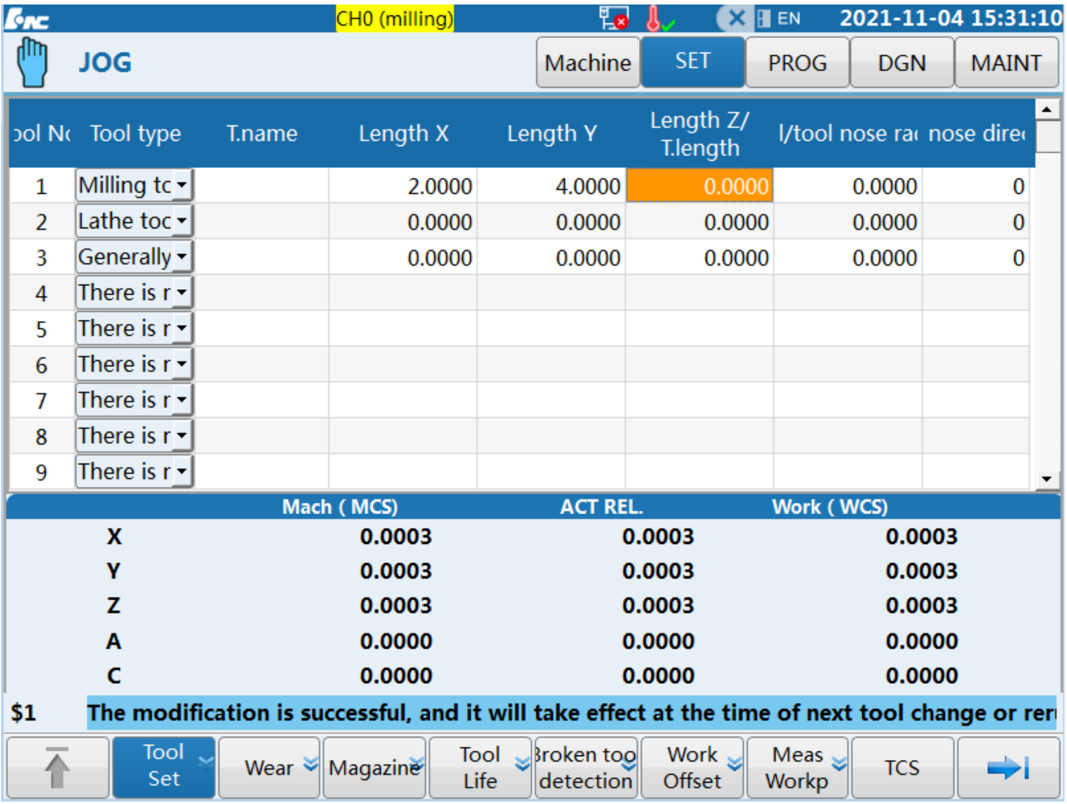
0: After the machine tool is powered on, the working mode before power-off is automatically selected (lathe or mill).

1: After the machine tool is powered on, current working mode is milling.

2: After the machine tool is powered on, current working mode is lathe.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 040094 | Type of machine powered off | 0 | Restart |

### 2.6.1.5. Management of Comprehensive Tool Table




Tool type:

No tool (ineffective tool), mill (effective tool), lathe (effective tool), and common tool (effective tool)

Where:

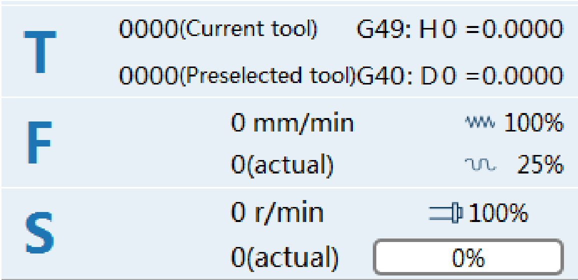
1. Other values than tool type cannot be set if there is no tool
2. When effective tool is set to ineffective tool, clear all information (length X-tool nose direction column) including wear value of the tool
3. Mill validation menu is: Current position (clear corresponding wear value), incremental input, relative actual (clear corresponding wear value). Invalid menus are not displayed
4. Common tool validation menu: Current position (clear corresponding wear value), incremental input, relative actual (clear corresponding wear value). Invalid menus are not displayed
5. Lathe validation menus:

Precutting diameter (clear corresponding wear value), Y axis precutting (clear corresponding wear value), precutting length (clear corresponding wear value). Invalid menus are not displayed

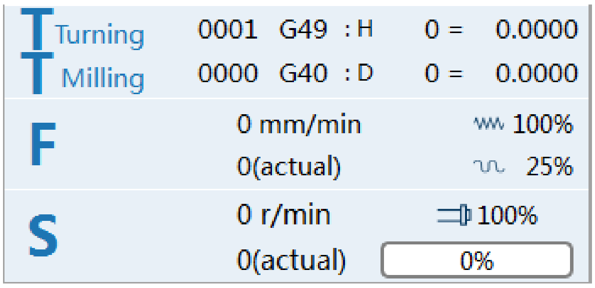
6. Clear all: Clear all tool information including wear

### 2.6.1.6. Number of Tools on Spindle

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

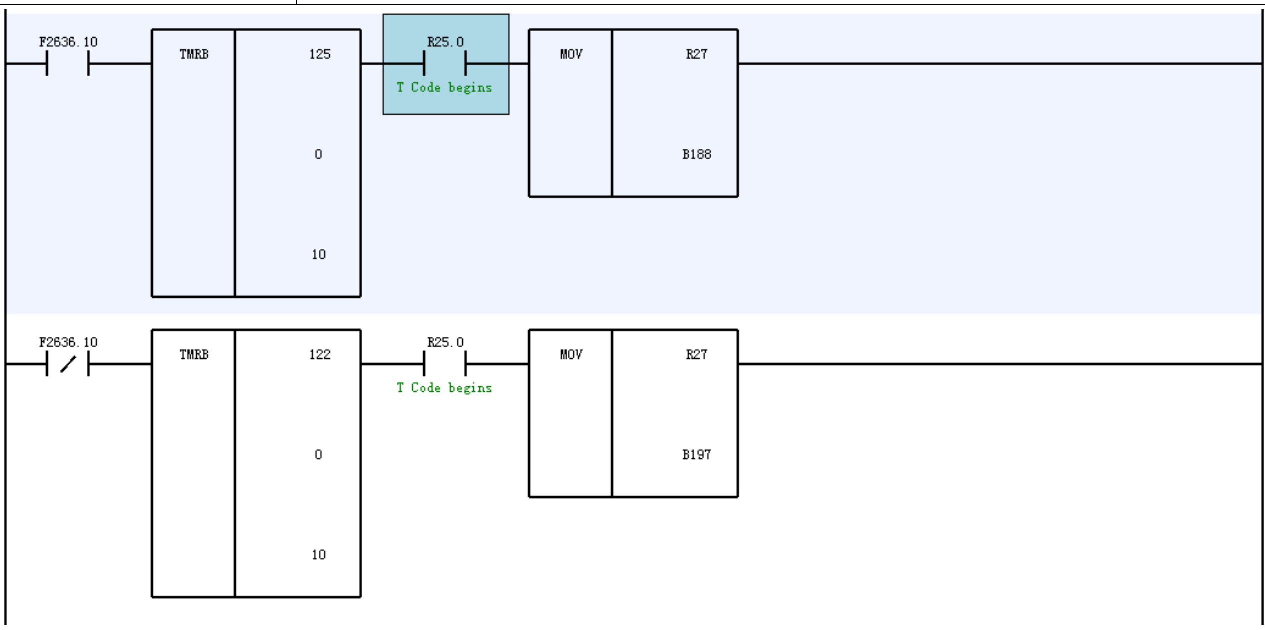



2. When channel parameter "040101 number of tools on spindle" is set to 2, current lathe tool and milling tool are simultaneously displayed in the T area display, as shown below:



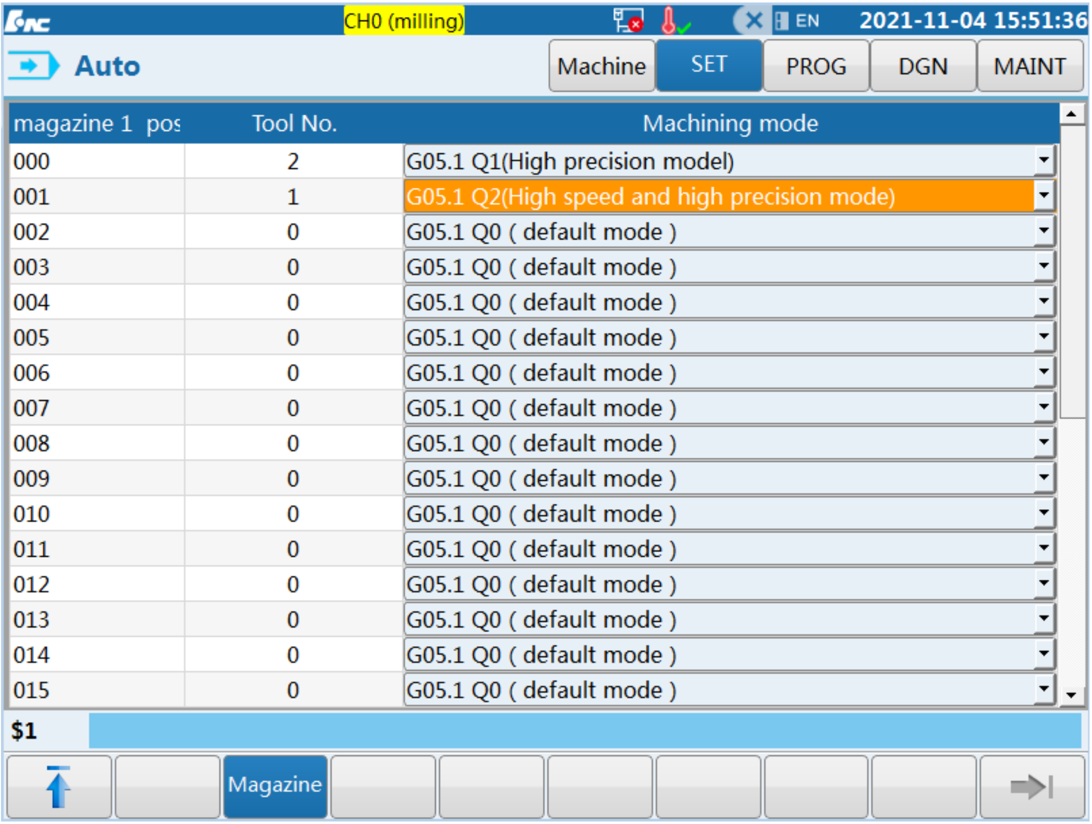

Tool number on spindle is acquired through reading register and PLC should be modified accordingly

| Register | Meaning | |
|---|---|---|
| F2636.10 | F236.10=0, it means that 0 channel is in milling mode | F236.10=1, it means that 0 channel is in lathe mode |
| B188 | Current tool number on spindle in lathe mode | |
| B197 | Current tool number on spindle in milling mode | |




### 2.6.1.7. Magazine Table display of Lathe Milling Combo

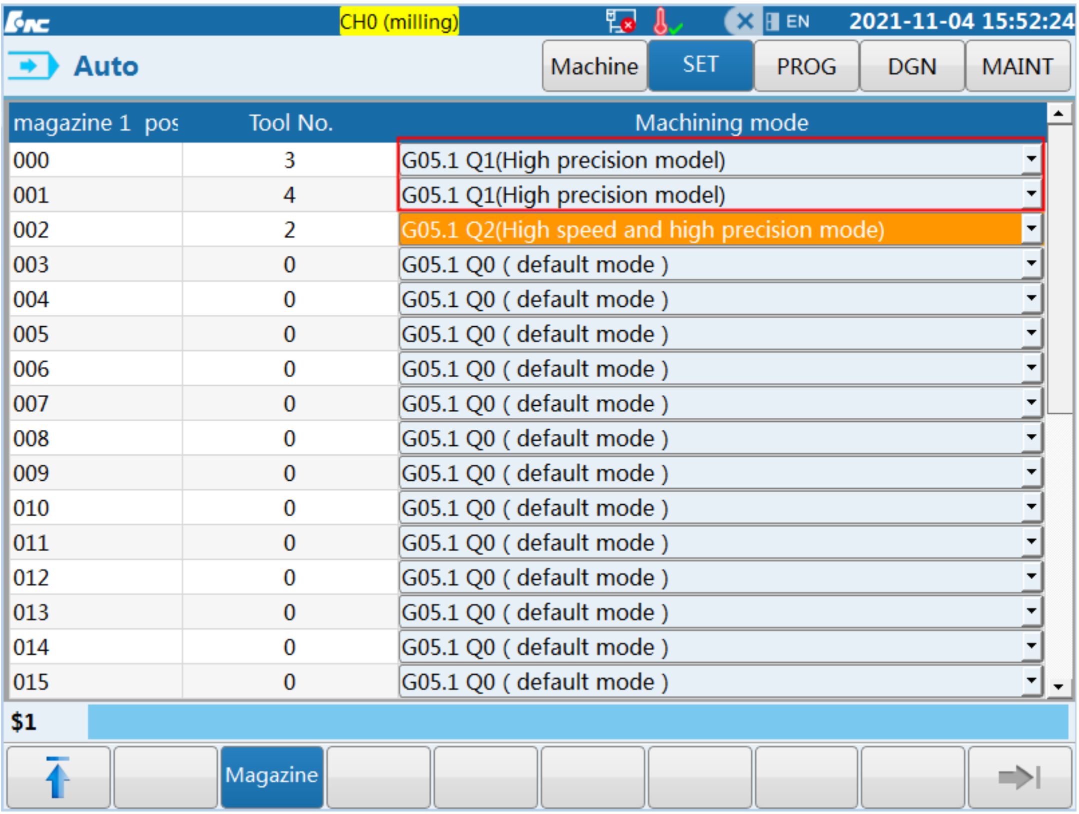
1. When channel parameter "040101 number of tools on spindle" is set to 1 (if it is not 2, it is treated as 1), it is displayed as below:




| magazine 1 pos | Tool No. | Machining mode |
|---|---|---|
| 000 | 2 | G05.1 Q1(High precision model) |
| 001 | 1 | G05.1 Q2(High speed and high precision mode) |
| 002 | 0 | G05.1 Q0 ( default mode ) |
| 003 | 0 | G05.1 Q0 ( default mode ) |
| 004 | 0 | G05.1 Q0 ( default mode ) |
| 005 | 0 | G05.1 Q0 ( default mode ) |
| 006 | 0 | G05.1 Q0 ( default mode ) |
| 007 | 0 | G05.1 Q0 ( default mode ) |
| 008 | 0 | G05.1 Q0 ( default mode ) |
| 009 | 0 | G05.1 Q0 ( default mode ) |
| 010 | 0 | G05.1 Q0 ( default mode ) |
| 011 | 0 | G05.1 Q0 ( default mode ) |
| 012 | 0 | G05.1 Q0 ( default mode ) |
| 013 | 0 | G05.1 Q0 ( default mode ) |
| 014 | 0 | G05.1 Q0 ( default mode ) |
| 015 | 0 | G05.1 Q0 ( default mode ) |

In the table, tool number and machining mode can be set. When tool number is 0 or tool type is "No tool", machining mode cannot be set.

2. When channel parameter 40101 Number of tools on spindle is set as 2, it is displayed as below




| magazine 1 pos | Tool No. | Machining mode |
|---|---|---|
| 000 | 3 | G05.1 Q1(High precision model) |
| 001 | 4 | G05.1 Q1(High precision model) |
| 002 | 2 | G05.1 Q2(High speed and high precision mode) |
| 003 | 0 | G05.1 Q0 ( default mode ) |
| 004 | 0 | G05.1 Q0 ( default mode ) |
| 005 | 0 | G05.1 Q0 ( default mode ) |
| 006 | 0 | G05.1 Q0 ( default mode ) |
| 007 | 0 | G05.1 Q0 ( default mode ) |
| 008 | 0 | G05.1 Q0 ( default mode ) |
| 009 | 0 | G05.1 Q0 ( default mode ) |
| 010 | 0 | G05.1 Q0 ( default mode ) |
| 011 | 0 | G05.1 Q0 ( default mode ) |
| 012 | 0 | G05.1 Q0 ( default mode ) |
| 013 | 0 | G05.1 Q0 ( default mode ) |
| 014 | 0 | G05.1 Q0 ( default mode ) |
| 015 | 0 | G05.1 Q0 ( default mode ) |

**Note:**

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

2: In the table, "Tool number" and "Machining mode" can be set. When tool number is 0 or tool type is "No tool", "Machining mode" cannot be set.

### 2.6.1.8. Lathe Milling Combo Configuration of Dual Channel

**1. Parameter configuration**

Parameter type of channel 0 and channel 1 is set as the type of lathe milling combo. Channel 0 is switched to milling mode and channel 1 is switched to lathe mode. Specific parameters are set as below.

| Parameter type | Parameter | | Parameter value | Description |
|---|---|---|---|---|
| NC parameter | 000065 | Lathe tool diameter display enable | 1 | X axis diameter display |
| Machine user parameter | 010001 | Cutting type of channel 0 | 2 | Lathe milling combo system |
| Machine user parameter | 010002 | Cutting type of channel 1 | 2 | Lathe milling combo system |
| Channel parameter (channel 0) | 040032 | Diameter programming enable | 1 | Enable X axis diameter programming mode |
| Channel parameter (channel 0) | 040094 | Machine type of power off | 1 | After the machine tool is powered on, the 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 | Enable X axis diameter programming mode |
| Channel parameter (channel 1) | 041094 | Machine type of power off | 2 | After the machine tool is powered on, the current working mode is milling |
| Channel parameter (channel 1) | 041101 | Number of tools on spindle | 1 | -- |

**2. Multi-magazine management**

(1) Channel 0 is in milling mode and all tools in the magazine are milling tools. Channel 1 is in lathe mode and all tools in magazine are lathe tools

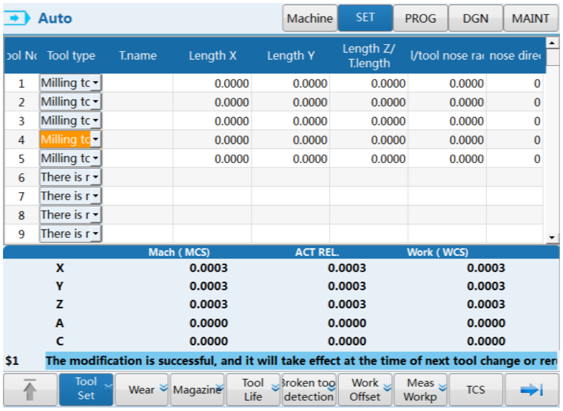
(2) Current tool number of spindle in channel 0 is acquired through register B188 and current tool number of spindle in channel 1 is acquired through register B204

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

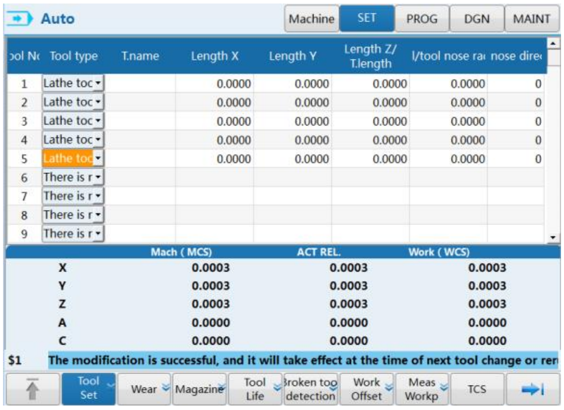
| Parameter type | Parameter | | Parameter value | Description |
|---|---|---|---|---|
| Channel parameter (Channel 0) | 040125 | Initial magazine number | 1 | -- |
| | 040126 | Number of magazines | 1 | -- |

| | | | | |
|---|---|---|---|---|
| | 040127 | Initial tool number | 1 | -- |
| | 040128 | Number of tools | 24 | The parameter can be modified based on actual situation |
| Channel parameter (Channel 1) | 041125 | Initial magazine number | 2 | -- |
| | 041126 | Number of magazines | 1 | -- |
| | 041127 | Initial tool number | 40 | The parameter should be set based on actual situation, but it must be greater than the last tool number of channel 0 |
| | 041128 | Number of tools | 24 | The parameter can be modified based on actual situation |

After parameters are set, the tool compensation interfaces of channel 0 and channel 1 are displayed as below.




Milling channel (channel 0)




Lathe channel (channel 1)

## 2.6.2. Vertical/Horizontal Lathe Conversion

### 2.6.2.1. Function Description

When machine tool is in lathe mode, there are 2 states, vertical lathe and horizontal lathe.

e.g.: For five-axis lathe milling combo with B swivel head and C rotary table, and spindle is installed on B swivel head, when B is 0°, it is in vertical lathe state; when B is 90°,it is in horizontal lathe state. Refer to the following diagram

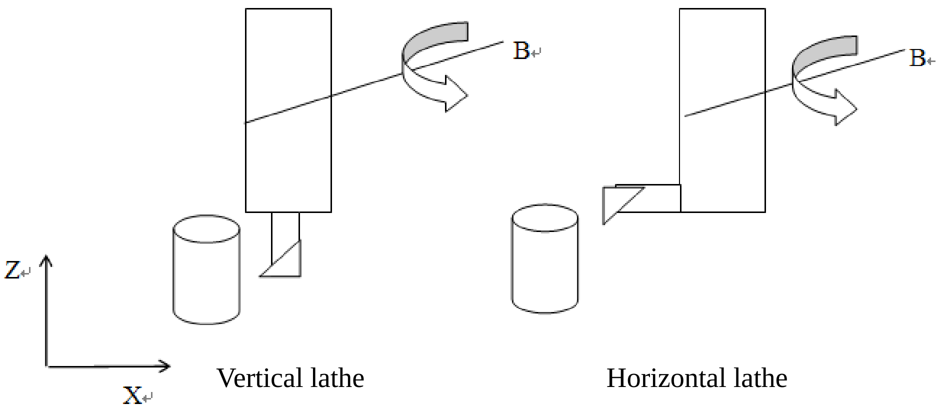



Figure 2-41 Vertical/horizontal lathe conversion

After vertical/horizontal lathe conversion, the following characteristics are satisfied:

1. After vertical/horizontal lathe conversion, programming is still performed in lathe mode with axial direction Z axis and radial direction X axis.

