# NK300CX-Phoenix Integrated CNC System Manufacturer Manual

Version: March 21, 2023 1st Edition
Author: Product Application Test Department


## Contents

# Preface

First of all, thank you for choosing **NK300CX-Phoenix Integrated CNC System**!

This manual gives a detailed introduction to the use of the **NK300CX-Phoenix Integrated CNC System**, including the host, operation panel and key panel, software main interface, electrical wiring, machine tool debugging, common operations, etc.

Before installing and using this product, please read this manual carefully, which will help you get familiar with the product quickly and use it better.

This product is subject to improvement or technical change without special notice. You can use the Weihong website http://www.weihong.com.cn check relevant information.

**Symbol Stipulation**

The following format is used to describe some contents that need attention when using this product.


This sign is used to provide warning information of device or ambient safety. If not avoided, it may cause device damage, data loss, device performance degradation or other unpredictable results.


This sign is used for contents requiring special attention except for safety, and provides some supplementary instructions and notes related to the text.

**Revision History**

| Version | Date | Reason |
|---|---|---|
| R1.0 | 2022-10-11 | 1st release |

# 1 System Introduction

The **NK300CX-Phoenix Integrated CNC System** (hereinafter referred to as NK300CX) consists of **Hardware** and **Software:**

**Hardware**

- NK300CX Host
- Operation panel and key panel
  - Ordinary type NK300CX-H：WH106C Operation Panel and WH201C Key Panel
  - Ordinary type NK300CX-V：WH108C Operation Panel
  - Woodworking special type：OP4425 Operation Panel and WH201C Key Panel
- Lambda controller
  Select according to axis configuration or software function：
  - Lambda 4S: pulse type, supporting up to 4 axes.
  - Lambda 5S: pulse type, supporting up to 5 axes.
  - Lambda 5E: pulse type, supporting up to 5 axes and absolute value.
  - Lambda 5M: bus type, supporting up to 8 axes and absolute value.
  - Lambda 21A: pulse type, supporting up to 5 axes.
  - Lambda 21B: bus type, supporting up to 16 axes and incremental type.
  - Lambda 21E: ETC bus type, supporting up to 32 axes and incremental type.
  - EX31A: It is configured according to port requirements.
- Type D two-row cable DB9M/F.

For the connection diagram between each hardware, please see NK300CX Connection Diagram.

**Software**

The system software is based on the Phoenix (V15) platform.

- Three-axis software, four-axis software (RTCP), five-axis software (RTCP), double-Z software
- Each software is compatible with three panels

This article mainly uses the **Five-axis RTCP Software** as a reference to introduce and explain.

For the software main interface, please see Software Main Interface.

## 1.1 NK300CX Host

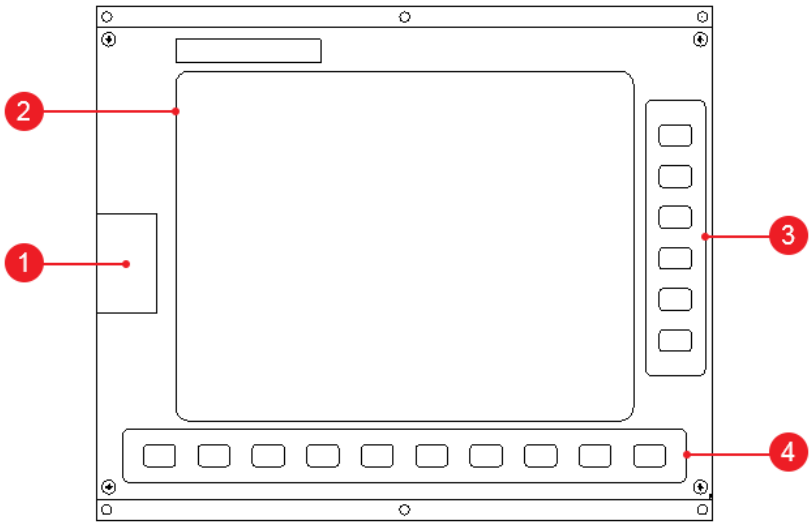
This section describes the layout and dimension drawing of the NK300CX host.