Exchange axes with GETD instruction. For Z axis programming, X axis is moved actually; for X axis programming, Z axis is moved actually.

2. After vertical/horizontal lathe conversion, the movement by handle and JOG are performed in the converted mode. e.g. if X axis is selected, the actual physical motion axis is Z axis

After vertical/horizontal lathe conversion, tool can be set normally according to lathe usage, and workpiece coordinate system and tool offset table can be built.

3. After vertical lathe is converted into horizontal lathe, ensure lathe tool nose points are superposed. The machine tool can complete machining of both vertical lathe mode and horizontal lathe mode only through performing tool setting once.

4. Interface display: Coordinate axes of the machine coordinate system can be exchanged, and the workpiece coordinate system remains unchanged;

If two columns are displayed and one of them is the machine coordinate system, the machine coordinate system plays leading role.

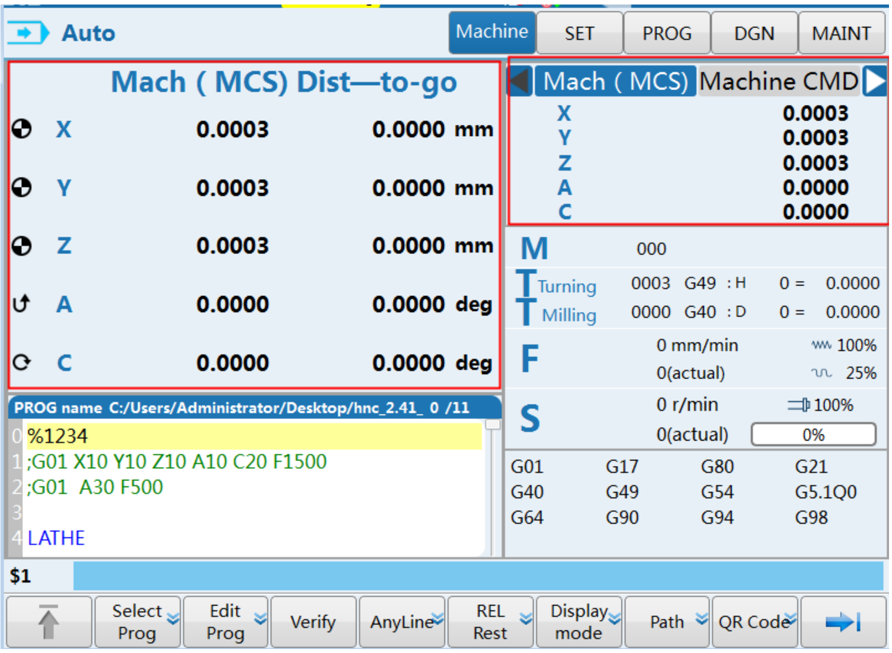



Figure 2-42 Coordinate display

### 2.6.2.2. Programming in Lathe Mode After Vertical/Horizontal Lathe Conversion

After vertical/horizontal lathe conversion, programming is still performed in lathe mode with axial direction Z axis and radial direction X axis. Exchange axes through GetD instruction. For Z axis programming, X axis is moved actually; for X axis programming, Z axis is moved actually.

Functional configuration

**(1) Parameter modification**

Channel parameter "040102 dynamic switching axis mask" is set according to axis to be switched, e.g.: To switch X and Z axes, fill in 0x1; to switch Y and Z axes, fill in 0x2.

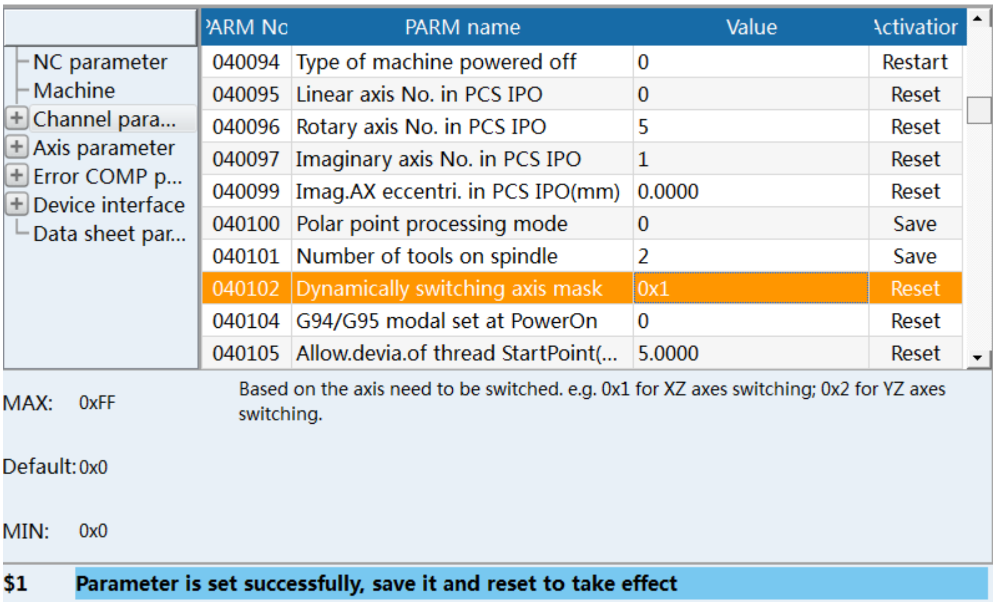



Figure 2-43 Parameter setup

**(2) Call canned cycle using M code and note that M code is not occupied**

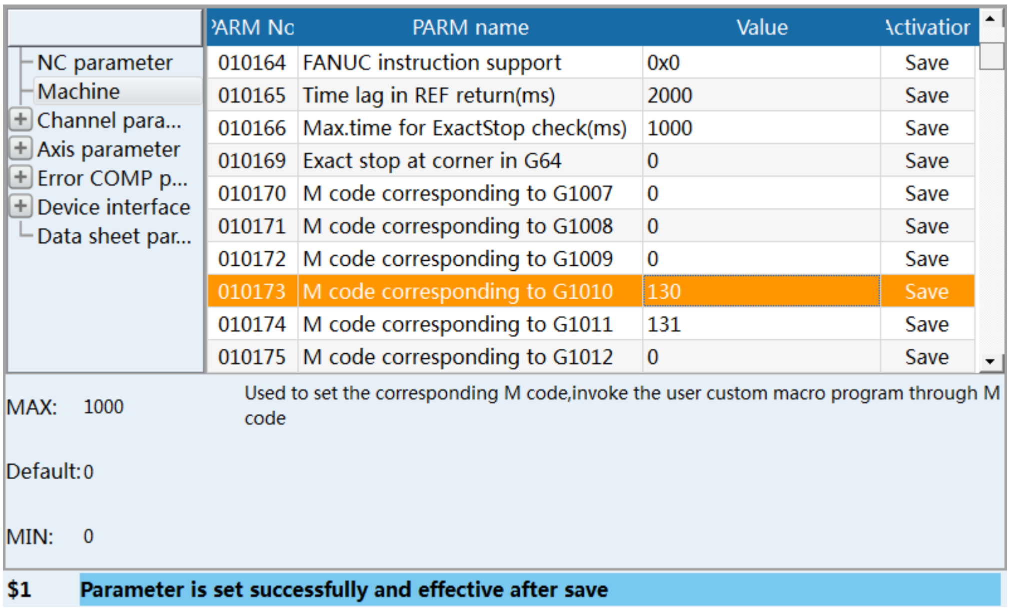



Figure 2-44 Parameter setting

**(3) Canned cycle file**

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

```
%1010
Vertical/horizontal lathe version (X axis and Z axis are exchanged)
FREE X-1 Z-1
GETD X2 Z0
M49 Cooperate with PLC, in JOG or handwheel mode, move in the converted mode
M99

%1011
Release vertical/horizontal lathe conversion
FERR X-1 Z-1
GETD X0 Z2
M48 ; Cooperate with PLC, in JOG or handwheel mode, move in the converted mode
M99
```

b) Description

FREE X-1 Z-1

The command means to release X axis and Z axis

GETD X2 Z0

The command means to map logical axis 2 to X axis and map logical axis 0 to Z axis

c) Example

%1234

G54

M130; vertical/horizontal lathe conversion, exchange X axis and Z axis

M131; release vertical/horizontal lathe conversion,

M30

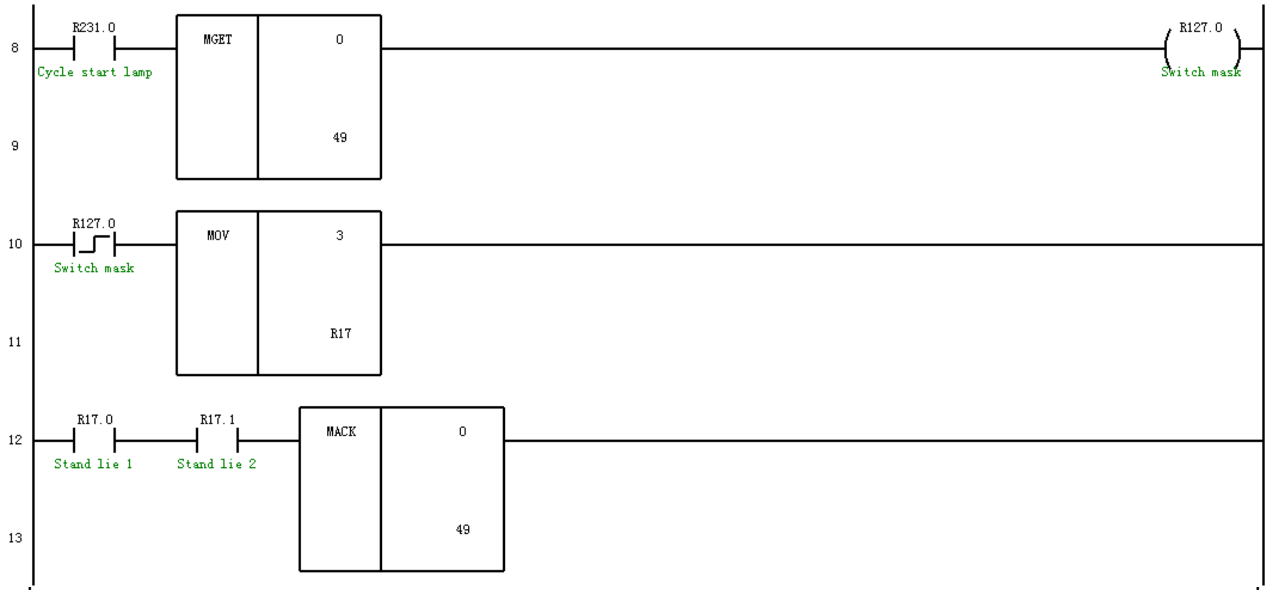
### 2.6.2.3. Movement by JOG and Handwheel is in Converted mode after Vertical/Horizontal Lathe Conversion

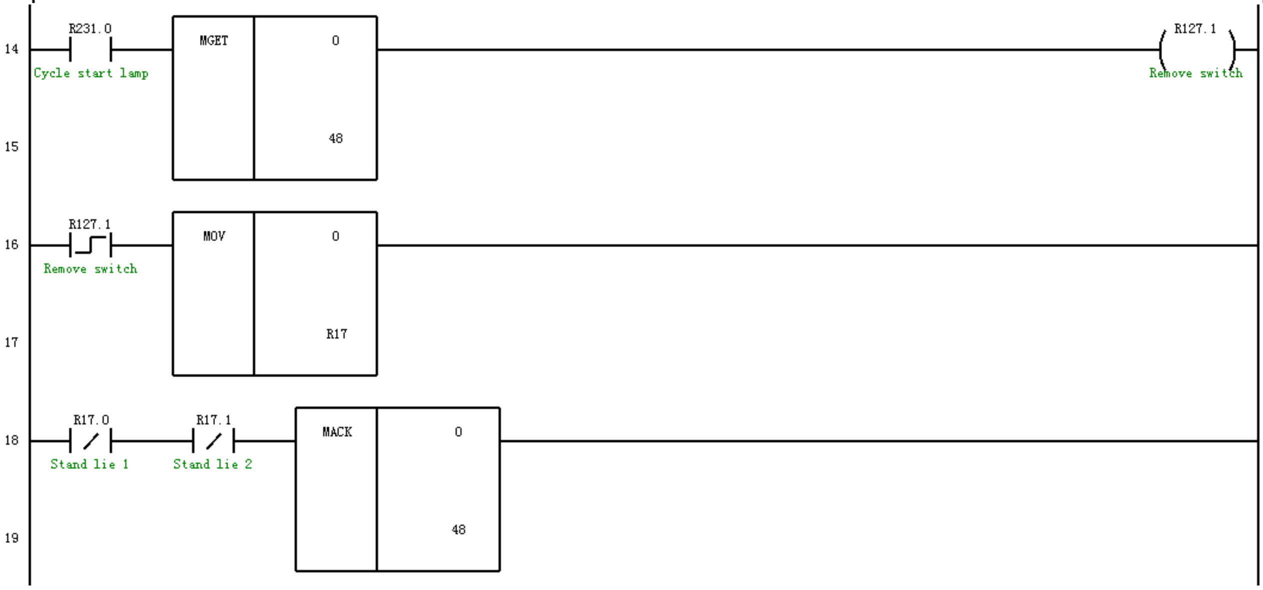
After vertical/horizontal lathe conversion, the movement by handwheel and jog is performed in the converted mode. e.g. Select X axis, and Z axis moves actually. After vertical/horizontal lathe conversion, the tool setting can be performed according to lathe using, and workpiece coordinate system and tool offset table are established.

Application example

**With 818D panel as an example**

(1) Add M48 (cancel vertical/horizontal lathe conversion) to PLC1 and M49 (vertical/horizontal lathe conversion)

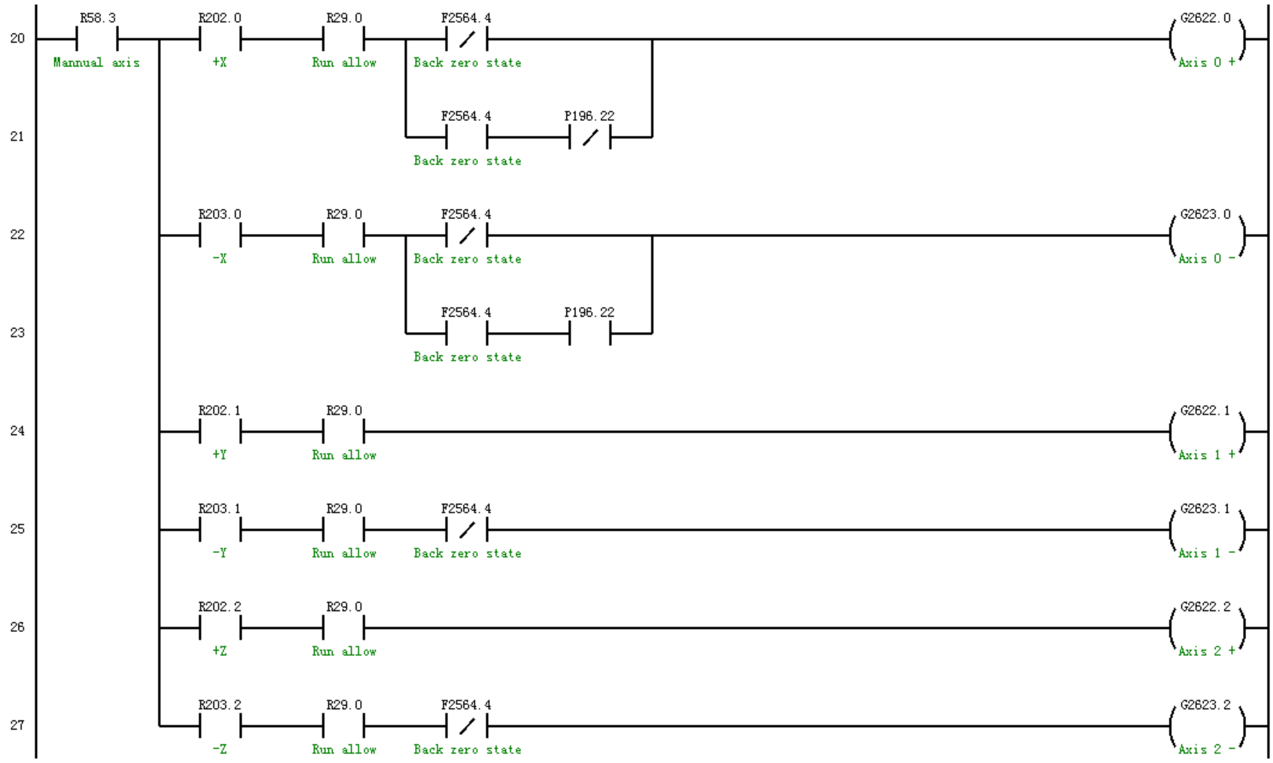


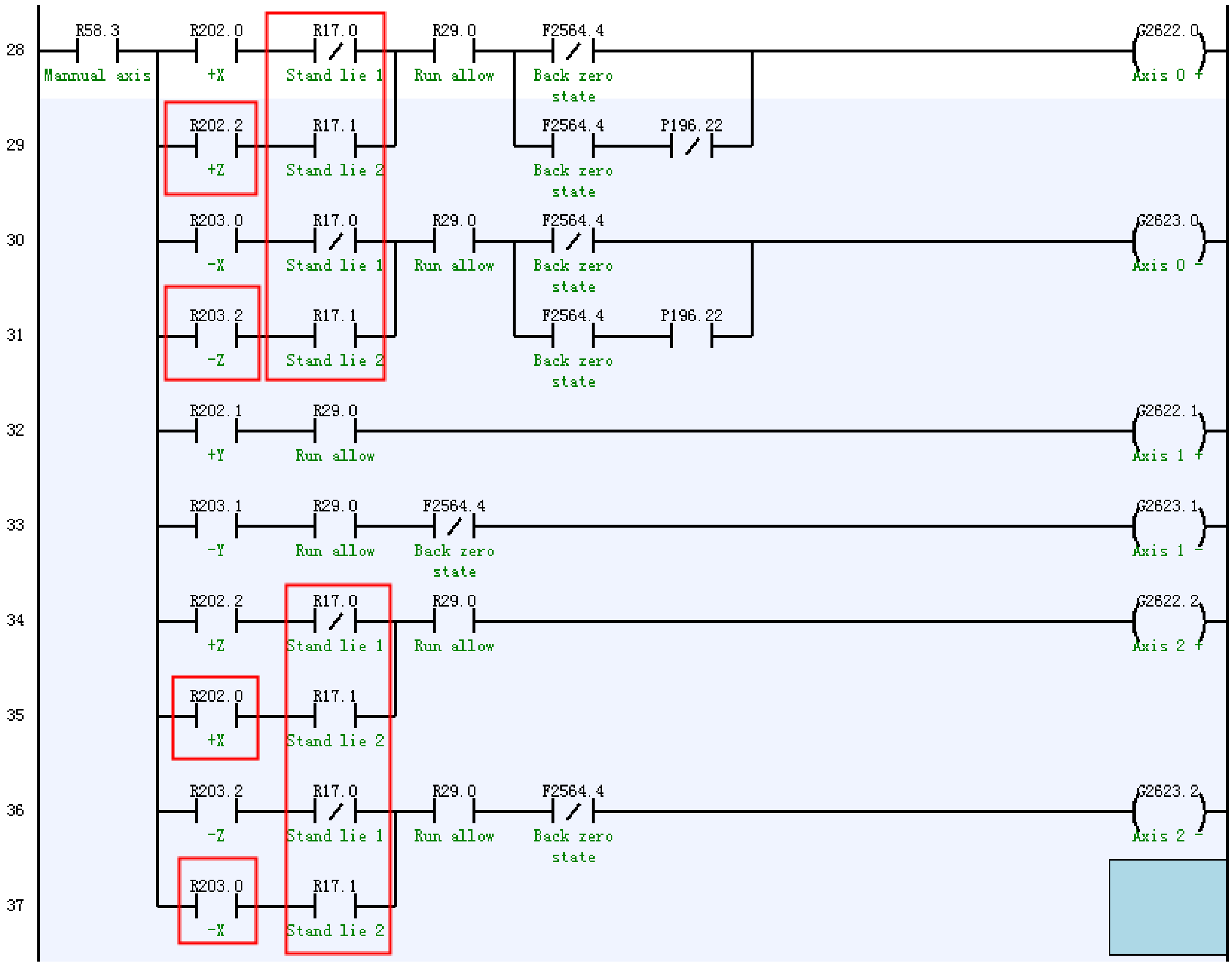

(2) Modification of the JOG part

Look for G2622.0 and modify switching conditions

PLC is shown below before modification



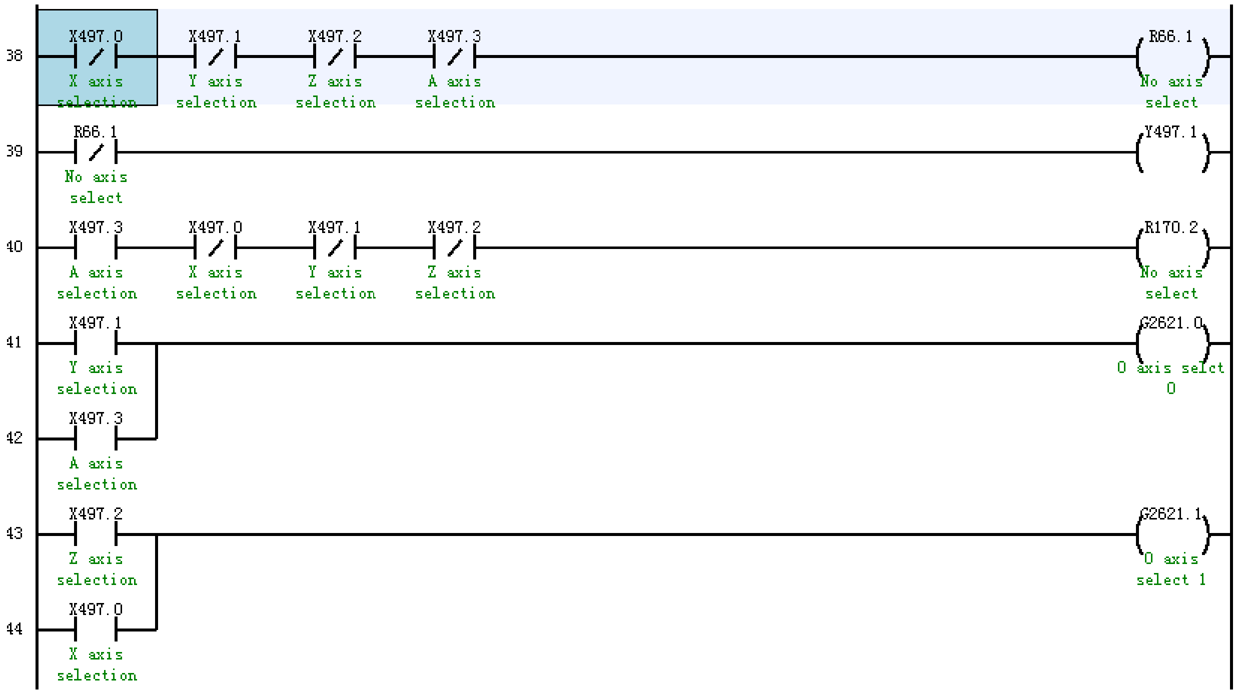

PLC is shown below after modification



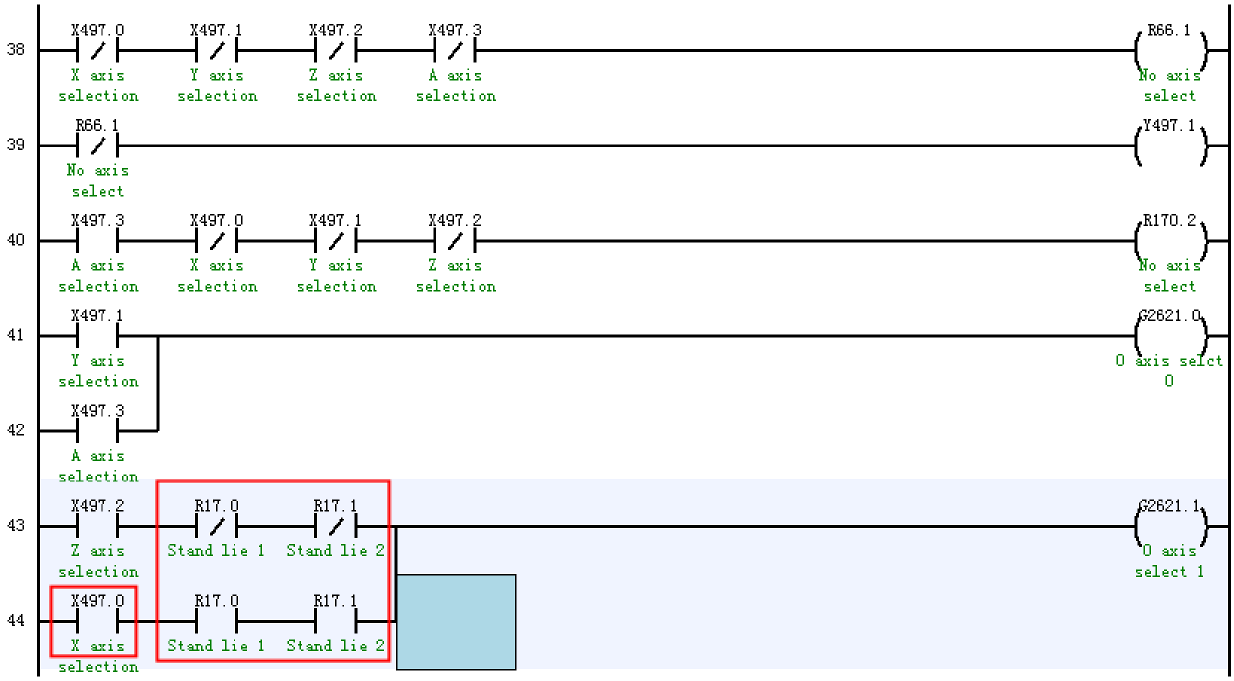

(3) Modification of the handwheel part

Look for G2621.0

PLC is shown below before modification




PLC is shown below after modification




### 2.6.2.4. Coincidence of Tool Nose After Vertical/Horizontal Lathe Conversion

After vertical lathe is converted into horizontal lathe, the lathe tool nose points are superposed. The machine tool can complete machining of both vertical lathe mode and horizontal lathe mode only through performing tool setting once.

**1. Function configuration**

(1) Parameter modification. Call canned cycle using M code and ensure M code is not occupied

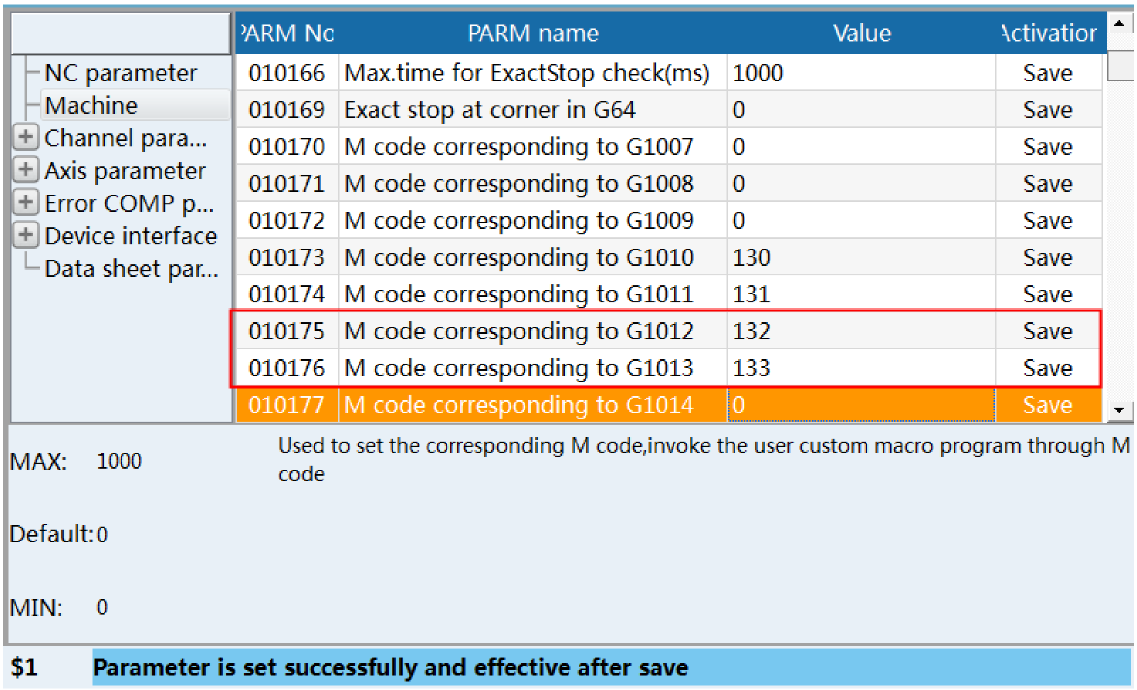



Figure 2-45 Parameter setting

(2) Canned cycle file USERDEF.CYC

Add G1012 and G1013 user canned cycles. If canned cycle number is occupied, users can choose others to use.

Content of G1012 canned cycle is shown below

```
 %1012
IF [AR[#7] EQ 0]    ; H number is not defined
    G110
ENDIF
IF #7 LT 0      ; H number 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
```

Content of G1013 canned cycle is shown below

```
%1013
G52 X0Y0Z0
G80
M99
```

**2. Description:**

G10 L43 P_ X_ Y_ Z_ A_ B_C_ R_

The command calculates machine coordinates under RTCP through the given workpiece coordinate position and writes it in system variable. Where, X, Y, Z, A, B, and C are the given workpiece coordinates, R is the written macro variable number, and P is the tool compensation number.

**3. Example**

%1234
G54
M132 H1; Under horizontal mode, current tool number is 1, call G1001 to complete vertical/horizontal lathe conversion and coincidence of tool nose
M133; Recover the coordinate system under vertical mode
M30

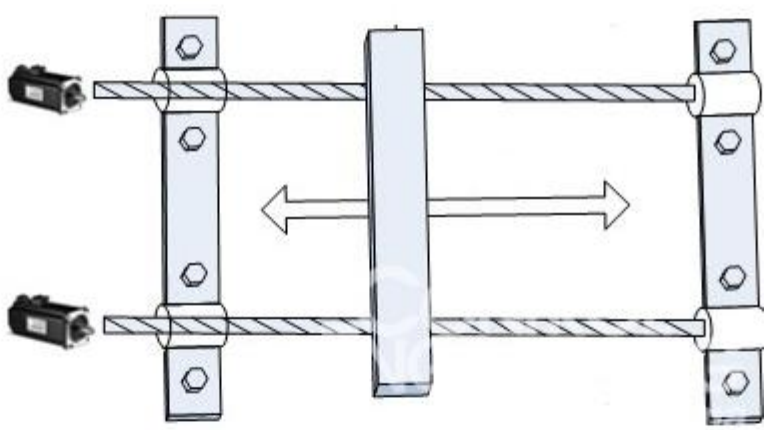
**4. Note**

(1) With this function, machine coordinate position of tool nose point under vertical mode and horizontal mode can be calculated through G10 L43 command in canned cycle, and the tool nose offset can be obtained. While executing canned cycle, RTCP simultaneous-movement is not performed. During displacement motion with subsequent commands, the system will compensate tool nose offset amount, and there is no need to perform tool setting again for users to perform turning.

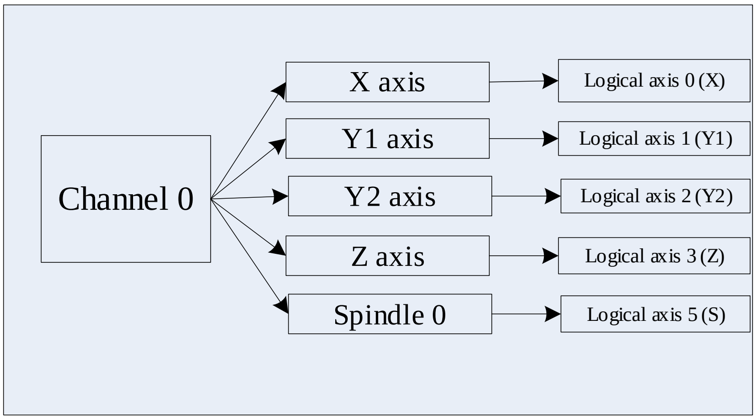
(2) Lathe tool compensation setting. Set channel parameter 【040409】 bidirectional tool length compensation of right angle head as 1, and fill lathe tool compensation value in the tool compensation table: Length X and length Z.

## 2.7. Gantry Synchronization Function

Gantry synchronization means that a mechanical axis is controlled by at least 2 servo motors, in which one is the master axis and the other is the slave axis. Generally this function can be used by machine tools with gantry milling mechanism.



Due to different feedback modes of synchronous axis (including incremental encoder, absolute encoder, distance-coded grating ruler, and absolute grating ruler), configuration of HNC-8 system is different. With the below figure as an example, configuration of synchronous axis is described here. Y axis is a synchronous axis consisting of Y1 and Y2 axes, and Y1 axis is the master axis and Y2 is the slave axis.




### 2.7.1. Commissioning of Synchronous Axis with Incremental Encoder

#### A. Parameter setting

1) Machine user parameter setting

Parameter 010050 total number of PMC and coupling slave stations is set as 1. There is only one slave axis Y2, so the parameter should be set as 1.

Parameter 010051 PMC and coupling axis number [0] is set to 2. Logical axis 2 in coordinate axis parameter is the slave axis, so the parameter is set as 2.

- NC parameter
- Machine
- Channel para...
  - Channel0
  - Channel1
  - Channel2
  - Channel3
- Axis parameter
- Error COMP p...
- Device interface
- Data sheet par...

| ʼARM Nc | PARM name | Value | ∖ctivatior |
|---|---|---|---|
| 010044 | Speed after radius compensation | 1 | Save |
| 010045 | RC=radius -/+ wear | 1 | Reset |
| 010046 | RC interference control | 0 | Reset |
| 010047 | Num.of blocks for RC interf.check | 2 | Reset |
| 010049 | Max.num.of axes on machine | 10 | Restart |
| 010050 | Total of PMC&coupl.slave axes | 0 | Restart |
| 010051 | PMC&coupl.slave axes No.[0] | -1 | Restart |
| 010052 | PMC&coupl.slave axes No.[1] | -1 | Restart |
| 010053 | PMC&coupl.slave axes No.[2] | -1 | Restart |
| 010054 | PMC&coupl.slave axes No.[3] | -1 | Restart |

2) Coordinate axis parameter setting

Logical axis 1 (master axis), parameter 101000 display axis name is set to Y1.

| ʼARM Nc | PARM name | Value | ∖ctivatior |
|---|---|---|---|
| 101000 | Axis name display | Y1 | Save |

Logical axis 2 (slave axis), parameter 102000 display axis name is set to Y2.

| ʼARM Nc | PARM name | Value | ∖ctivatior |
|---|---|---|---|
| 102000 | Axis name display | Y2 | Save |

Logical axis 2 (slave axis), axis type, gear ratio, axis movement speed, and axis acceleration/deceleration are set as per parameters for logical axis 1.

| ʼARM Nc | PARM name | Value | ∖ctivatior |
|---|---|---|---|
| 102000 | Axis name display | Y2 | Save |
| 102001 | Axis type | 1 | Save |
| 102004 | ELG ratio NUMERA[position](um) | 12000 | Restart |
| 102005 | ELG ratio DENOM[pulse] | 131072 | Restart |

Note: When the movement direction of master axis is opposite to that of slave axis, positive and negative signs of numerator of electronic gear ratio can be modified.

Logical axis 2 (slave axis), parameter 102100 axis motion control mode is set as 1.

| | | | |
|---|---|---|---|
| 102100 | Axis motion control mode | 1 | Reset |

Note: The setting of 1 indicates synchronous axis.

Logical axis 2 (slave axis), parameter 102101 master axis 1 number is set as 1.

| | | | |
|---|---|---|---|
| 102101 | Master axis 1 No. | 1 | Reset |
| 102102 | Master axis 2 No. | -1 | Reset |
| 102103 | Master axis 3 No. | -1 | Reset |
| 102104 | Master axis 4 No. | -1 | Reset |
| 102105 | Master axis 5 No. | -1 | Reset |

Master axis is Y1 axis and the corresponding logical axis is 1, so master axis 1 number is set as 1, and corresponds to spindle number.

Logical axis 2 (slave axis), parameter 102062 automatic adjustment of flexible synchronization is set to 0.

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 0 | Reset |

Automatic adjustment of flexible synchronization should be disabled before initialization, so the value is set as 0.

Logical axis 2 (slave axis), settings of relevant synchronization thresholds.

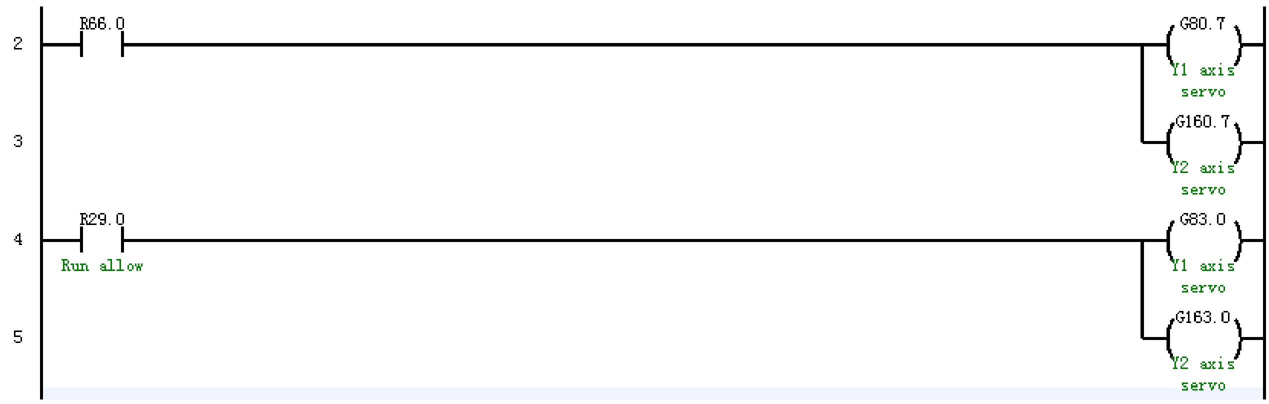
| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

Upon initialization, the above synchronization thresholds are set as 0, and detection is not enabled.

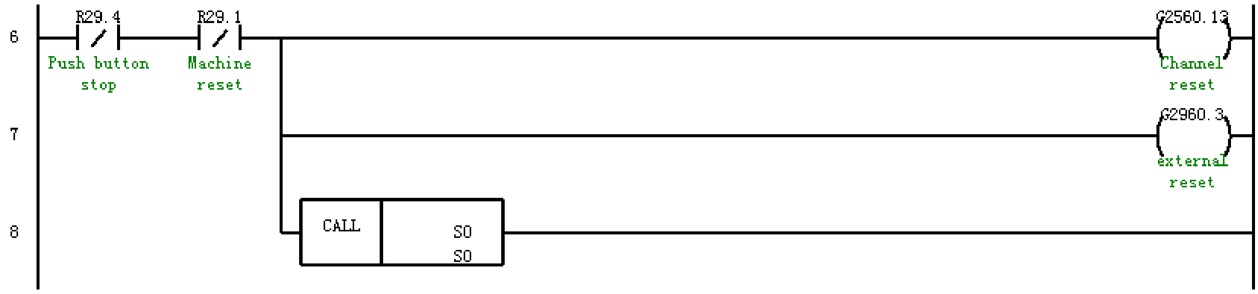
Save parameters, power off, and restart the system.

### B. PLC setting

Add slave axis enable signal



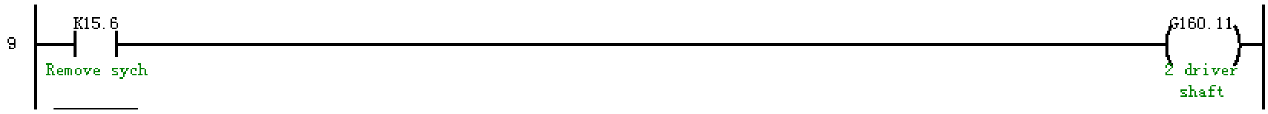

Add external reset flag G2960.3 upon reset.




Note:

When external reset flag G2960.3 is not added, and the prompt alarm of "excessive tracking error of slave axis" appears on the interface, this prompt message cannot be eliminated by reset, and can be eliminated only after the flag is added to PLC.

Add Release synchronization of slave axis to PLC.




When the position of synchronous axis needs to be calibrated, users can use the handwheel for adjustment of synchronous axis after releasing synchronization.

The system releases emergency stop and switches to handwheel mode. Meanwhile, users release synchronization of synchronous axis in PLC. Then adjust the position of synchronous axis and enable synchronization function in PLC.

Switch the channel to reference point return mode and then start reference point return.

After reference point return succeeds, enable automatic adjustment of synchronous axis (set Parm102062 "Automatic adjustment of flexible synchronization" as 1).

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 1 | Reset |

Set compensation threshold and alarm threshold of synchronous axis to complete configuration of synchronous axis.

| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

Alarm threshold of synchronous position error and alarm threshold of synchronous current error should be set based on actual situation.

Alarm threshold of synchronous position error and alarm threshold of synchronous speed error are system standby parameters and needs not to be set.

Note:

- After automatic adjustment of flexible synchronization is enabled, if synchronous position error value is less than synchronous position error alarm threshold, the position of slave axis motor will be adjusted automatically when the emergency stop is released to keep machine actual coordinates of master axis and slave axis consistent. If synchronous position error value is greater than synchronous position error alarm threshold, the system alarm "Synchronization out of tolerance", and the position of slave axis motor will not be adjusted automatically either.
- If automatic adjustment of flexible synchronization is disabled, the position of synchronous axis will not be adjusted automatically when synchronous position error occurs and emergency stops is released.

After settings are completed, restart the system to complete configuration of synchronous axis.

### 2.7.2. Configuration of Synchronous Axis with Absolute Encoder (Absolute Grating Ruler)

#### A. Parameter setting

Machine user parameter setting

Parameter 010050 total number of PMC and coupling slave stations is set as 1. There is only one slave axis Y2 axis, so the parameter should be set to 1.

Parameter 010051 PMC and coupling slave axis number [0] is set as 2. Logical axis 2 in coordinate axis parameters is the slave axis, so the parameter should be set to 2.

- NC parameter
- Machine
- Channel para...
- Axis parameter
- Error COMP p...
- Device interface
- Data sheet par...