### 1.1.1 Layout

Including front and back layout.

#### 1.1.1.1 Front

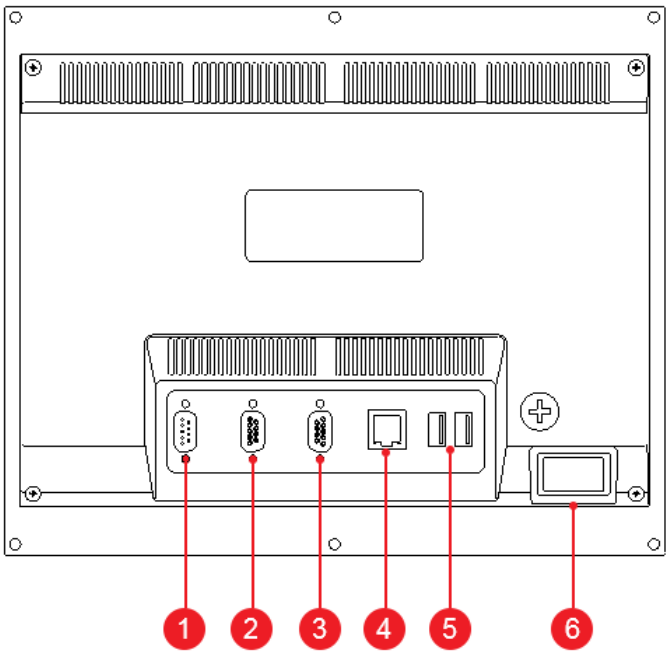
The diagram of the front of the NK300CX host is as follows:




| No. | Name | Description |
|---|---|---|
| 1 | USB interface | After opening the cover of the USB interface, you can connect the mouse, keyboard and USB flash disk. |
| 2 | UI | Friendly interface and easy operation. |
| 3 | Function selection key area | There are six functional areas, namely, **Machining, Advanced, Program, System, Parameter** and **Diagnosis**. Select the corresponding function key to enter the main interface of the function area. |
| 4 | Operation key area | Including **F1**~**F8** and PgUp and PgDnkeys.<br>It is used to activate the 8 horizontally arranged software functions and turn pages at the bottom of the user interface. |

### 1.1.1.2 Back

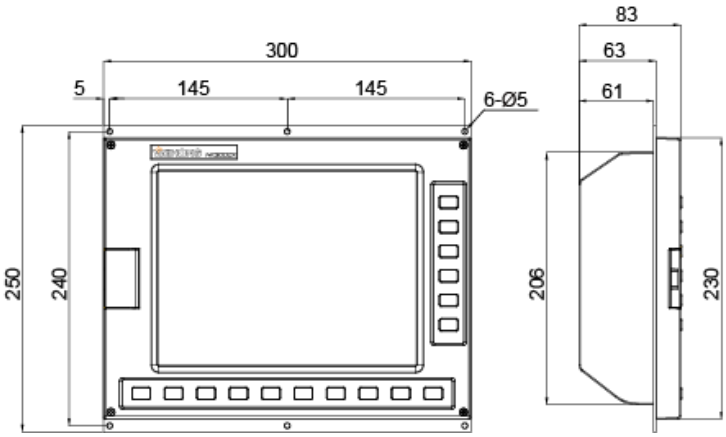
The diagram of the back of the NK300CX host is as follows:



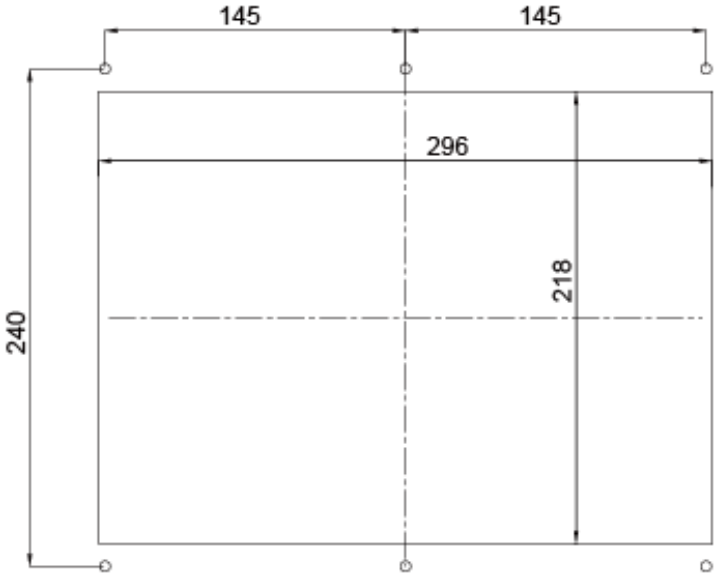
| No. | Name | Description |
|---|---|---|
| 1 | DB9 core interface | NK300CX-H system: used to connect the key panel.<br>NK300CX-V system: used to connect the operation panel.<br>Woodworking special type: used to connect the operation panel. |
| 2 | Controller interface | Used to connect the Lambda controller. |
| 3 | VGA interface | It is used to connect VGA devices, which can be connected as required (normally not connected). |
| 4 | Network port | The transmission rate is 100Mbps. |
| 5 | USB interface | There are 2 for external mouse, keyboard and USB flash disk. |
| 6 | Power supply interface | Used to connect 220V power supply. |

## 1.1.2 Dimension Drawing (unit: mm)

NK300CX display panel diagram




Dimension drawing of metal plate trepanning



# 1.2 WH106C Operation Panel and WH201C Key Panel

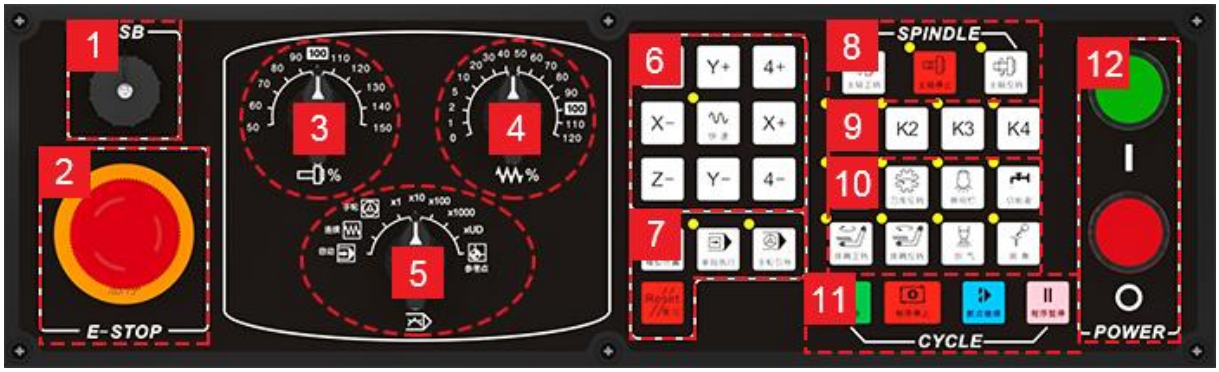
This part introduces the layout and dimension drawing of the WH106C operation panel and WH201C key panel used by the **Ordinary Type NK300CX-H**.

## 1.2.1 Operation Panel

Introduce the layout and dimension drawing of WH106C operation panel.