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 000001 | Interpolation cycle (us) | 1000 | Restart |
| 000002 | Periodic num.of PLC2 statements | 200 | Restart |
| 000005 | Angle calculation resolution | 100000 | Restart |
| 000006 | Length calculation resolution | 100000 | Restart |
| 000010 | Allowable Cir.IPO contour error(... | 0.0050 | Restart |
| 000011 | Allowable Cir.radius deviation(mm) | 0.1000 | Restart |
| 000012 | Tool axis selection mode | 0 | Reset |
| 000013 | G00 interpolation Enable | 1 | Save |
| 000014 | Restoring TLC after G53/G28 | 1 | Save |
| 000018 | System time display | 1 | Save |

Logical axis 1 (master axis), parameter 101000 display axis name is set to Y1.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 101000 | Axis name display | Y1 | Save |

Logical axis 2 (slave axis), parameter 102000 display axis name is set to Y2.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 102000 | Axis name display | Y2 | Save |

Logical axis 2 (slave axis), axis type, gear ratio, axis movement speed, and axis acceleration/deceleration are set as per parameters of logical axis 1.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 102000 | Axis name display | Y2 | Save |
| 102001 | Axis type | 1 | Save |
| 102004 | ELG ratio NUMERA[position](um) | 12000 | Restart |
| 102005 | ELG ratio DENOM[pulse] | 131072 | Restart |

Note: When the movement direction of master axis is opposite to that of slave axis, users can change positive and negative signs of electronic gear ratio.

Logical axis 2 (slave axis), parameter number 102100 motion control mode of axis is set to 1.

| | | | |
|---|---|---|---|
| 102100 | Axis motion control mode | 1 | Reset |

The setting of 1 represents synchronous axis.

Logical axis 2 (slave axis), parameter number 102101 master axis 1 number is set to 1.

| | | | |
|---|---|---|---|
| 102101 | Master axis 1 No. | 1 | Reset |
| 102102 | Master axis 2 No. | -1 | Reset |
| 102103 | Master axis 3 No. | -1 | Reset |
| 102104 | Master axis 4 No. | -1 | Reset |
| 102105 | Master axis 5 No. | -1 | Reset |

Master axis is Y1 axis and the corresponding logical axis is 1, so master axis 1 number is set as 1, and corresponds to master axis number.

Logical axis 2 (slave axis), parameter number 102062 automatic adjustment of flexible synchronization is set to 0.

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 1 | Reset |

Automatic adjustment of flexible synchronization should be disabled before initialization, so the value is set as 0.

Logical axis 2 (slave axis), setting f relevant synchronization thresholds.

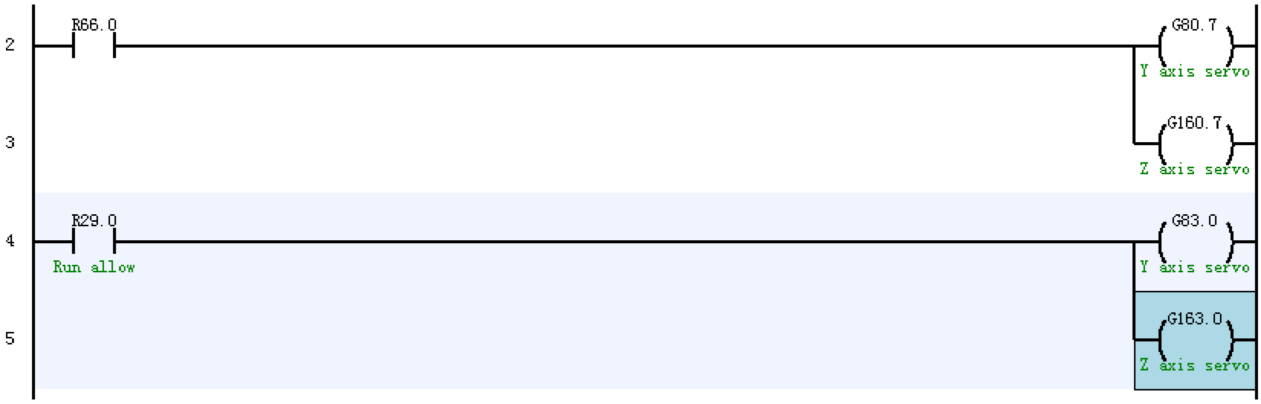
| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

Upon initialization, the above synchronization thresholds are set as 0 and detection is not enabled.

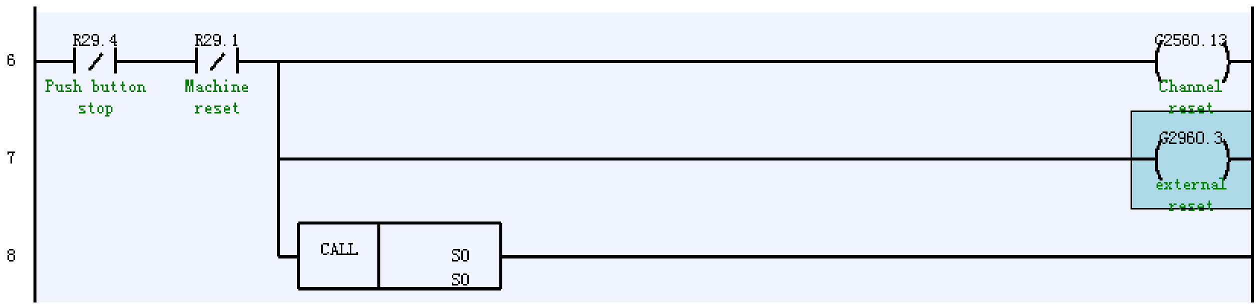
Save parameters, power off, and restart the system.

## B. PLC setting

Add slave axis enable signal

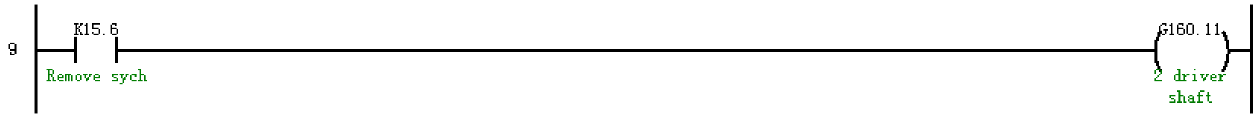



Add external reset flag G2960.3 upon reset.




Note: When external reset flag G2960.3 is not added, and the alarm prompt “Excessive tracking error of slave axis" appears on the interface, this message cannot be eliminated by reset but by adding this flag into PLC.

Add synchronization release of slave axis to PLC.




When the position of synchronous axis needs to be calibrated, users can use handwheel for adjustment of synchronous axis after synchronization release.
The system releases emergency stop and switches to handwheel mode. Meanwhile, PLC releases synchronization of synchronous axis. Adjust the position of synchronous axis and then set the coordinate zero point.
Setting of coordinate zero point
Set the zero point of master axis and slave axis by "Auto offset" key.
Click on "Auto offset" and the dialog box "Please Enter axis number:" will appear. Type 1 and press Enter. The system dialog box displays "Encoder feedback offset of axis 1 is set to XX". Continue to click on "Auto offset" and the dialog box "Please enter axis number:" will appear. Type 2 and press Enter. The system dialog box displays "Encoder feedback offset of axis 2 is set to XX". Click on Save. After save succeeds, press and release emergency stop once.
Move the axis to the position to be set as the zero point and reset the coordinate zero point.
Setup steps for the coordinate zero point are consistent with those in Section 3.5.

### C. Enable automatic adjustment of synchronous axis

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 1 | Reset |

Set compensation threshold and alarm threshold of synchronous axis to complete configuration of synchronous axis.

| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

Alarm threshold of synchronous position error and alarm threshold of synchronous current error should be set based on actual situation.
Alarm threshold of synchronous position error and alarm threshold of synchronous speed error are system standby parameter and needs not to be set.
Note:
After automatic adjustment of flexible synchronization is enabled, if synchronous position error value is smaller than synchronous position error alarm threshold, the position of slave axis motor will be adjusted automatically when emergency stop is released in order that actual machine coordinates of master axis and slave axis are consistent. If synchronous position error value is greater than synchronous position error alarm threshold, the system will give an alarm "Synchronization out of tolerance" and the position of slave axis motor will not be adjusted automatically either.
When automatic adjustment of flexible synchronization is disabled, the position of synchronous axis will not be adjusted automatically when synchronous position error occurs and the system releases emergency stop.
After setup is completed, restart the system to complete configuration of synchronous axis.

## 2.7.3. Configuration of Synchronous Axis with Distance-coded Grating Ruler

### A. Machine user parameter setup

Parameter 010050 total number of PMC and coupling slave stations is set to 1. There is only one slave axis Y2 axis, so the parameter should be set as 1.

Parameter 010051 PMC and coupling slave axis number [0] is set as 2. Logical axis 2 in coordinate axis parameters is the slave axis, so the parameter should be set as 2.

- NC parameter
- Machine
- Channel para...
- Axis parameter
- Error COMP p...
- Device interface
- Data sheet par...

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 010046 | RC interference control | 0 | Reset |
| 010047 | Num.of blocks for RC interf.check | 2 | Reset |
| 010049 | Max.num.of axes on machine | 10 | Restart |
| 010050 | Total of PMC&coupl.slave axes | 1 | Restart |
| 010051 | PMC&coupl.slave axes No.[0] | 2 | Restart |
| 010052 | PMC&coupl.slave axes No.[1] | -1 | Restart |
| 010053 | PMC&coupl.slave axes No.[2] | -1 | Restart |
| 010054 | PMC&coupl.slave axes No.[3] | -1 | Restart |
| 010055 | PMC&coupl.slave axes No.[4] | -1 | Restart |
| 010056 | PMC&coupl.slave axes No.[5] | -1 | Restart |

Coordinate axis parameter setup

Logical axis 1 (master axis), parameter number 101000 display axis name is set as Y1.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 101000 | Axis name display | Y1 | Save |

Logical axis 2 (slave axis), parameter 102000 display axis name is set as Y2.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 102000 | Axis name display | Y2 | Save |

Logical axis 2 (slave axis), axis type, gear ratio, axis movement speed, and axis acceleration/deceleration are set as per parameters of logical axis 1.

| PARM No | PARM name | Value | Activation |
|---|---|---|---|
| 102000 | Axis name display | Y2 | Save |
| 102001 | Axis type | 1 | Save |
| 102004 | ELG ratio NUMERA[position](um) | 12000 | Restart |
| 102005 | ELG ratio DENOM[pulse] | 131072 | Restart |

Note: When the movement direction of master axis is opposite to that of slave axis, users can change positive and negative signs of electronic gear ratio.

Logical axis 2 (slave axis), parameter 102100 motion control mode of axis is set as 1.

| | | | |
|---|---|---|---|
| 102100 | Axis motion control mode | 1 | Reset |

The setting of 1 represents synchronous axis.

Logical axis 2 (slave axis), parameter 102101 master axis 1 number is set as 1.

| | | | |
|---|---|---|---|
| 102100 | Axis motion control mode | 1 | Reset |
| 102101 | Master axis 1 No. | -1 | Reset |
| 102102 | Master axis 2 No. | -1 | Reset |
| 102103 | Master axis 3 No. | -1 | Reset |
| 102104 | Master axis 4 No. | -1 | Reset |
| 102105 | Master axis 5 No. | -1 | Reset |

Master axis is Y1 axis and corresponding logical axis is 1, so guide axis 1 number is set as 1, corresponding to master axis number.

Logical axis 2 (slave axis), parameter number 102062, automatic adjustment enable of flexible synchronization is set as 0.

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 0 | Reset |

Automatic adjustment of flexible synchronization should be disabled before initialization, so the value is set as 0.

Logical axis 2 (slave axis) setting of relevant synchronization thresholds.

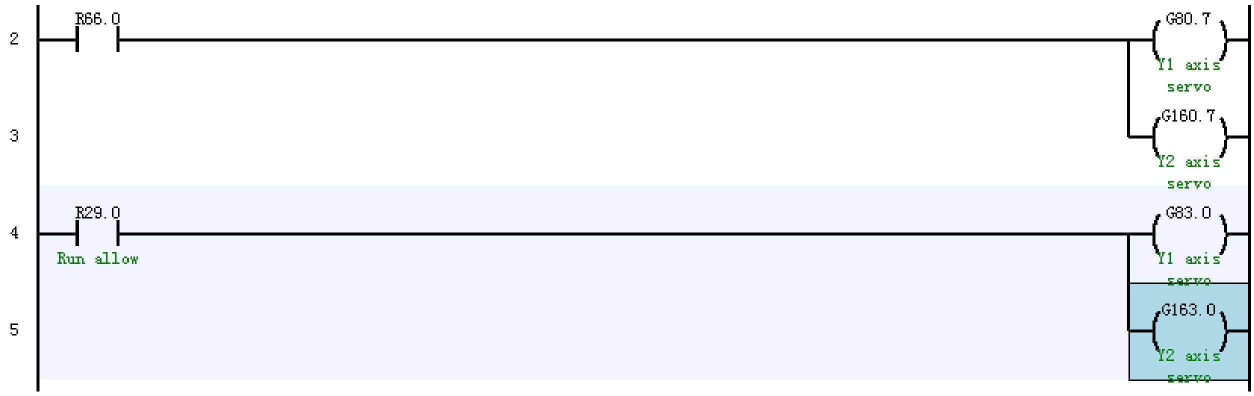
| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

Upon initialization, the above synchronization thresholds are set as 0 and detection is not enabled.

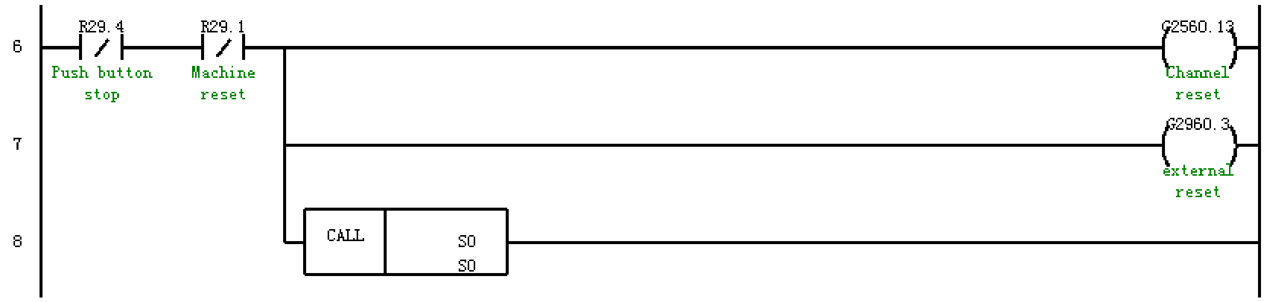
Save parameters, power off, and restart the system.

## B. PLC setting

Add slave axis enable signal

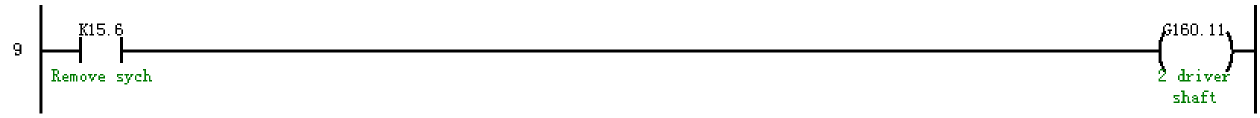



Add external reset flag G2960.3 upon reset.




Note: When external reset flag G2960.3 is not added, and the prompt alarm "Excessive tracking error of slave axis" appears on the interface, this message cannot be eliminated by reset but by adding this sign into PLC.

Add synchronization release of slave axis to PLC.




When the position of synchronous axis need to be calibrated, users can use handwheel for adjustment of synchronous axis after synchronization release.

The system releases emergency stop and switches to handwheel mode. Meanwhile, PLC releases synchronization of synchronous axis. Adjust the position of synchronous axis and then start reference point

return.
Setting of coordinate zero point
After reference point return succeeds, move the synchronous axis to the position to be set as the zero point. Set the actual machine position into coordinate value of reference point.

| | Work ( WCS) | Mach ( MCS) |
|---|---|---|
| Y1 | 22.7499 | 22.7499 mm |
| Y2 | 17.1499 | 17.1499 mm |

| | | | |
|---|---|---|---|
| 102012 | Encoder feedback offset(mm) | 17.7500 | Restart |

Enable automatic adjustment of synchronous axis.

| | | | |
|---|---|---|---|
| 102062 | Flexible SYNC auto-adujusting E... | 0 | Reset |

✧ Set compensation threshold and alarm threshold of synchronous axis to complete configuration of synchronous axis.

| | | | |
|---|---|---|---|
| 102106 | SYNC POS ERR COMP THR(mm) | 0.0000 | Reset |
| 102107 | SYNC POS ERR warning THR(mm) | 0.0000 | Reset |
| 102108 | SYNC S ERR warning THR(mm/m... | 0.0000 | Reset |
| 102109 | SYNC current error warn THR(A) | 0.0000 | Reset |

Alarm threshold of synchronous position error and alarm threshold of synchronous current error should be set based on actual situation.
Threshold of synchronous position error compensation and threshold of synchronous speed error alarm are system standby parameters and need not to be set.
Note:

- After automatic adjustment of flexible synchronization is enabled, if synchronous position error value is less than synchronous position error alarm threshold, the position of slave axis motor will be adjusted automatically when emergency stop is released in order that actual machine coordinates of master axis and slave axis are consistent. If synchronous position error value is greater than synchronous position error alarm threshold, the system will give an alarm "Synchronization out of tolerance", and the position of slave axis motor will not be adjusted automatically either.
- If automatic adjustment of flexible synchronization is disabled, the position of synchronous axis will not be adjusted automatically when synchronous position error occurs and the system releases emergency stop.

After setup is completed, restart the system to complete configuration of synchronous axis.

### 2.7.4. Other Notes

**Applicable version of servo**

DSP: HSV180UD_V283_19.11.6
FPGA: FPGA_4CE22_V4662

**Improvement of main function**

- Add synchronization error compensation function of gantry synchronous axis to reduce synchronization error incurred during dynamic operation of gantry synchronous axis (especially in reverse direction);
- Increase troubleshooting of gantry synchronous axis alarm to realize two axes stops synchronously when

the alarm of one axis occurs, and prevent mechanical twist arising from big error of mechanical position of gantry synchronous axis.

## Relevant parameters of servo

PB64 Synchronization function control word 1

| | | |
|---|---|---|
| Units: Enable axis synchronization | 0: Disable synchronizing function | 1: Allow synchronization function |
| Tens: Selection of axis type | 0: Master axis | 1: Slave axis |
| Hundreds: Master-slave synchronization mode | 0: Speed following | 1: Torque following |
| Thousands: Torque deviation detection switch | 0: Enable torque deviation detection | 1: Disable torque deviation detection |
| Ten thousands: Inverse slave axis command | 0: Slave axis command is not inverted | 1: Slave axis command is inverted |

PB65 Synchronization function control word 2

| | | |
|---|---|---|
| Units: Gantry axis synchronization compensation | 0: Disable synchronization compensation | 1: Enable synchronization compensation |
| Tens: Synchronous alarm of gantry axis | 0: Disable synchronous alarm function | 1: Enable synchronous alarm function |

PB66 Synchronization error compensation coefficient of gantry axis/current compensation coefficient of master and slave axes

## Parameter setup

## 1. Synchronization error compensation function

When synchronization error compensation function is enabled, the data exchange between mater and slave axes of gantry axis is implemented through XS1 interface, and synchronization error between synchronous axes is compensated on the slave axis side.

(1) Parameter setup of master axis:

Set the units of PB64 as 1, and hundreds as 0,

Set the units of PB65 as 1;

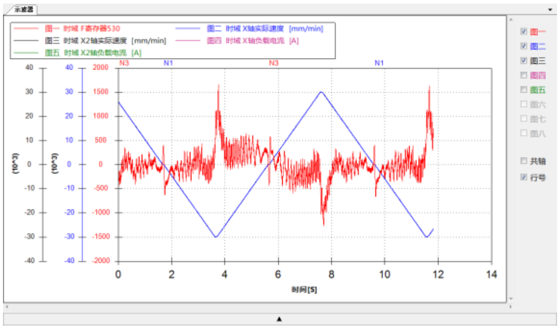
(2) Parameter setup of slave axis:

Set the units and hundreds of PB64 as 1, and set the ten thousands according to moving direction of slave axis and master axis motors;

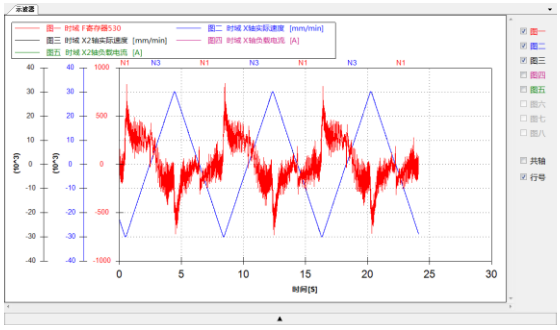
Set the units of PB65 as 1;

PB66 is used to adjust synchronization error compensation coefficient of gantry axis. Collect synchronization error of gantry axis with SSTT to directly display the synchronization error.

Waveform collected on a gantry machine tool is shown below. When PB66 is 0, maximum value of synchronization error is 15um. When PB66 is 5000, maximum value of synchronization error decreases to 8um. However, it should be noted that noise in the system will be constantly amplified by compensation parameter with the synchronization error magnitude reducing. PB66 cannot completely eliminate but reduce synchronization error.



(a) PB66 is 0



(b) PB66 is 5000

## 2. Synchronous processing of gantry axis alarm

(1) Alarm classification of servo

Alarm of servo can be classified into two levels:

- Emergency alarm: In case of fault, the servo cannot control the motor, including power module fault, overcurrent, and overvoltage of DC bus.
- General alarm: Data processed by the servo is abnormal, including excessive tracking error, overspeed, overload, locked rotor, and bus communication error,etc.

(2) Motor stop mode when the servo issues an alarm

- Coasting: When the servo gives an alarm, directly disable it, and the motor coasts;
- Emergency brake: Brake the motor, disable it, and it alarms.
- Dynamic brake: Not supported.

Stop mode of servo while setting alarm with the parameter PB45:

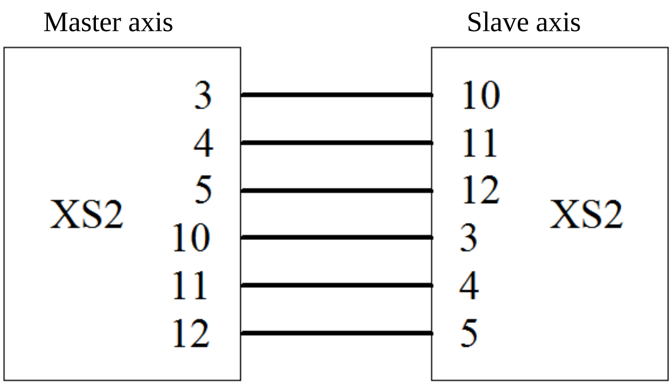
PB45 = 0, coasting;

PB45 = 10, emergency brake;

## 3. Usage method

When this function is enabled, the alarm message is transmitted through I/O point of XS2 interface between master axis and slave axis. When an axis gives an alarm, the alarm and alarm level are notified to the other axis through I/O so that the other axis gives an alarm quickly. Two synchronous axes of gantry axis process the alarm in the same way in order to ensure synchronization of gantry axis.

Wiring is defined as below:




Parameter PB64 needs not to be set. The tens of parameter PB65 is to set the synchronous alarm of gantry axis (the setting of 1 is to enable the synchronous alarm). Select the motor stop mode with parameter PB45 when the servo gives an alarm.

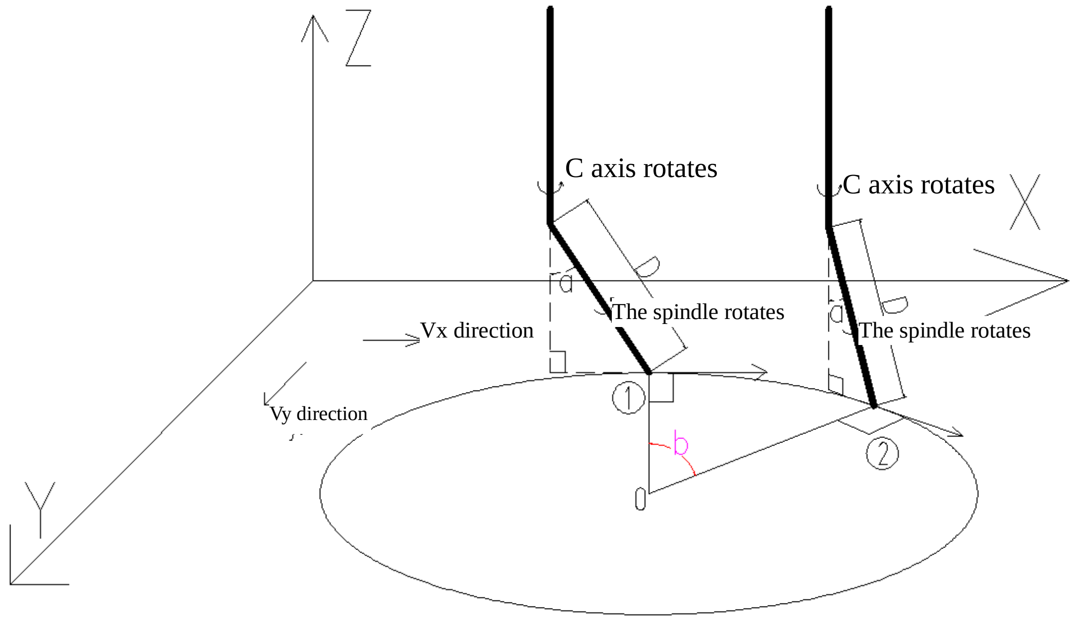
**Note: Parameters PB65 and PB45 of two synchronous axes of gantry axis should be consistent**

- When a synchronous axis gives an emergency alarm, the servo internally closes PWM signal, turn the enable signal off, and the motor coasts. After receiving the alarm message through XS2 interface, the other synchronous axis gives a 483 alarm, turns PWM signal and enable signal off, and the motor coasts.

- When a synchronous axis gives a general alarm, and the other synchronous axis receives the alarm message through XS2 interface, gives a 483 alarm, and the two axes stops in the way set by PB45.

## 2.8. Tangential Following Function

The follower axis moves on the tangent determined by leader axis. Thus, a tool can be adjusted to be parallel with the contour. Tool can be positioned relative to tangent according to the programmed angle in TANGON command.




### 2.8.1. Function Command

**Enable tangential following:**

tangon P_

P: 0 is A axis, 1 is B axis, 2 is C axis

**Cancel tangential following:**

tangoff P_

P: 0 is A axis, 1 is B axis, 2 is C axis

### 2.8.2. Parameter Description

**Leader axis wait 【134】:** When it is set as 1, rotary axis is positioned in the target position at the transition point of block, and then the leader axis moves. It is recommended to be set as 1.

**Follower axis number 【135】:** The setting of 2 indicates that C axis follows XY tangent

**Tangential following offset angle 【136】:** Angle when C axis points to the positive direction of X axis. The tangent angle of return is the positive direction of X axis. The parameter is to set the variation between C axis coordinate and positive direction of X axis.

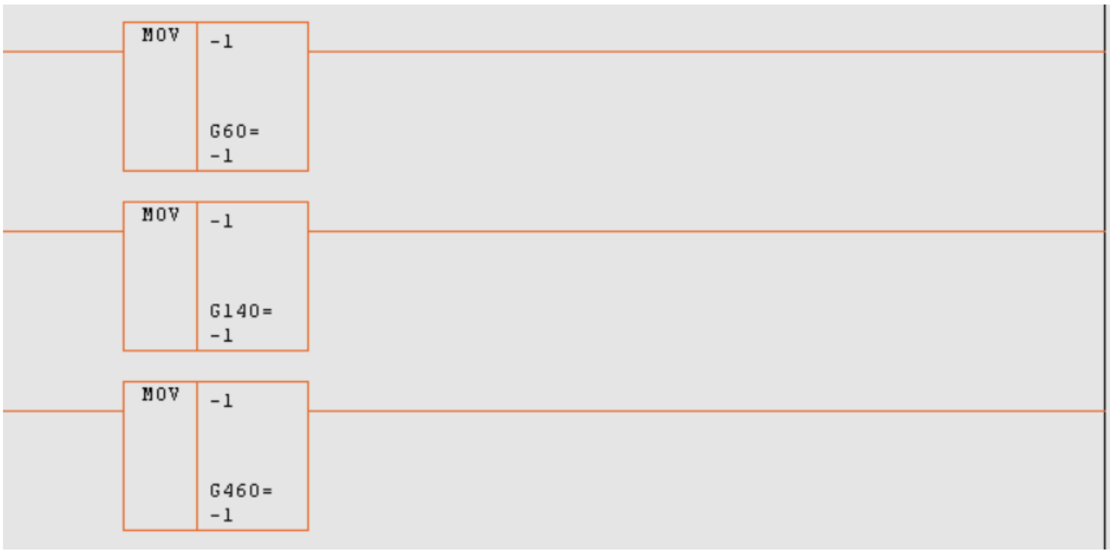
**Tangent following offset value 【137】:** Distance from the center of C axis to friction head, which is used to calculate offset of X/Y axis. Generally, when there is no angle of spindle and C axis, this value is set as 0.

**Leader axis number 1 【101】:** Logical axis number of leader axis

**Leader axis number 2 【102】:** Logical axis number of leader axis

### 2.8.3. PLC Control

Add the statement of axis control mode clearing in the reset S3 module. The control mode of both follower axis and leader axis needs to be cleared. Set G【X*80+60】 as -1. As shown below:




## Example

%1234

G01 G90 X-56 Y0 F1000

C89.9734

tangon p2; activate tangential following

N50 G01 X-61.276 Y1.711

N60G02X-65.031Y7.702I1.118J4.873

N80G02X-60.392Y21.058I63.452J-14.556

N90G02X-53.731Y23.431I4.517J-2.144

N100G01X-51.365Y22.308

tangoff p2; cancel tangential following

M30

### 2.8.4. Note

（1）When the specified tangential follower axis is not the rotary axis, the alarm syntax error will be issued.

（2）When the specified leader axes 1 and 2 are not configured, the alarm syntax error will be issued.

（3）The system will verify programming speed based on maximum machining speed of rotary axis to prevent overspeed.

（4）When the parameter leader axis wait is set to 1, rotary axis needs not to be positioned in the tangential position of the first line of contour before cutting following is activated. The system will adjust the position of rotary axis automatically and move along the contour track. When it is set as 0, rotary axis must be positioned in the tangential position of the first line of contour before cutting following is activated.

**Acquiring of contour tangential angle:**

G115 L5 P1 Q10

L: Read ahead a line of commands. If the command is G01/G02/G03 and there is X/Y movement, calculate tangential angle and save it in #1319 variable

P: The program to be read ahead belongs to, 0: Current layer; 1: Previous layer; 2. The layer before previous layer

Q: Lines down from the current line

**Position rotary axis to tangential angle:**

G01 C[#1319]

# 3. Machining And Commissioning

## 3.1. Five-axis NAS Workpiece

### 3.1.1. Basic Information of Test NAS Truncated Cone

The text NAS truncated cone here is made according to the following 3 standards. The original NAS test workpiece is modified on workpiece size, contour slope, mounting angle on worktable of machine tool, test item, and test standard in order to test machining accuracy and performance of machine tool in more aspects.

1) NAS979 test piece produced by National Aeronautics and Space Administration (NASA);

2) Test piece of cone frustum in national standard JB/T 10792.1-2007 *The 5-axes simultaneous vertical machining centers-Part 1: Testing of the accuracy*;

3) NAS979 test piece and the test standard of Germany Zimmermann Company.

### 3.1.2. 1.2 Model Analysis

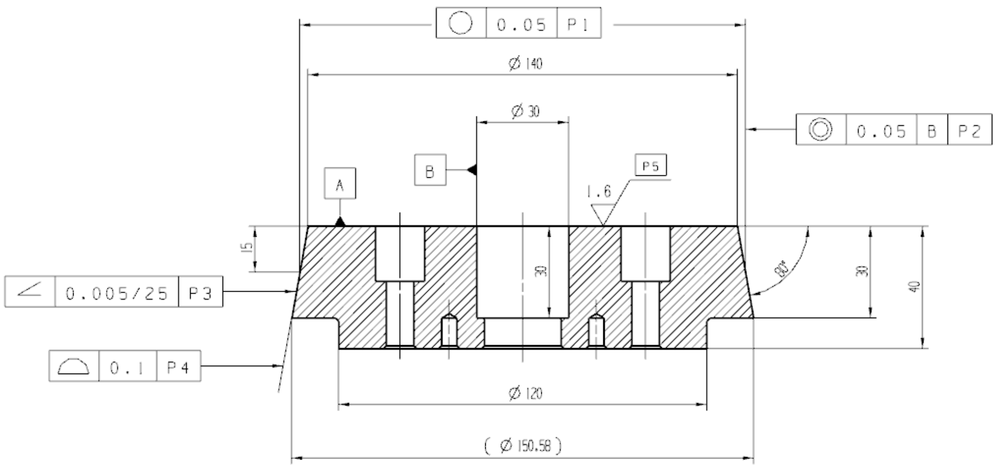



Figure 3-1 Model size, test item and test standard (Size and standard of model can be adjusted based on size and type of machine tool)

As shown in Fig. 3-1, the test piece is a cone frustum. To display simultaneous five-axis machining and 3+2 orientation machining, the test piece is installed on a base, and the upper surface of the base inclines at an angle of 10° (as shown in Fig. 3-2). The lower surface of the base fits the worktable surface and with screws or pressing plates.

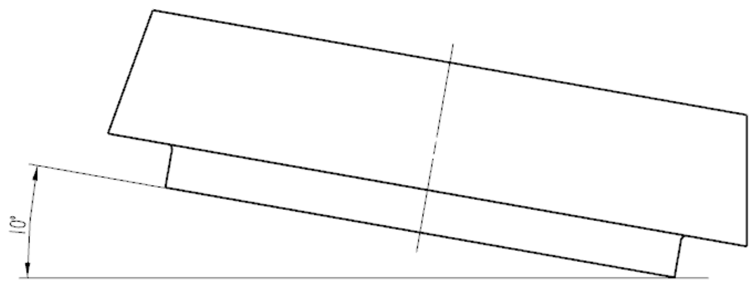



Figure 3-2 Workpiece installation

Parts to be machined: Mill the upper surface of cone frustum with face mill, machine the cylindrical bore with boring tool, and mill the side surface of cone frustum with end mill. After machining, the cone frustum can be removed from the base for measurement.

Dimension adjustment: The model uses five-axis vertical machining center as the test object. When test object is a large five-axis machine tool (such as gantry five-axis machining center), dimensions of model can be adjusted accordingly (suggestion: adjust basal diameter of cone frustum to 254mm, axial thickness to 63.5mm, diameter of center cylindrical bore to φ50.8mm, and side slope of frustum to 15°, installed at 7.5°), and relevant cutters should be adjusted or replaced accordingly.

Test piece should be in the middle of the X-axis travel as much as possible and should be installed along Y axis and Z axis where is suitable for positioning of test piece and fixture as well as the tool length. Test piece of different structural styles can be installed as shown in Fig. 2. When necessary, base and machine tool can be connected with transition. There are 4 screw holes on test piece and test piece is fixed to base with 4 screws.

Other requirements for location and fixing mode of test piece should be pointed out in manufacturers' or users' agreement.

Test piece can be reused in cutting test and diameter, and the axial thickness and the diameter should be no less than 90% of the size given here or adjusted size. When test piece is reused, users should cut thin layer once to clean the original surface before finishing test is conducted again.

### 3.1.3. Machining Technology Analysis

1.3.1 Workblank premachining

Shape and size of workblank are shown in Fig. 1. Before finishing, users should leave sufficient working allowance on the curved face, upper surface, and center cylindrical bore of cone frustum. For other sizes of workblank, refer to Fig. 1.

1.3.2 Test piece material, cutting tool, and cutting parameter can be specified in manufacturers' and users' agreement. When test piece is used for acceptance check, the following clauses can be adopted. However test piece is used, records should be kept.

1.3.3 Test piece material: Cast aluminum or other aluminum alloy materials.

1.3.4 Cutting tool: φ20 or φ16 hard alloy end mill, φ50 face mill, φ30 precision boring tool

1.3.5 Cutting parameters of simultaneous five-axis machining:

a) Cutting speed: 400m/min;

b) Feed amount: 0.05-0.10mm;

c) Cutting depth: 0.2mm.

1.3.6 Tool path planning

According to machining and detection needs, the test piece can be processed by 3 steps: 3+2 orientation machining of upper surface of cone frustum (φ50 face mill) and center boring hole machining (boring tool), simultaneous five-axis machining of curved side surface of cone frustum (φ20 or φ16 end mill). Tool path is planned with relevant CAM software, as shown in Fig. 3-3 and Fig. 3-4:

It should be noted that tool position point in the tool path and vector of tool axis should be uniform and smooth during simultaneous five-axis machining of the test piece.

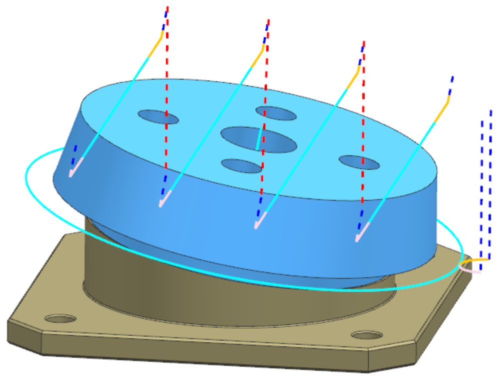


Figure 3-3 Tool path planning of test piece

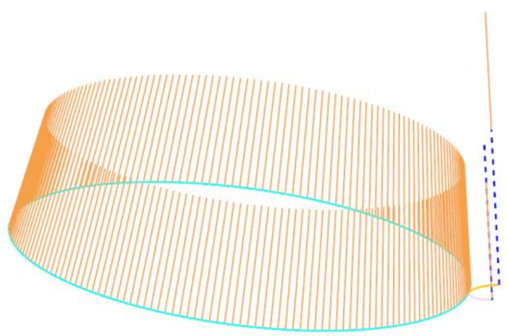


Figure 3-4 Tool path and tool axis vector of simultaneous five-axis

machining of curved side of test piece

### 3.1.4. Test of Test Piece

Test item and standard:

| | **Test item** | **Recommended allowance** | **Measured value** |
|---|---|---|---|
| **P1** | Roundness | 0.05 | |
| **P2** | Coaxiality | 0.05 | |
| **P3** | Angle | 0.005/25 | |
| **P4** | Contour | 0.1 | |
| **P5** | Roughness | Ra1.6 | |

Note: If size of workpiece changes, the above tolerance value can be changed accordingly

Test tool:

Coordinate measuring instrument.

Information record:

While testing NAS test piece, record the following information in the test report as completely as possible:

a) Test piece material;

b) Cutting tool;

c) Cutting speed;

d) Feed amount;

e) Cutting depth.

### 3.1.5. Common Problems

| | Problems | Possible cause |
|---|---|---|
| 1 | Coaxiality out-of-tolerance | Test positioning accuracy, repeatability, and RTCP calibration accuracy of machine tool |
| 2 | Contour out-of-tolerance | Test RTCP calibration accuracy of machine tool |
| 3 | Over cutting/undercutting | Test backlash, reverse jump, and speed loop gain of machine tool |
| 4 | Chatter mark | Check the tool, vibration of spindle, gain of all loops, and vibration suppression |

### 3.1.6. Description

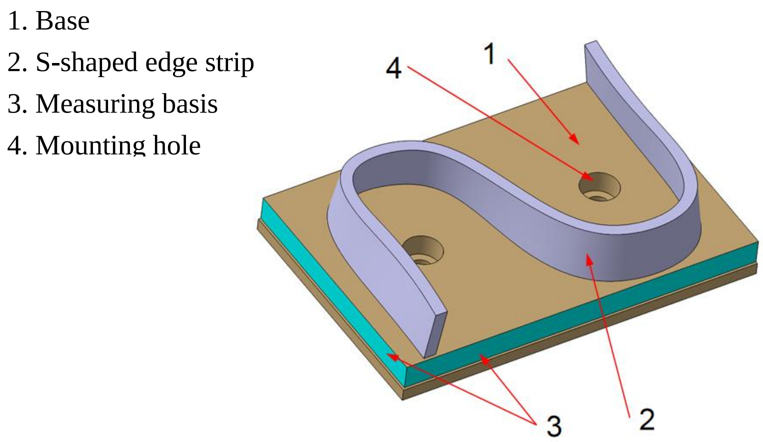
NAS test piece of cone frustum is the most commonly used test piece for dynamic accuracy test of five-axis CNC machine tool. For this test piece, 3+2 axes orientation machining and simultaneous five-axis machining are easy for accuracy test and RTCP accuracy test of linear axis and rotary axis. However, in the aspect of simultaneous five-axis machining, the test piece has shortcomings in inspecting the complex curved surface machining, dynamic acceleration and deceleration performance, dynamic rigidity and simultaneous multiaxis motion due to unity of change in taper and curvature. The “S” test piece becomes an effective supplement of dynamic accuracy and dynamic characteristic test method of machine tool due to its opening and closing angle change mechanism and continuous complex change of curvature.

## 3.2. S-shaped Test Piece

### 3.2.1. Basic Information of S-shaped Test Piece

S-shaped test piece is a five-axis test piece which is used to verify performance of machine tool during open and closed loop conversion, and better reflects comprehensive accuracy and dynamic response characteristic of motion parts during simultaneous multiaxis machining.

S-shaped test piece consists of a s-shaped straight-grained surface edge strip and a rectangular base, as shown below:




### 3.2.2. Model Analysis

Machining material: Aluminum alloy 6061    300mm×200mm×70mm

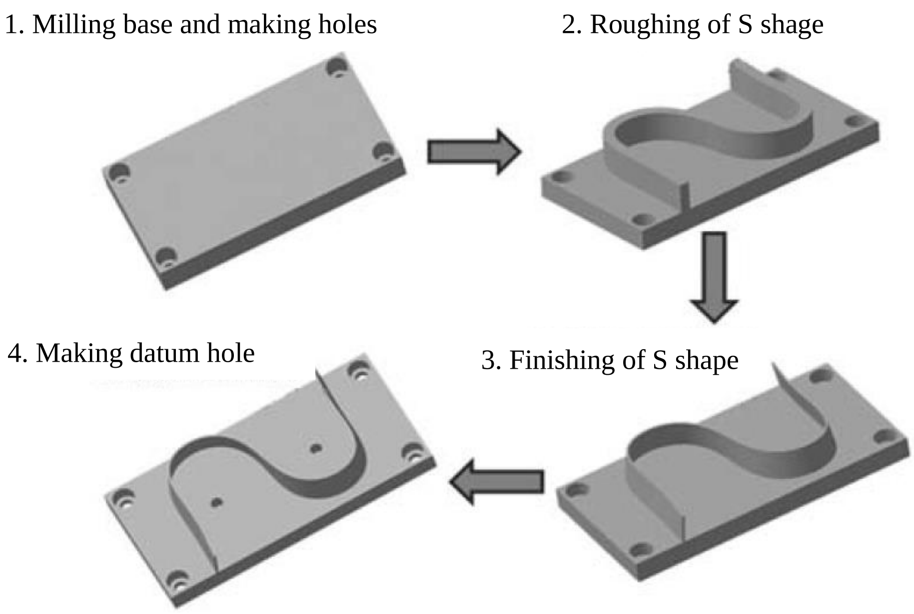
Tool and shank system: Generally D20R0 flat-bottomed tool is adopted. Select an appropriate tool according to material of workpiece. Such as Sandvik tool.

Test piece of type test consists of an S-shaped equal thickness edge strip and a rectangular base. There are 4 stepped holes for clamping on the rectangular base. 2 of them are positioning holes for installation, positioning, and measurement datum. Suppose the upper surface of the rectangular base is a Z=0mm plane, build a reference coordinate system with the positioning hole φ16H9 on the left as the center. The rectangular base is 30mm high and 4 stepped holes are at four corners. Equal thickness edge strip of S-shaped test piece is located on the rectangular base, 3mm thick and not vertical to the rectangular base.

### 3.2.3. Machining Technology Analysis

Material composition is a basic characteristic of machined part, and it determines selection of part machining method and cutting parameter. S-shaped test piece is made from aluminum alloy 6061, and 3mm thick. It is a thin-walled workpiece. Besides, edge strip is not vertical to rectangular base and is not a thin-walled part.