### 1.2.1.1 Layout

The diagram of the front of the operation panel is as follows:

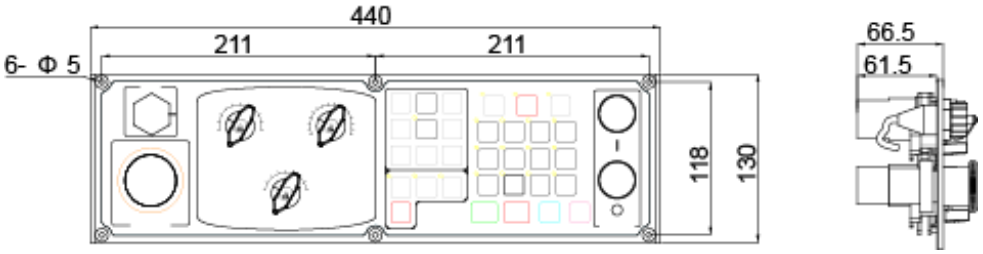



| No. | Name | Description |
|---|---|---|
| 1 | USB interface | Used to connect removable disks. |
| 2 | E-stop switch | When the machine is in a dangerous state, it is used to stop the operation of the machine tool and protect the safety of the person and device. After the danger is removed, the E-stop alarm can be released by rotating the button clockwise. |
| 3 | Spindle override knob | Used to adjust the spindle speed (50~150%). |
| 4 | Feed override knob | It is used to adjust the feed speed (0~120%). |
| 5 | Mode selection knob | Used to select the machining mode, including auto, jog, handwheel, step, reference point. |
| 6 | Axis direction key | It is used to manually move each axis of the machine tool or jiggle function. In **Manual** mode:<br>• After pressing the **Rapid** key, press any axis direction key, and the machine tool will move at high speed manually.<br>• Only press any axis direction key, and the machine tool will move at a low speed manually. |
| 7 | Common operation key | It is used to perform common operations, including **Simulation**, **Single Segment Execution**, **Handwheel Guide** and **Reset** keys. |
| 8 | Spindle control key | Used to control spindle movement, including **Spindle CW**, **Spindle Stop** and **Spindle CCW** key. |
| 9 | Extended key area | It is used to customize key functions, including K1, K2, K3 and K4 extension keys. |
| 10 | Common port key area | Used to ON/Off common ports. |
| 11 | Motion control key | **Program Start, Program Pause** and **Program Stop** are used to control the machine tool movement**.**<br>**Breakpoint Resume** is used to quickly move the machine tool |

| No. | Name | Description |
|---|---|---|
| | | to the breakpoint and continue to execute the program from the breakpoint line in case of power failure, E-stop and other error conditions during machining. |
| 12 | Power supply switch | The switch that controls system power. |

### 1.2.1.2 Dimension Drawing (unit: mm)

Dimension drawing of WH106C operation panel



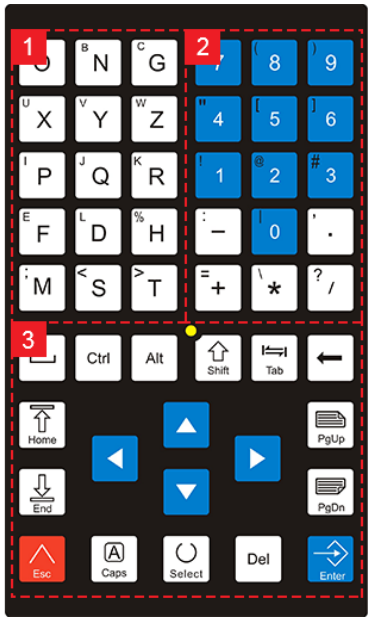

Dimension drawing of metal plate trepanning


## 1.2.2 Key Panel

Introduce the layout and dimensions of the WH201C key panel.