CNC process card is made based on material and structural characteristic of S-shaped test piece. S-sahped test piece can be processed by four steps: Rough machining of workblank (milling the base, making the compression hole); rough machining of S-shaped edge strip; finishing of S-shaped edge strip; datum hole machining. Basic flow chart of S-shaped workpiece machining is shown in Fig. 3-5.




| Step | Machining content | Machining method | Machining tool |
|---|---|---|---|
| 1 | Through hole | Drilling | Drill Φ20 |
| 2 | Counterbore hole | Drilling | Countersink Φ32 |
| 3 | Roughing of S-shaped face | Cavity milling | End millΦ32 |
| 4 | Finishing of S-shaped fac | Multiaxis milling | End mill Φ20 |
| 5 | Upper surface of rectangular base | Surface milling | Face mill |
| 6 | Center hole | Drilling | Drill Φ16 |

Figure 3-5 Basic process flow and tools

The tools:

Center drill: φ20mm, 65mm long, used to make positioning hole;

Drill: φ20mm, 65mm long, used to make φ20 through hole;

Drill: φ16mm, used to make datum hole;

Countersink: φ32mm, 50mm long, used to process φ32 counterbore;

Fillet end mill: φ32 mm, fillet radius: 3mm, mainly used for rough machining;

End mill: φ20 mm, mainly used for finishing of S-shaped edge strip;

Face mill: Finish-milling of datum surface.

Step 1 and step 2 are to process counterbore. That is, drill center hole using center drill, φ20 through hole using drill, and φ32 counterbore using countersink;

Step 3 is to create the cavity milling program to perform roughing with fillet end mill;

Step 4 is to finish S-shaped edge stripwith φ20 end mill. The tool path is shown in Fig.3-6.

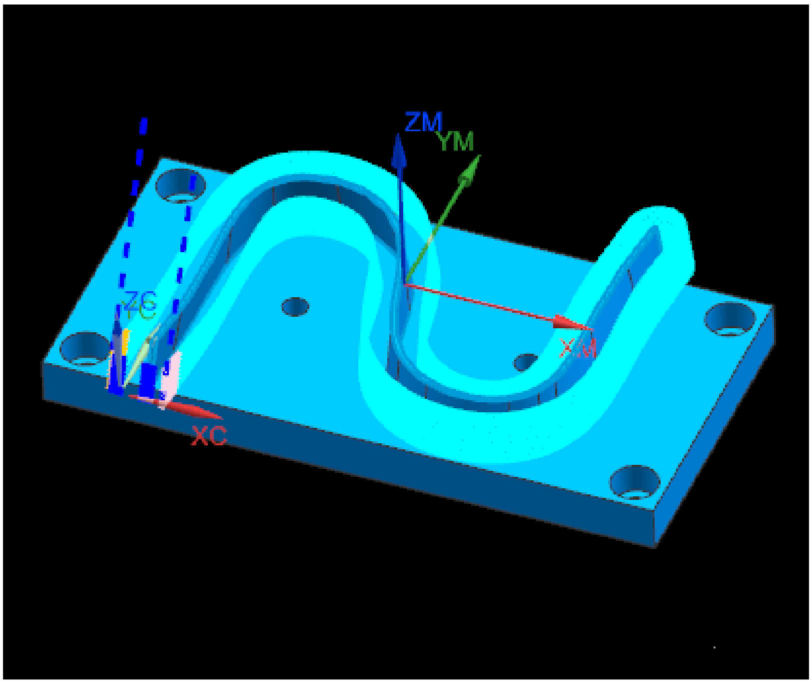


Figure 3-6 Finishing path of S-shaped edge strip

Whereas finishing program has a great impact on surface quality and machining accuracy of parts, only finishing parameters are listed:

| Part | Diameter of tool (mm) | Number of blades | Linear speed (m/min) | Feed per blade (mm/T) | Speed (r/min) | Feedrate (mm/min) |
|---|---|---|---|---|---|---|
| 1 | 20 | 2 | 408 | 0.07 | 6500 | 1000 |

Step 5 and step 6 are to finish datum surface and datum hole, and prepare for measurement of contour error of S-shaped test piece. It should be noted that the measurement datum is the reference element for three-coordinate measurement, which directly affects the test result. In actual machining, the machining datum should be consistent with the measurement datum, or the relative position relationship between the two should be accurately transmitted to the three-coordinate measuring department.

Aluminum tool must be adopted for machining. After the tool is installed in the shank, the radial run-out must be less than 0.005MM.

Bottom of processed materials must be flat and protected from deformation incurred during clamping in order to avoid three-coordinate test error.

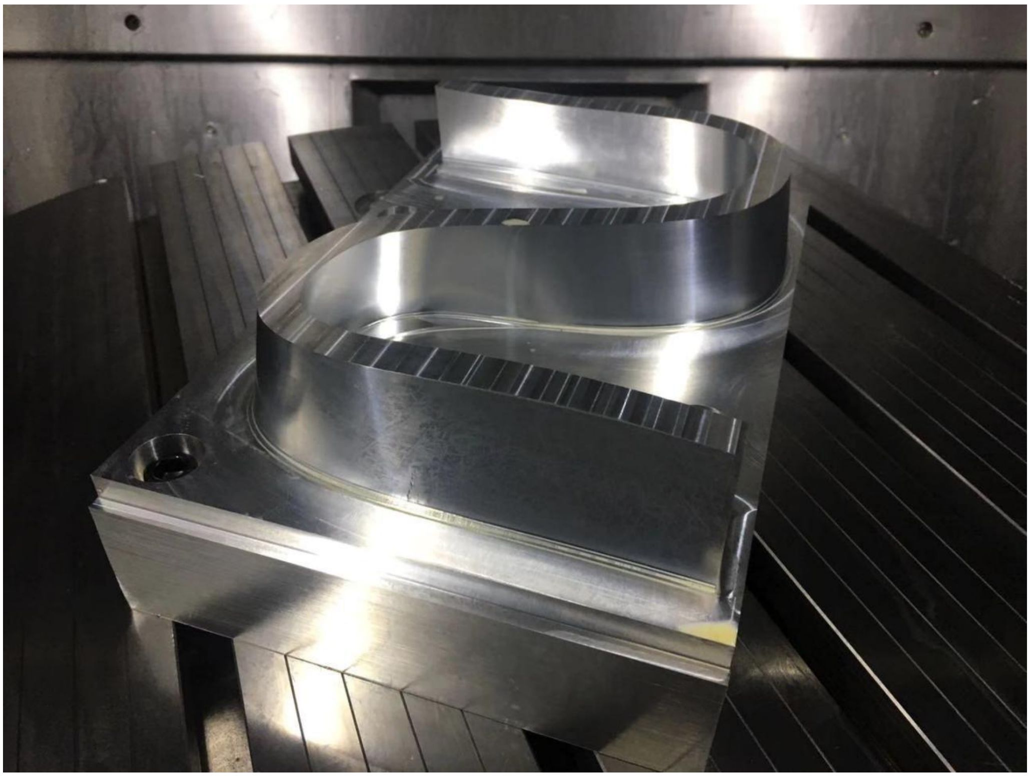


Figure 3-7 S-shaped test piece machining completion

### 3.2.4. Three-coordinate Test of S-shaped Test Piece

Deviation between machined surface and theoretical curved surface should be measured and calculated using three-coordinate measuring machine.

#### 3.2.4.1. Measurement datum

Place the S-shaped test piece on the platform of the measuring machine, and gently press part with 2-M16 bolt. A1-A4 points determine Z=0 plane, B1 and B2 points determine X straight line and Y=0 plane, C1 point determines, and the measurement coordinate system is established, as shown in Fig. 3-8.

It should be noted that the measurement datum is a reference element of three-coordinates measurement, which directly affects test result. In actual machining, the machining datum should be consistent with the measurement datum, or the relative position relationship between the two should be accurately transmitted to the three-coordinate measuring department.

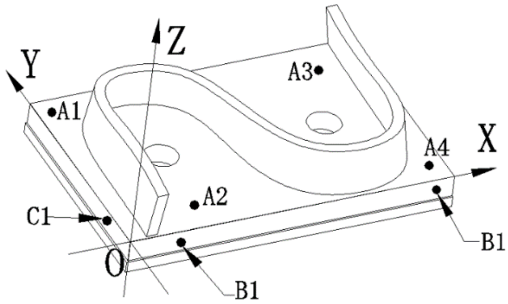



Figure 3-8 Establishment of measurement datum

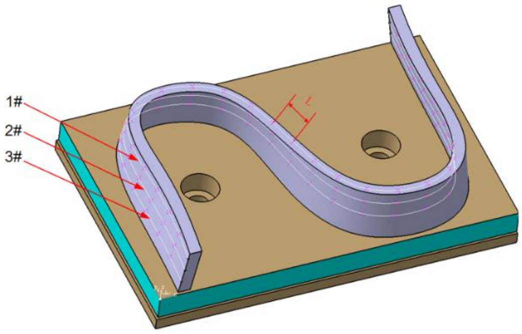



Figure 3-9 Selection of test point

### 3.2.4.2. Selection of test point

Take 3 transversals along the height direction of S-shaped edge strip, namely 1#4, 2#, and 3# transversals, and measure 30 points at equal distance L on each transversal, as shown in Fig. 3-9.

### 3.2.4.3. Measurement report

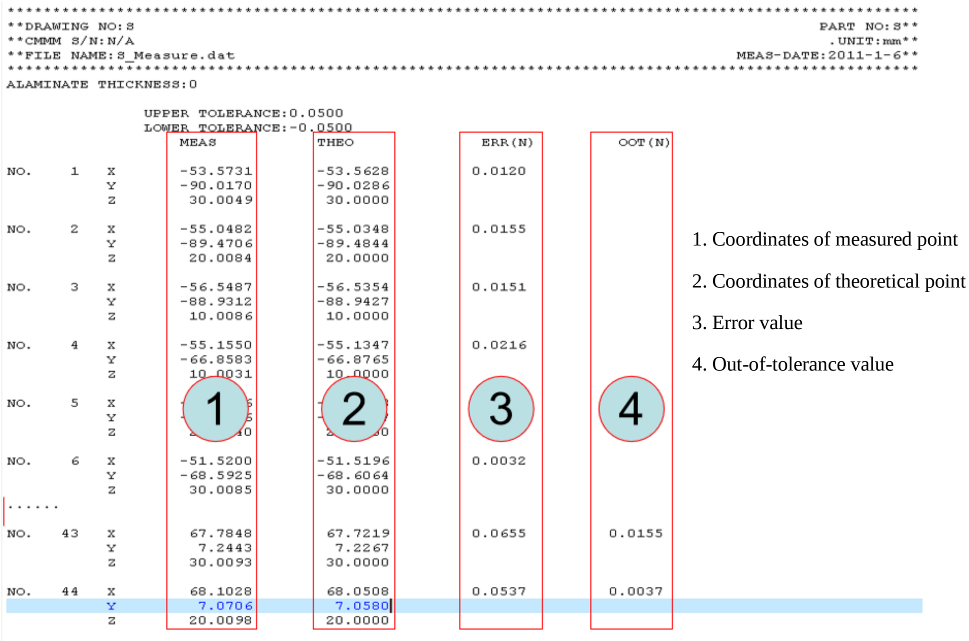



Figure 3-10 Test report of S-shaped test piece

## 3.2.5. Influence of machine tool accuracy on Processing of S-shaped Test Piece and Improvement Measures

Influence factor and phenomenon of directly or indirectly measured accuracy of machine tool on machining of "S" test piece

| | Accuracy test item | Performance of influence |
|---|---|---|
| 1 | Run-out of spindle | Tool ring and ripple on the surface of S test piece |
| 2 | Backlash of linear axes | Ripple in corresponding direction |
| 3 | Backlash of swivel axes | Pits formed by overcut of reversing part |
| 4 | Positioning accuracy and perpendicularity of linear axes | Deviation of overall test accuracy |
| 5 | Roundness error | Overcutting at corner |
| 6 | …… | |

Generally machining defects of S-shaped test piece are analyzed in the following three aspects:

①Whether the process is reasonable. The main consideration is to effectively control the swing stability of the tool axis and the accuracy of the point during the processing of the edge strip, so as to eliminate the influence

caused by the process;
②Modify system parameters to improve dynamic performance.
③Check and adjust geometric accuracy of machine tool, such as parallelism and perpendicularity between axes.

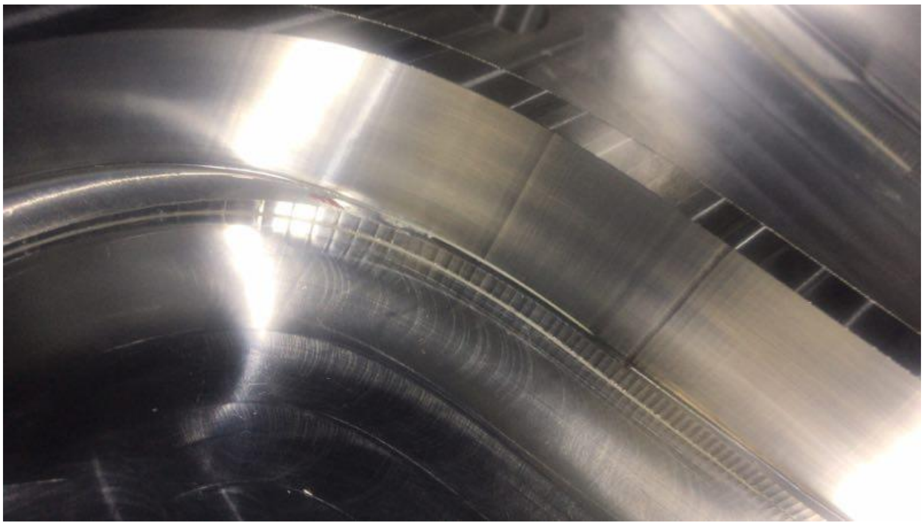


Figure 3-11 S-shaped piece with machining defects

Common problems of finishing program for edge strip of S-shaped test piece: the program step is uneven, and tool axis swings too much between steps, as shown in Fig.3-12 and Fig. 3-13.

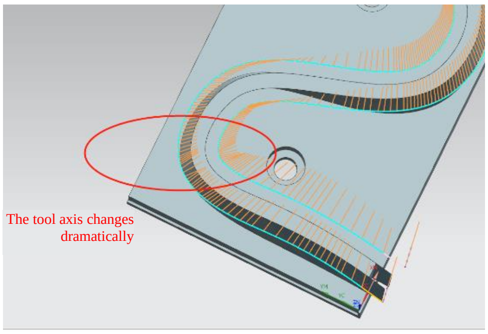



Figure 3-12 Abrupt change when tool axis swings

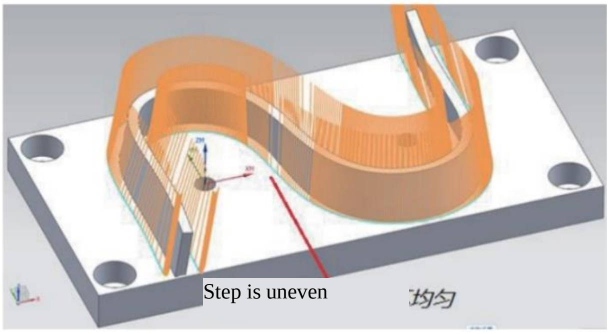



Figure 3-13 Uneven step

After pre-processing is adjusted in CAM software:

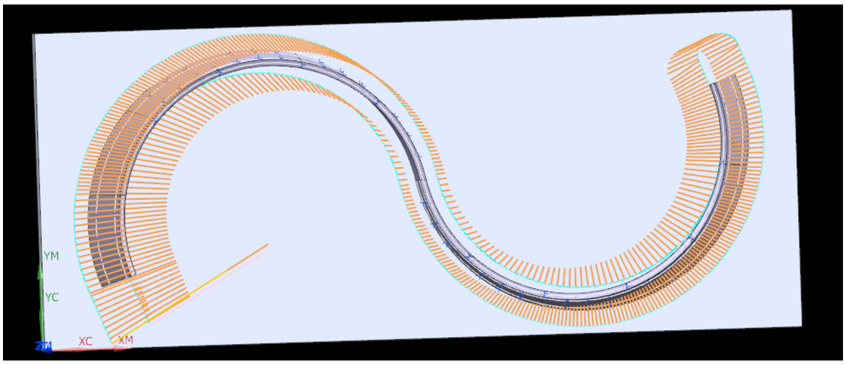



Figure 3-14 Machining path after adjustment

Modify process parameters and perform remachining to eliminate the effect of machining program on machining defect. If machining defects still exist, modify the following parameters to improve dynamic performance of machine tool:

① Before S-shaped test piece is processed, static geometric accuracy of machine tool meets manufacturer's delivery standard, and calibration accuracy of RTCP parameters of rotary axis is consistent with *Huazhongcnc Five-axis Machine Tool Calibration Specification*;

② Inspect whether position gain of 5 axes of machine tool is consistent, and keep position gain of all axes consistent with the minimum position gain when there is no noise of machine shaft;

③Properly modify "Number of command smoothing cycles" of the system for a smooth speed during machining. The reference value: “35”;

④Adjust system parameter "Allowable contour error of small line block planning" to “0.01”, and "Minimum smooth interior angle of corner" to “80-120”.

⑤Adjust system parameter "Centripetal acceleration" to “2500-4000”.

In order to ensure machining quality and fineness, if three-coordinate test still cannot meet the acceptance standard, it is required to test whether positioning accuracy of linear axis and parallelism of rotary axis and corresponding linear axis are limited within the allowable value.

# 4. IScope Software and Machining Analysis

During five-axis machining, workpiece may have poor surface quality due to problems of assembly, machining technology, and system. For quantitative analysis of such problems and easier commissioning, Huazhongcnc developed Iscope analysis software HNC system. Assisted by SSTT software, Iscope can quickly identify and solve problems through visualized analysis. Analysis procedures are as follows:

1) SSTT sampling. Collect command position and actual position of interpolation motion axis. The file format is .dat;

2) Import .dat file collected by SSTT into Iscope and carry out functional analysis using chromatogram.

## 4.1. Iscope Software Function

With real-time data sampling of HNC-8 five-axis CNC system and RTCP parameter in the system, 3D position visualization of command data and actual data can be realized, as well as the functions of data chart and 3D chromatogram, which are easy for analysis and evaluation of simultaneous moition performance of system, and applied to analysis and optimization of machining technology effectively.

1) Tool path display. Generate 3D tool path model based on machining code so that users can view it easily.

2) Display time domain waveform of machining data. Display the oscillogram of machining speed, error, current and other data through loading machining data file.

3) Machining parameter management.

### 4.1.1. Software Installation and Operating Environment

#### 4.1.1.1. Hardware

Minimum: Dominant frequency is above 1GHz, memory is greater than 1G, and graphics card supports OpenGL.

#### 4.1.1.2. Software

System requirements: Windows platform, system version is win7 or above.

#### 4.1.1.3. Software Installation

The software needs not run installation program. Users can directly run iScope.exe executable file initiator under program folder.

### 4.1.2. Using Instructions

#### 4.1.2.1. Software Interface

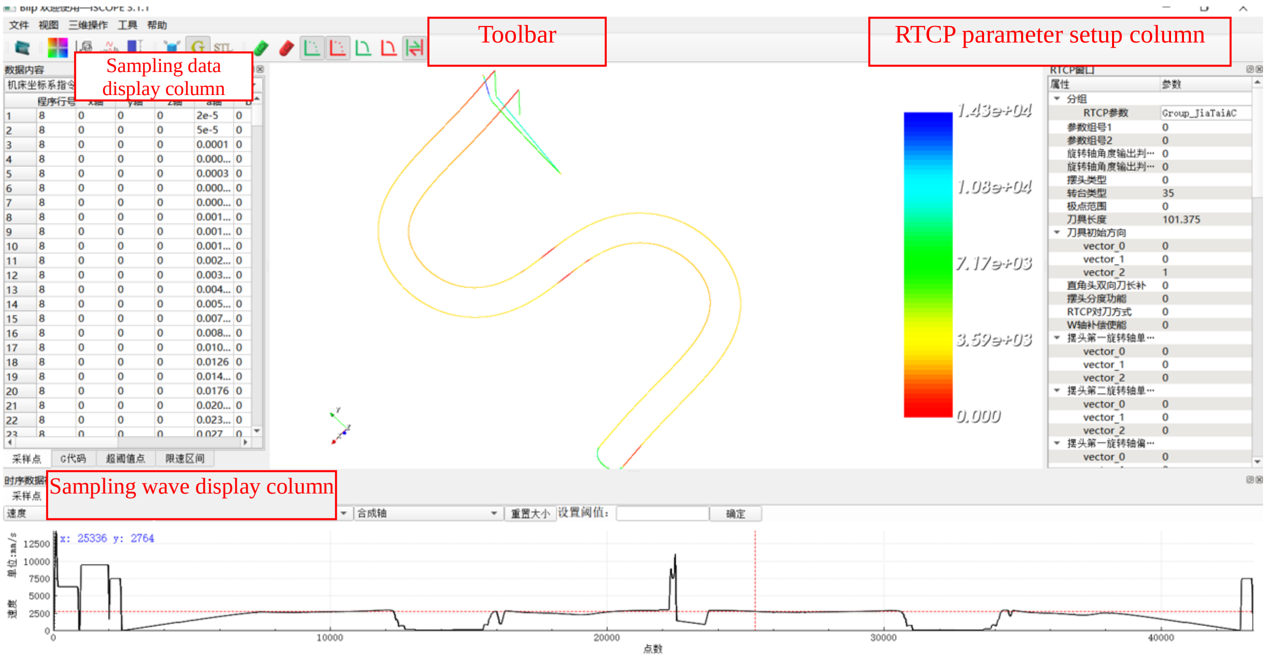



Figure 4-1 Iscope software interface

#### 4.1.2.2. 3D Tool path Display and Operation

3D tool path display can display tool machining track and enables translation, zoom, rotation and other operations of tool path.

## A. Data import

1. Select or set corresponding machine parameters in the properties window
2. Load sampling data file (*.dat) through "File"-"Open" in the menu bar. As shown in Fig. 4-2

Files supported by IScope include:

1) *.dat file is data of actual machining or simulated machining of HNC CNC system. After it is loaded, 3D path can be displayed, and speed, acceleration, jerk, current, tracking error, contour error and other time domain curves can be displayed by oscillogram.

2) *.NC; *.ptp machining code file, which can realize 3D display of tool position after loading

3) *.stl; common 3D model file, which can be displayed in the 3D window after loading.

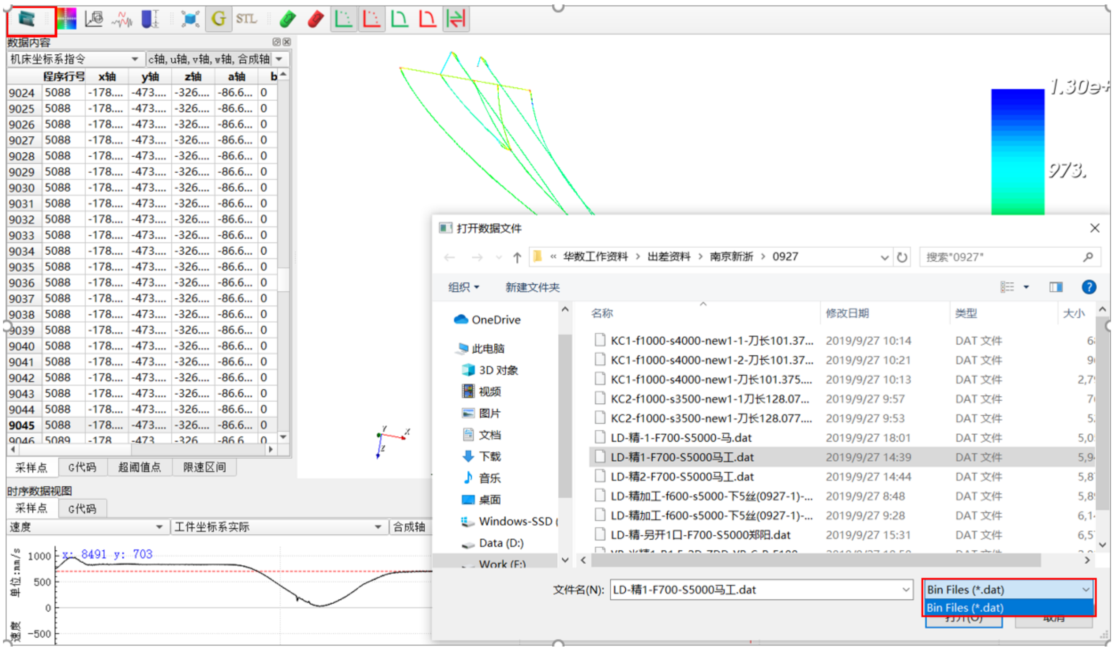


Figure 4-2 Open data file

3. After the data is loaded, geometric information of sampling point will be displayed in 3D view, as shown in Fig. 4-3.

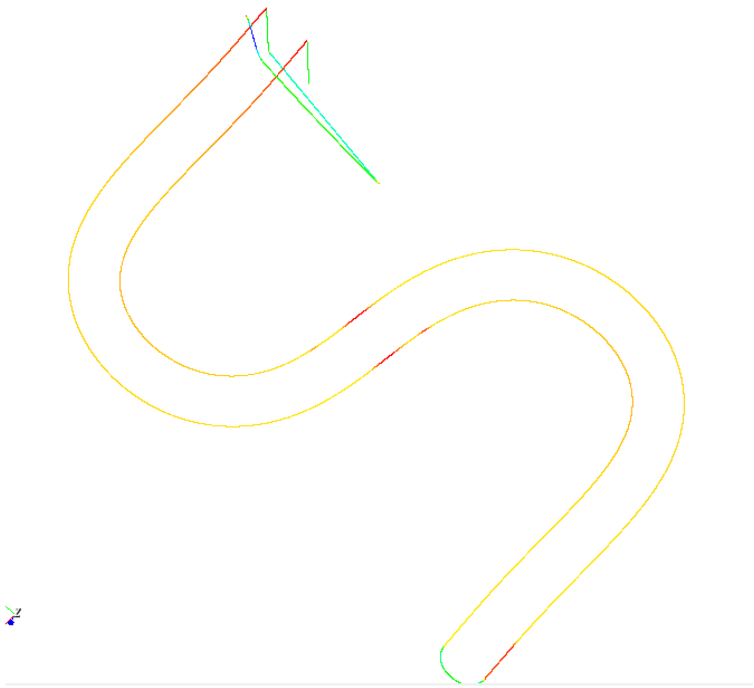


Figure 4-3 Displayed data point

## B. Operation of 3D model

Table 1-1 Graphics 3D operation instructions

| Action | Operation |
|---|---|
| Translation | Ctrl +mouse wheel, drag the mouse |

| | |
|---|---|
| Rotation | Press the center key, drag the mouse |
| Scaling | Roll the mouse wheel |
| Circle to amplify | Left mouse click to select an area |
| Select | Shift+ left mouse click |

Reset: Reset the display model by "Reset" in "3D operation" on menu bar.

## C. 3D data display toolbar

From left to right:

Open interpolation data file: Import sampling file into Iscope software

Chromatogram option: Chromatogram ON/OFF and relevant configurations;

Self-adaptive window: Adapt the graphics in the 3D window to the center of the window;

Display command tool axis vector: Display/hide tool axis vector of command sampling point;

Display actual tool axis vector: Display/hide tool axis vector of actual sampling point;

Command tool position: Display/hide sampled command tool position;

Actual tool position: Display/hide sampled actual tool position;

Command path: Display/hide the straight line between sampled adjacent command tool positions;

Actual path: Display/hide the straight line between sampled adjacent tool positions;

Switching between workpiece coordinate system and machine coordinate system: Sampling point is displayed in workpiece coordinate system/machine coordinate system;

### 4.1.2.3. Interpolation Data Analysis

## A. Machining data chromatogram display

Open data and enable color display function , and color display setup dialog box pops up, as shown in Fig. 4-4.


Figure 4-4 Color display setup dialog box

Fig. 4-5 is the chromatogram display results. (Color display diagram of X axis speed under the workpiece coordinate system)

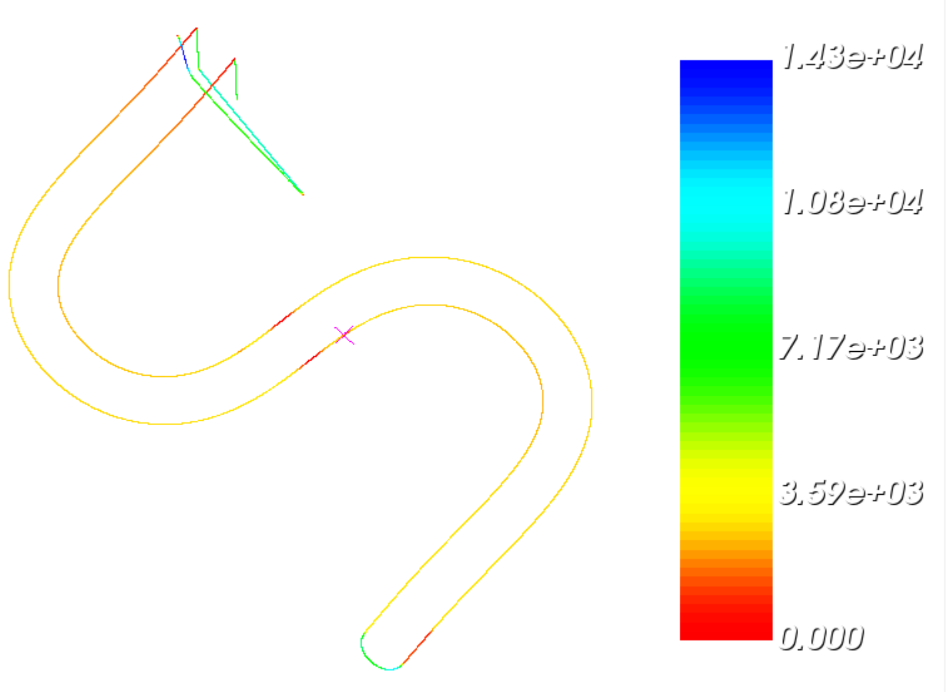



Figure 4-5 Display results

## B. Time domain waveform display of machining data

After data file is loaded, speed, acceleration, jerk, spindle current, tracking error, and other data of machine coordinate axis can be displayed in the oscillogram, and number of display points can be set. As shown in Fig. 4-6.

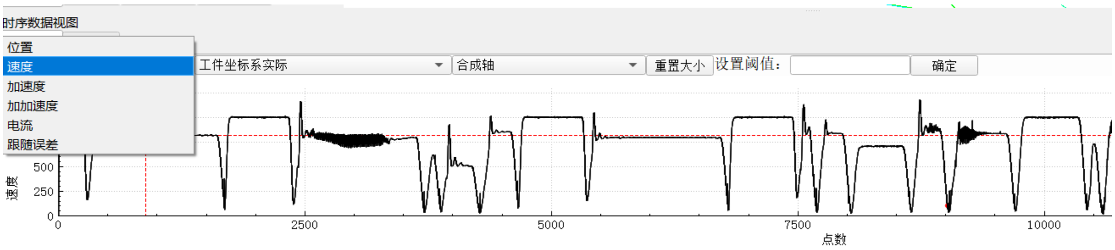



Figure 4-6 Oscillogram display

While the mouse moving on the oscillogram, X axis coordinate of the point pointed by the mouse and the corresponding data can be accurately displayed in real time, as shown in Fig. 2-6.

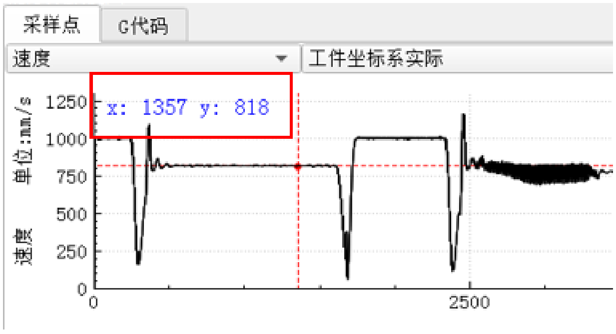



Figure 4-7 Real-time display of data of the point pointed by the mouse

①. move the mouse near the area to be amplified, slide the mouse wheel up and down and the curve will zoom in and out in equal proportion along X and Y directions

②. Place the mouse on X axis, move to the place near the projection of the selected area on the X axis, slide the mouse wheel and the curve will room in or out in one direction of X axis

③. Place the mouse on Y axis, move to the place near the projection of the selected area on the Y axis, slide the mouse wheel and the curve will room in or out in one direction of Y axis

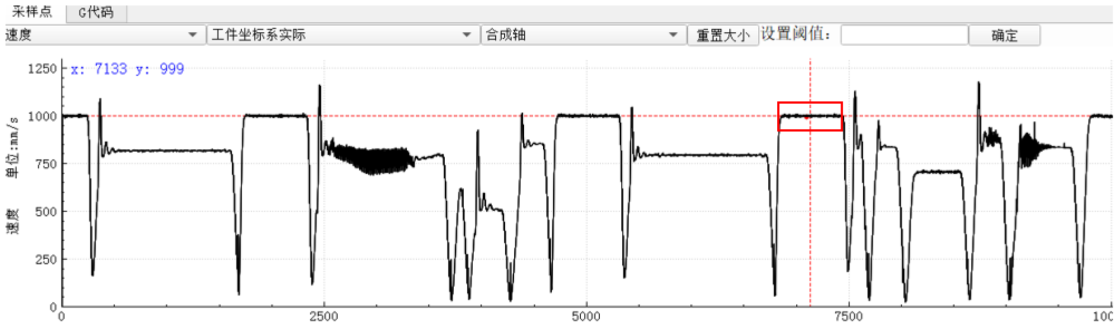



Figure 4-8 Select waveform in the red box

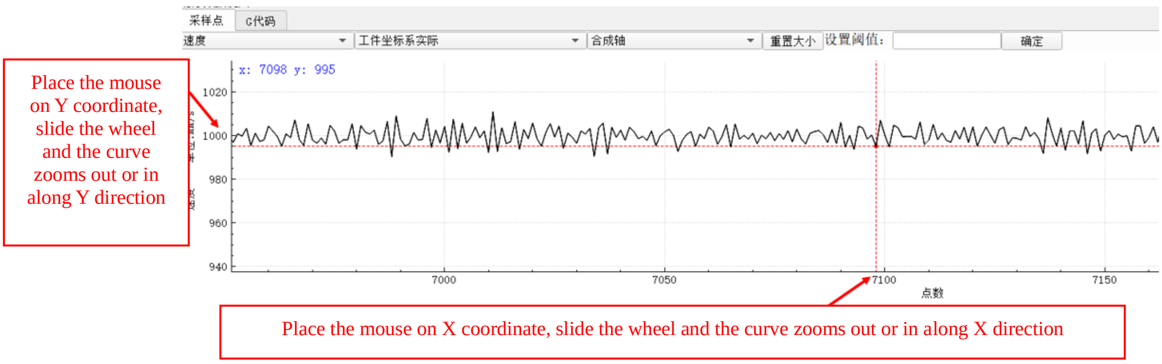



Figure 4-9 The amplified selected waveform

Double click on a point of the oscillogram and the point will be marked in 3D view and sampling point view simultaneously. As shown in Fig. 4-10.

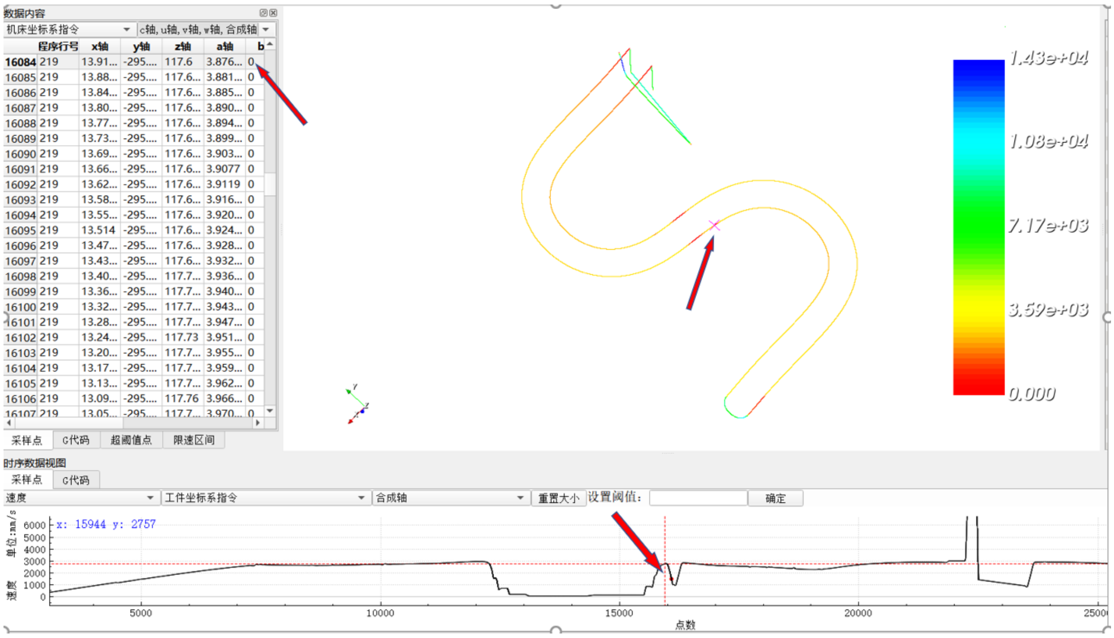



Figure 4-10 Double click to select a point

#### 4.1.2.4. Parameter Management

Parameter management function can be used to manage RTCP parameters and other machine parameters. Fig. 4-11 displays basic parameters, swivel head parameter, and rotary table parameters of RTCP parameters, which correspond to RTCP parameters of system channel parameters. Tool length corresponds to tool length compensation information of system. 3/4/5 of swivel head/rotary table type represents A/B/C respectively and has the same meaning as parameters in the CNC system.

Table 4-1 Machine type parameter setup

| Machine type / Iscope parameter | AC dual rotary table | CB dual swivel head | B swivel head C rotary table |
|---|---|---|---|
| Swivel head type | 0 | 54 | 4 |
| Rotary table type | 35 | 0 | 5 |

Iscope can support RTCP parameters of no more than 9 axes. The correspondence between RTCP parameters of Iscope software and RTCP parameters of the CNC system is shown below.

Table 4-2 Correspondence between RTCP parameters of Iscope software and RTCP parameters of CNC system

| **IScope parameter** | **CNC system parameter** |
|---|---|
| The first rotary axis of swivel head | Null |
| The second rotary axis of swivel head | The first rotary axis of swivel head |
| The third rotary axis of swivel head | The second rotary axis of swivel head |
| The first rotary axis of rotary table | Null |
| The second rotary axis of rotary table | The first rotary axis of rotary table |
| The third rotary axis of rotary table | The second rotary axis of rotary table |

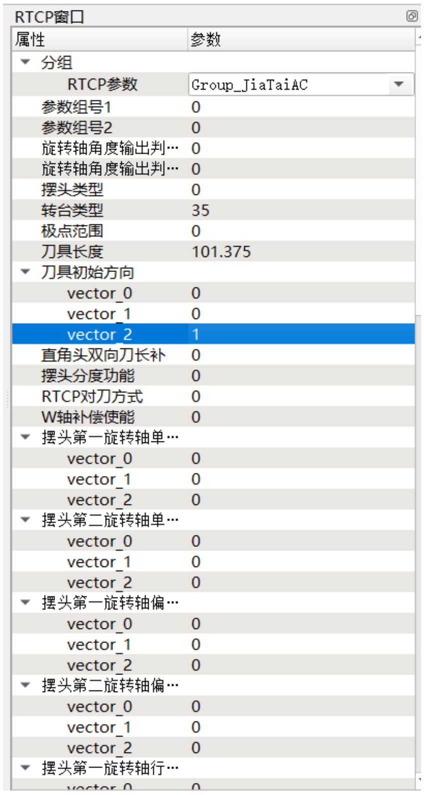



Figure 4-11 Parameter display

## A. Select parameters

The software can store multiple sets of parameters. While loading data, select corresponding parameter in column "RTCP parameter" to realize parameter switching.


Figure 4-12 Parameter selection

## B. Edit parameters

Select a parameter set to be edited and modify corresponding parameters.

## C. Parameter validation time

Parameters must be modified/selected before loading data. If parameters are modified after loading data, loaded data will not be validated.

To modify parameters of currently loaded data, data must be reloaded after parameter modification.

## 4.2. Iscope Problem Analysis and Solution

Solution and analysis method

Position loop gain and acceleration/deceleration time constant of 5 axes should be consistent

1) Observe abnormal machining point of workpiece to identify corresponding area in the chromatogram
2) Optimize system command speed
   ①. Check "Speed"- " workpiece coordinate system command" - "Combined axis", and determine whether the system command speed has a significant slowdown
   ②. If system command is abnormal, modify small line block parameters and optimize system command so that system command does not slow down obviously
3) If the abnormal machining area still exist, identify the faulted servo axis and optimize it
   ①. Check "Speed" - "Machine coordinate system actual" - "X axis" (check other axes successively) and identify the slowdown axis
   ②. After determining which axis is of abnormal deceleration, consider where the abnormal deceleration point based on the program. The abnormal deceleration point usually contains the position that a certain axis is in reverse direction or multiple axes are in reverse direction.
   ③. In case of reverse direction, consider program optimization to avoid reverse direction as far as possible
   ④. Optimize servo parameters of: Speed loop gain, position loop gain, and acceleration/deceleration time constant of axis

## 4.3. Iscope Problem Analysis Case

### 4.3.1. Impeller Machining

# A. Field problem description

The machine tool is an AC dual rotary table structure used to process impeller. Surface quality of finishing reaches customer’s requirements, but there are pits in the runner roughing, which is unacceptable for customer.

Analysis: To improve machining efficiency, the roughing path is a triangle and there are obvious sharp corners, so there are obvious pits

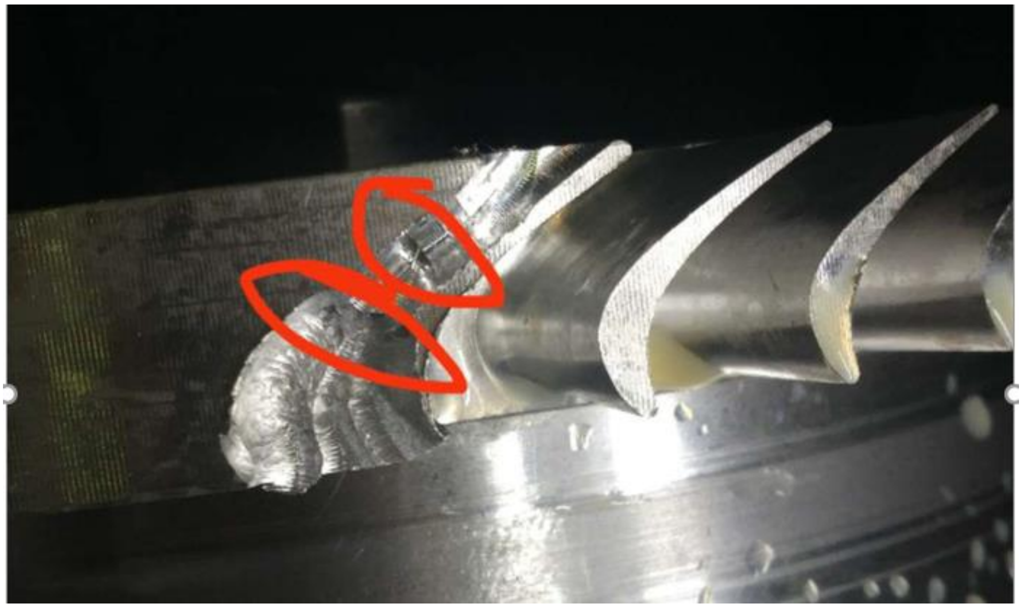
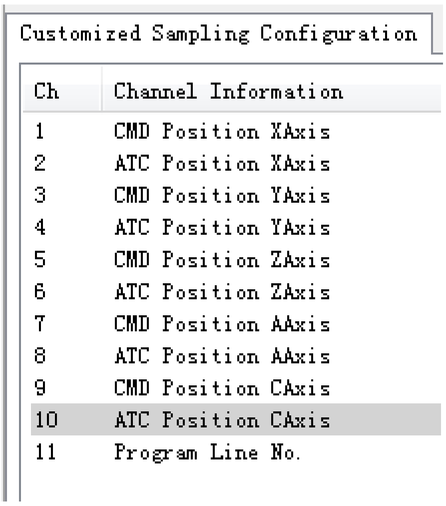


Figure 4-13 Picture of field machining before commissioning

Through analysis, pits in the blade are caused during rough machining, which are unacceptable for customer. The machine tool shakes obviously at the time pits.