### 1.2.2.1 Layout

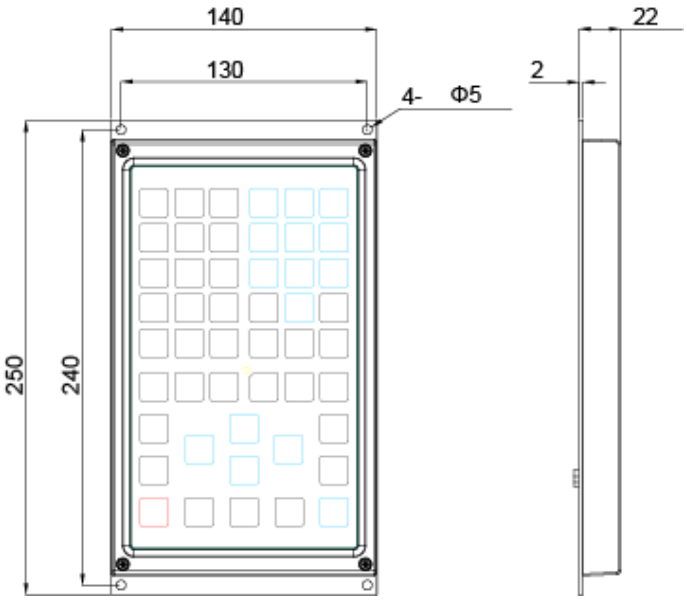
The front view of the key panel is as follows:



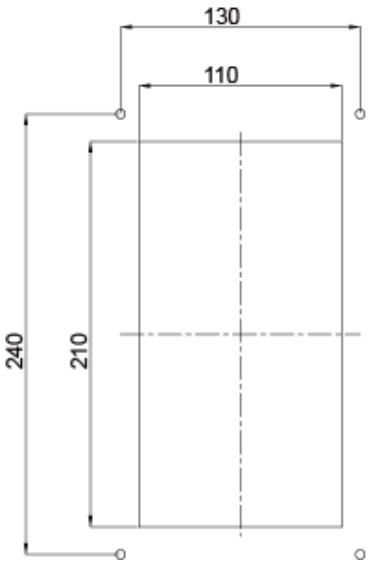

1. Character key area 2. Direction and number key area 3. System operation key area

#### 1.2.2.2 Dimension Dawing (unit: mm)

Dimension drawing of WH201C operation panel




Dimension drawing of metal plate trepanning

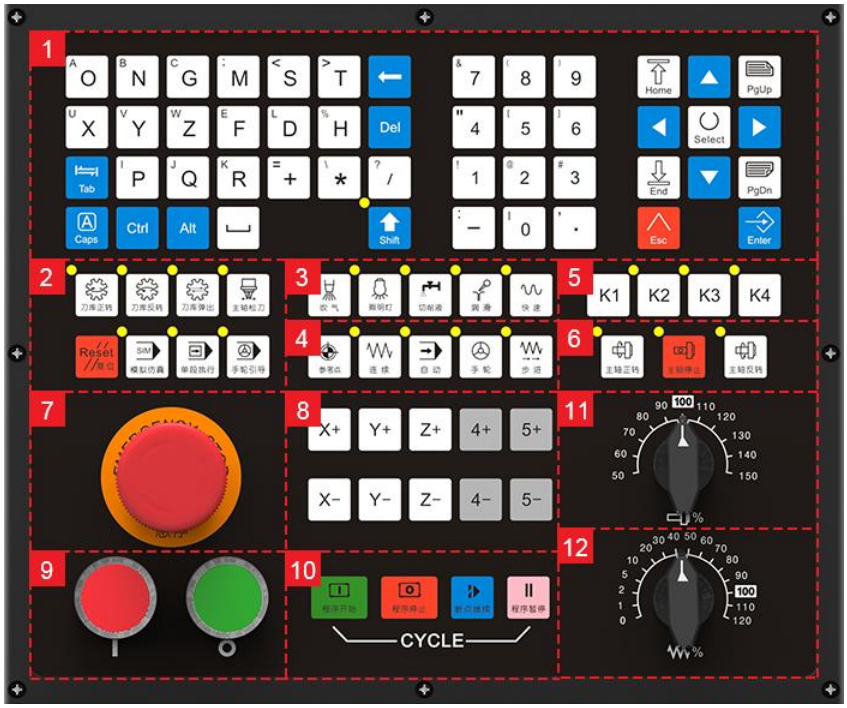



## 1.3 WH108C Operation Panel

This section describes the layout and dimension drawing of the WH108C operation panel used by the **Ordinary Type NK300CX-V**.

### 1.3.1 Layout

The front view of the operation panel is as follows:

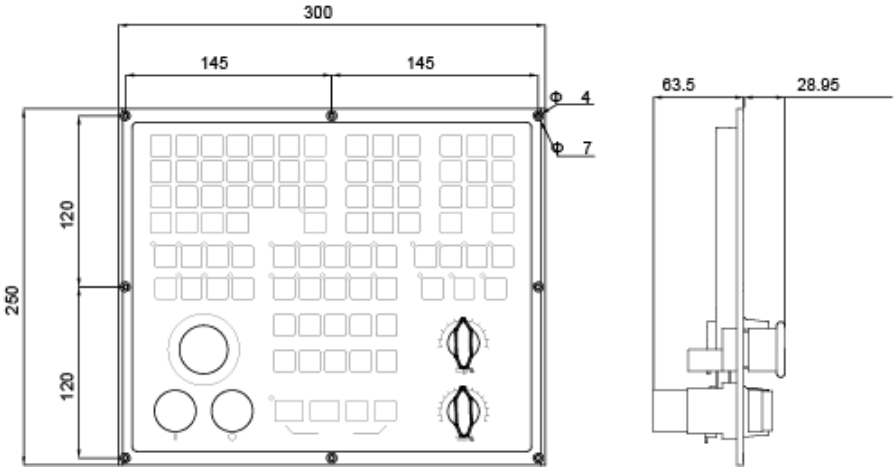



| No. | Name | Description |
|---|---|---|
| 1 | Key panel | Used to enter characters. The basic rules of use are the same as the computer input keyboard:<br>• Press the **Character** key and enter the lower right character.<br>• Press **Shift**+character key and enter the upper left character. |
| 2 | Common operation key | It is used to enable common functions, including **Magazine CW**, **Magazine CCW**, **Magazine Pushed Out**, **Spindle Unclamp Tool**, **Reset**, **Simulation** and **Single Segment Execution** keys. |
| 3 | Common port key | Used to enable common ports, including **Blow**, **Lamp**, **Cutting Fluid**, **Lube** keys, and **High Speed Movement**, including **Rapid** key. |
| 4 | Mode selection area | It is used to select the machining mode, including **Reference Point**, **Jog**, **Auto**, **Handwheel** and **Step** mode. |
| 5 | Extended key area | It is used to customize key functions, including K1, K2, K3 and K4 extension keys. |
| 6 | Spindle control key | Used to control spindle movement, including **Spindle CW**, **Spindle Stop** and **Spindle CCW** keys. |
| 7 | E- stop switch | When the machine is in a dangerous state, it is used to stop the operation of the machine tool and protect the safety of the person and device.<br>After the danger is removed, the E-stop alarm can be released by rotating the button clockwise. |
| 8 | Axis direction key | It is used to manually move each axis of the machine tool or jiggle function. In **Manual** mode:<br>• After pressing the **Rapid** key, press any axis direction key, and the machine tool will move at high speed manually.<br>• Only press any axis direction key, and the machine tool will |