## B. Commissioning process

1) Collect command position and actual position of each axis with SSTT software. The setup is as below.



Customized Sampling Configuration

| Ch | Channel Information |
|---|---|
| 1 | CMD Position XAxis |
| 2 | ATC Position XAxis |
| 3 | CMD Position YAxis |
| 4 | ATC Position YAxis |
| 5 | CMD Position ZAxis |
| 6 | ATC Position ZAxis |
| 7 | CMD Position AAxis |
| 8 | ATC Position AAxis |
| 9 | CMD Position CAxis |
| 10 | ATC Position CAxis |
| 11 | Program Line No. |

Figure 4-14 SSTT sampling setup

Sample type is saved in the format of dat

2) Import to Iscope software and enable chromatogram function. Observation results are as follows

According to chromatogram function, the corresponding position of pits is in red, indicating that the there is an obvious slowdown of speed

3) Curve analysis area setup: Check "Speed"- "Workpiece coordinate system actual"- "Combined axis" , and the curve fluctuates obviously

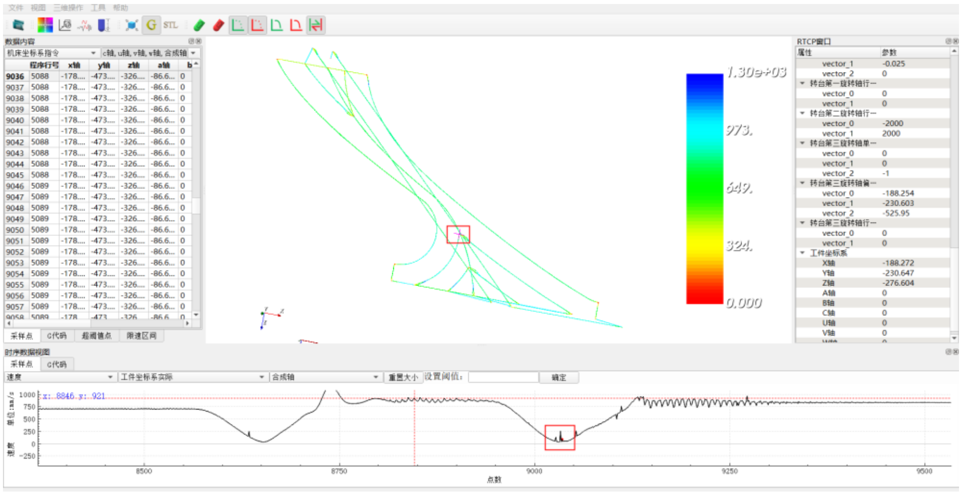



Figure 4-15 Pit-Actual resultant speed declines obviously

4) Check "Speed"- "Machine coordinate system actual"-"A axis", A axis speed turns from negative direction to positive direction, and A axis is reverse.

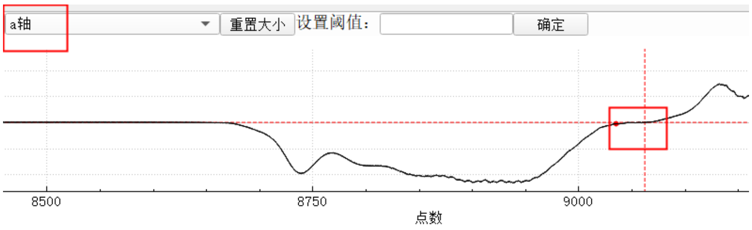



Figure 4-16 A axis is reverse and actual speed turns from negative to positive

5) Check "Speed" - "Machine coordinate system actual"- "Y axis" and Y axis speed fluctuates obviously

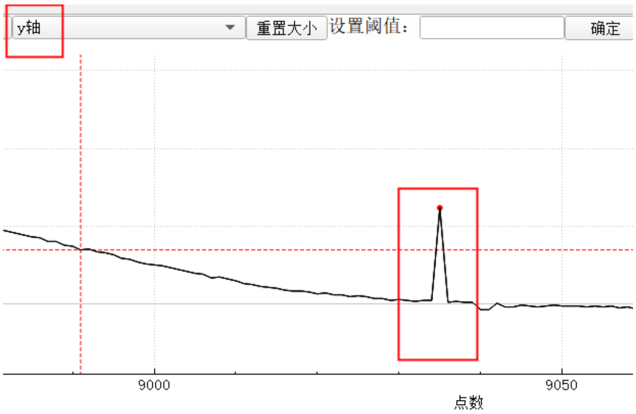



Figure 4-17 Abrupt change of Y axis speed

Thus, pits here mainly exist in A axis and Y axis

6) Focus on optimizing A axis and Y axis

Table 4-3 Servo optimization comparison table

| Parameter name / Parameter state | Position loop gain of each axis | Speed loop gain of A axis | Speed loop gain of Y axis |
|---|---|---|---|
| Before commissioning | 2000 | 1000 | 1500 |
| After commissioning | 3500 | 3000 | 3200 |

Machining effect improves obviously


Figure 4-18 Picture of machining effect

Slowdown problem of Iscope chromatogram improves obviously

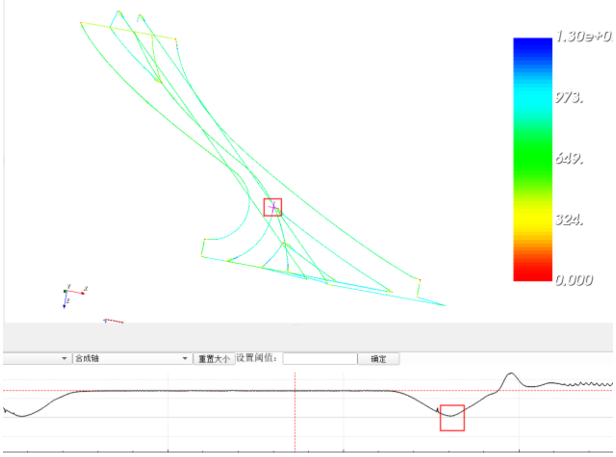



Figure 4-19 Machining abnormalities are obviously optimized via Iscope analysis

### 4.3.2. S-shaped Test Piece Machining

## A. Field environment and system parameter

For field machining of 6m S-shaped test piece, due to deep tool axis of S-shaped test piece, frequent reversing, servo dynamics of machine tool, forward tilt of A/B axis tool head by gravidty, and mechanical and structural dynamic, accuracy and surface quality will be affected. After field sampling analysis, adjust small line block parameters of system channel and servo matching of rotary axis and linear axis, and re-inspect geometric accuracy and RTCP accuracy of machine tool for the second machining test. Afterwards, the machining surface and accuracy are improved.

| | |
|---|---|
| Field device: | Jingrui AB dual swivel head |

## B. Commissioning process

Debugging and optimizing of the surface finish. There is no testing equipment, and it is not considered whether the accuracy reaches the standard.

**Problem 1: Adjust servo matching**

Some adjustments made for servo matching of rotary axis and linear axis.

BY axis has a large reverse jump 0.026mm before adjustment. B axis improves obviously and the reverse control is limited to 1 after adjusting PA3 speed proportional gain and integral time constant and applying static friction force of servo.

After adjustment, surface quality improves and vertical edges still exists and becomes smaller obviously.

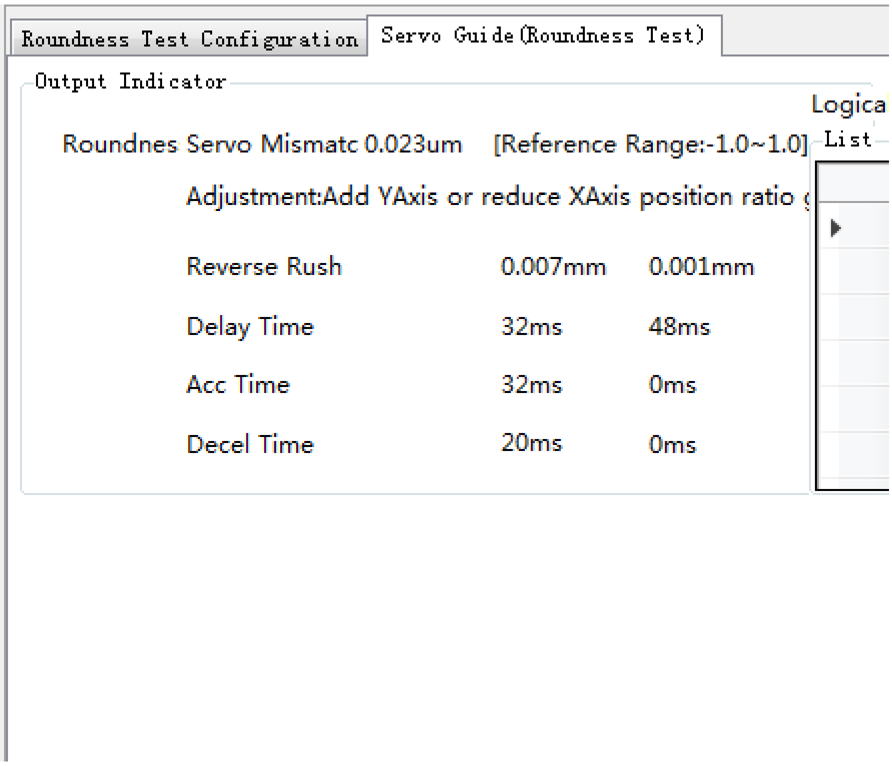



Figure 4-20 Servo matching commissioning

**Problem 2: 4 machining defects**

After servo optimization, there are still 4 vertical edges while machining S-shaped test piece at F2000 or F5000, as shown in below figures. There are 2 serious vertical edges, which are unacceptable for the user.

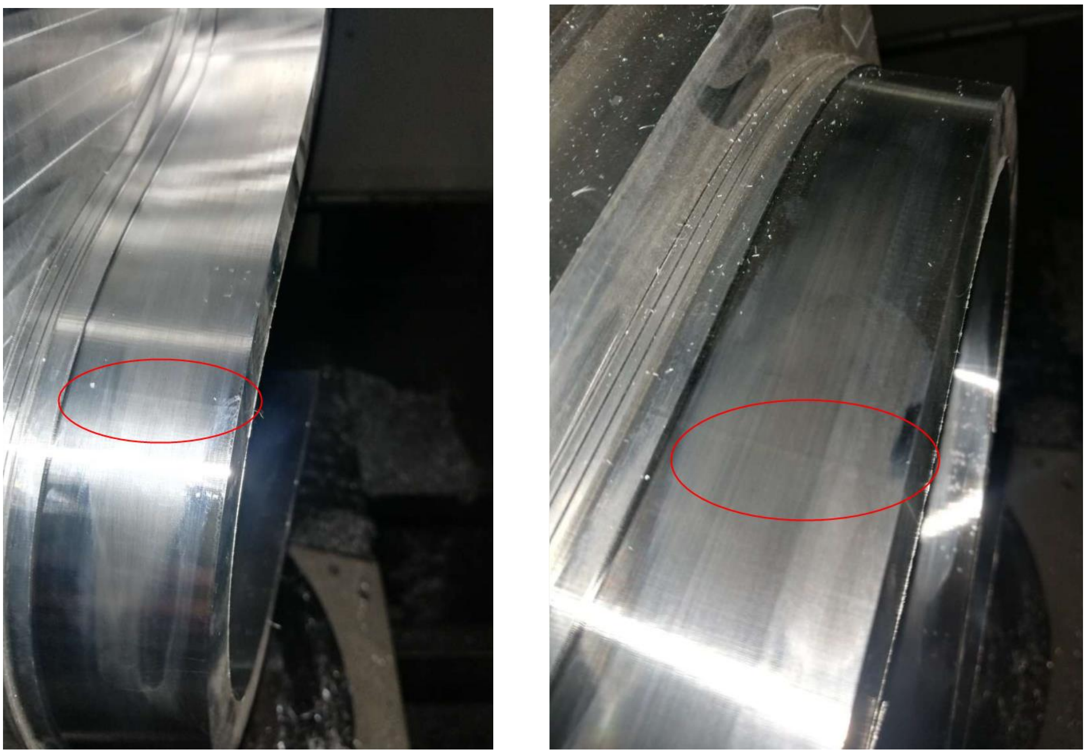


Figure 4-21 Vertical edges of S-shaped test piece

According to Iscope sampling observation, speed of tool nose does not fluctuate. Navigate to the specified line number according to actual machining defects.

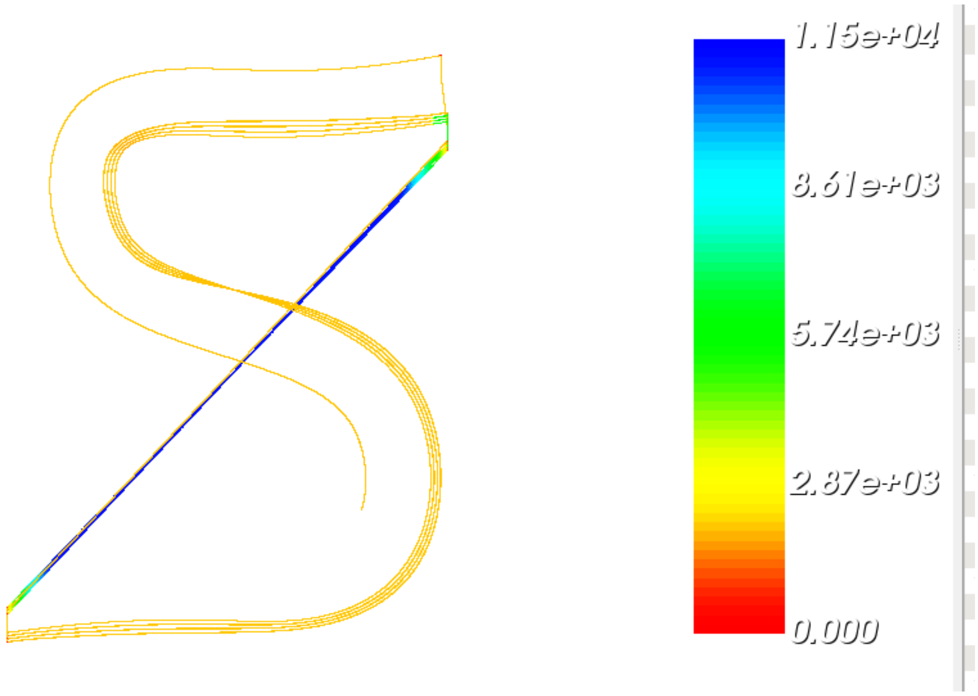



Figure 4-22 S-shaped test piece machining chromatogram

Four positions are shown below, where figure is line number:

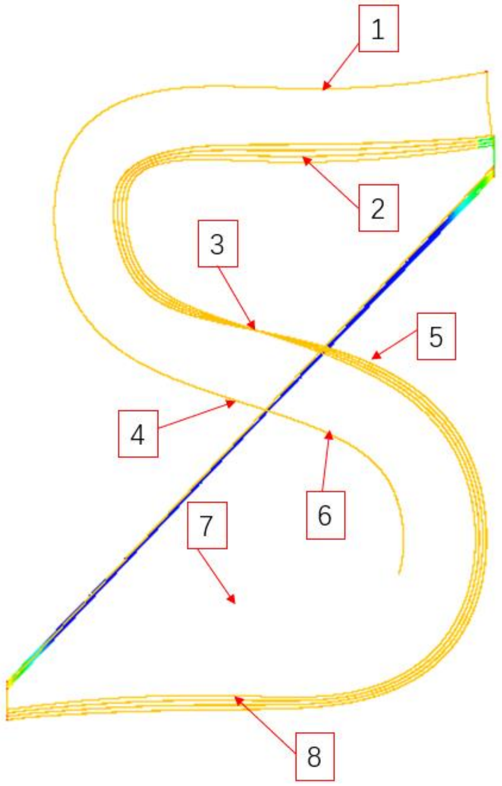



Figure 4-23 Vertical edge characteristic analysis

| SN | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| Program line number | N784 | N1521 | N1127 | N1859 | N1215 | N1953 | N1424 | N2169 |
| Program characteristic | Y/A axis reverse | Y/A axis reverse | A axis reverse | A axis reverse | A axis reverse | A axis reverse | Y/A axis reverse | Y/A axis reverse |

Thus, there may be a problem of A axis. It is found from speed waveform of iscope single axis that A axis speed fluctuates in the form of sawtooth. Full-closed loop control may result in great speed fluctuation. It is suspected to be caused by excessive acceleration at first, but it still exists after adjustment, as shown below

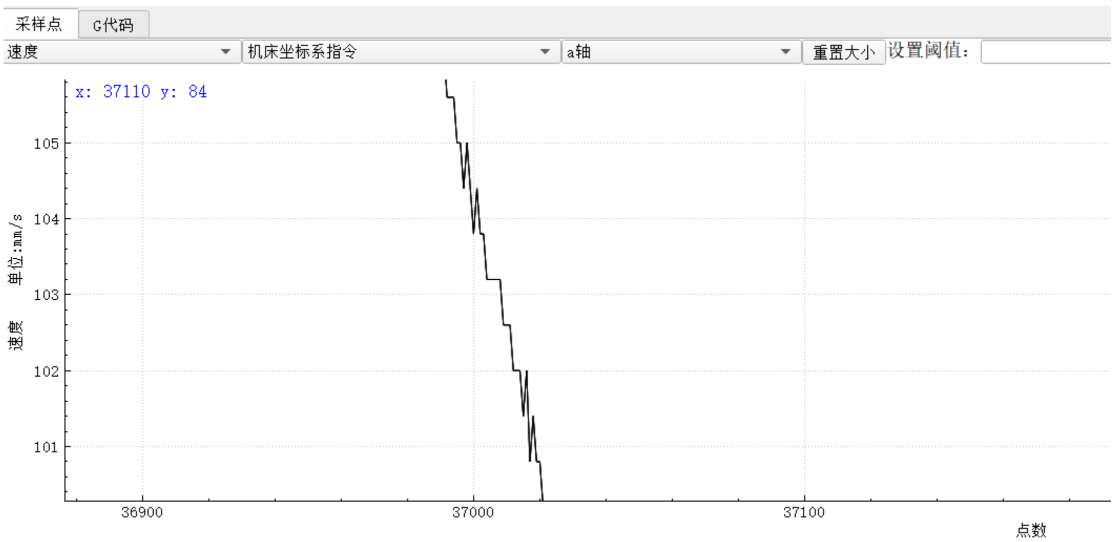



Figure 4-24 A axis speed of S-shaped test piece machining vertical edge

It is not caused by acceleration but excessively dense small line blocks. Jingrui modifies machining accuracy and maximum discontinuous angle, as shown below.

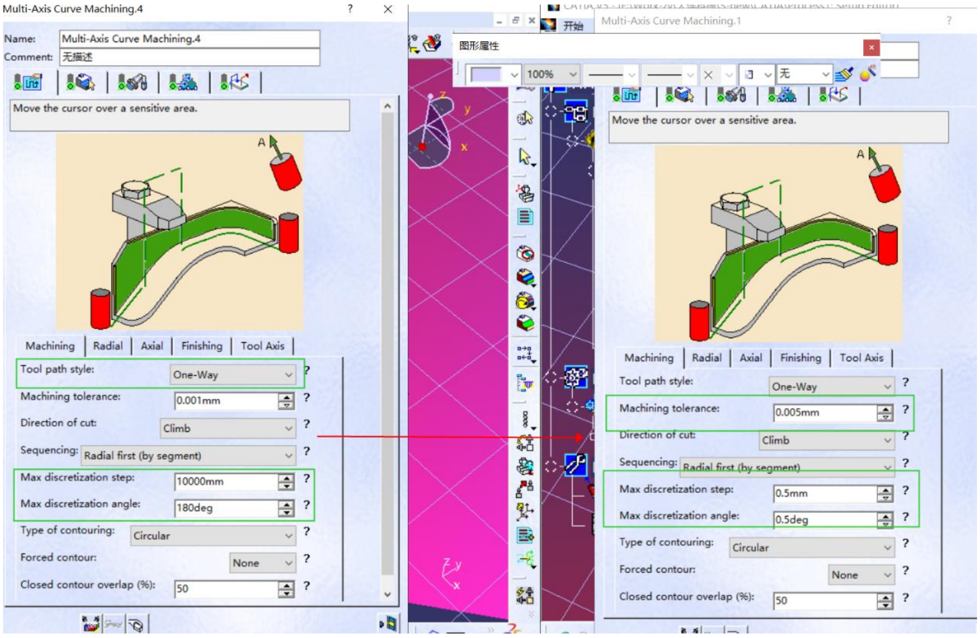



Figure 4-25 Modify machining technology

Machining is performed again after modification The machining effect is shown below, the significant improvement can be seen, and there are no obvious defects. T machining quality is accepted by the user Jingrui.

Figure 4-26 S-shaped test piece machining effect after optimization