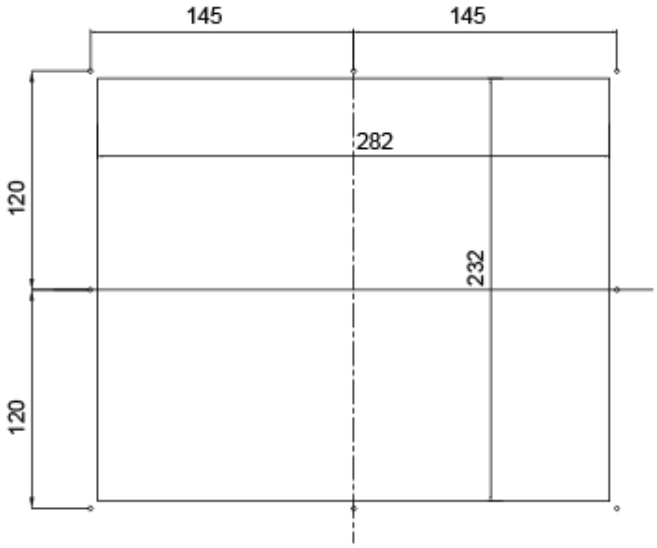
| No. | Name | Description |
|---|---|---|
| | | move at a low speed manually. |
| 9 | Power supply switch | The switch that controls system power. |
| 10 | Motion control key | **Program Start, Program Pause** and **Program Stop** are used to control the machine tool movement**.**<br>**Breakpoint Resume** is used to quickly move the machine tool to the breakpoint and continue to execute the program from the breakpoint line in case of power failure, E-stop and other error conditions during machining. |
| 11 | Spindle override knob | Used to adjust the spindle speed (50~150%).<br>When **G00 Speed Fixed** is disabled, this knob is actually G00 override knob. The spindle override of 50%~150% corresponds to G00 override of 0%~100%. |
| 12 | Feed override knob | It is used to adjust the feed speed (0~120%). |

## 1.3.2 Dimension drawing (unit: mm)

Dimension drawing of WH108C operation panel




Dimension drawing of metal plate trepanning



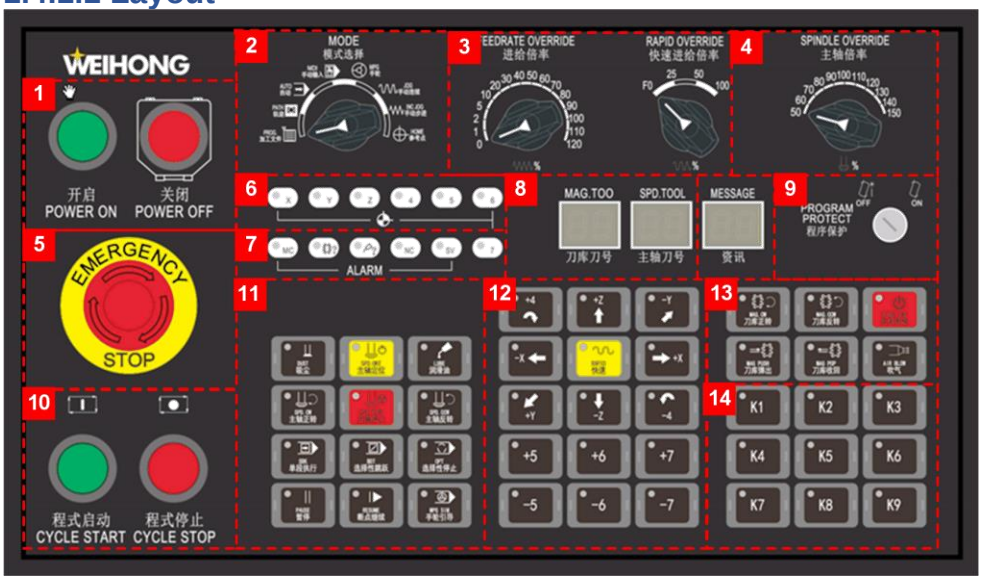
# 1.4 OP4425 Operation Panel and WH201C Key Panel

This section introduces the layout and dimension drawing of OP4425 operation panel and WH201C key panel used by **Woodworking Special Type NK300CX**.

## 1.4.1 Operation Panel

Describe the layout and dimension drawing of OP4425 operation panel.

### 1.4.1.1 Layout

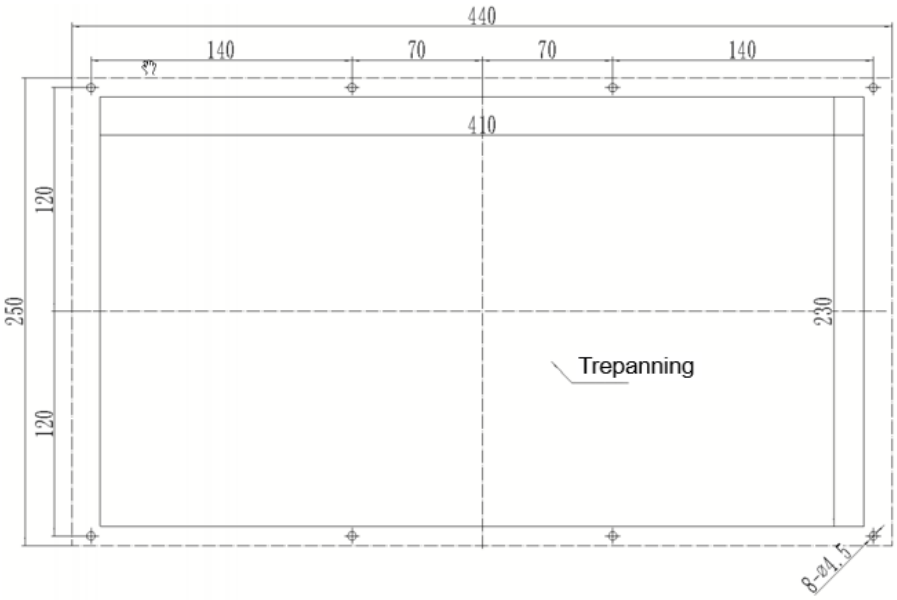



| No. | Name | Description |
|---|---|---|
| 1 | Power supply switch | The switch that controls system power. |
| 2 | Mode selection area | It is used to select the machining mode: **Reference Point**, **Jog**, **Auto**, **Handwheel** and **Step** mode;<br>Switching interface: program interface and track interface. |
| 3 | Feed override knob | It is used to adjust the feed speed (0~120%) and rapid feed speed (0~100%). |
| 4 | Spindle override knob | Used to adjust the spindle speed (50~150%). |
| 5 | E- stop switch | When the machine is in a dangerous state, it is used to stop the operation of the machine tool and protect the safety of the person and device.<br>After the danger is removed, the E-stop alarm can be released by rotating the button clockwise. |
| 6 | Origin display area | After each axis has the origin mark, the corresponding light will always be on. |
| 7 | Alarm display area | Including limit alarm, spindle alarm, etc., when relevant alarms enter, the corresponding lights will always be on. |
| 8 | Tool display area | Including tool magazine tool No., spindle tool No. display. |
| 9 | Program protection knob | Turn on and off the program protection function. |
| 10 | Motion control key | **Program Start** and **Program Stop** are used to control the machine movement. |

| No. | Name | Description |
|---|---|---|
| 11 | Machining control key | Including spindle control keys: **Spindle CW**, **Spindle Stop** and **Spindle CCW**;<br>Common operation keys: **Breakpoint Resume**, **Program Pause** and **Reset**, etc;<br>And some auxiliary function keys: **Lube** and **Dust Absorption**. |
| 12 | Axis direction key area | It is used to manually move each axis of the machine tool or jiggle function includes X, Y, Z, and extended axes.<br>In **Manual** mode:<br>• After pressing the **Rapid** key, press any axis direction key, and the machine tool will move at high speed manually.<br>• Only press any axis direction key, and the machine tool will move at a low speed manually. |
| 13 | Tool magazine key area | It is used to perform relevant operations on the tool magazine. The **Magazine CW** and **Magazine CCW** keys are only valid when the **Disc Magazine** is enabled. |
| 14 | Extended key area | There are 9 custom keys, which can be configured through NcConfig. |

#### 1.4.1.2 Dimension Drawing (unit: mm)

Dimension drawing of OP4425 metal plate trepanning




### 1.4.2 Key Panel

For the layout of WH201C key panel, see Key Panel .

## 1.5 NK300CX Connection Diagram

According to the type of control system, the NK300CX connection diagram can be divided into:

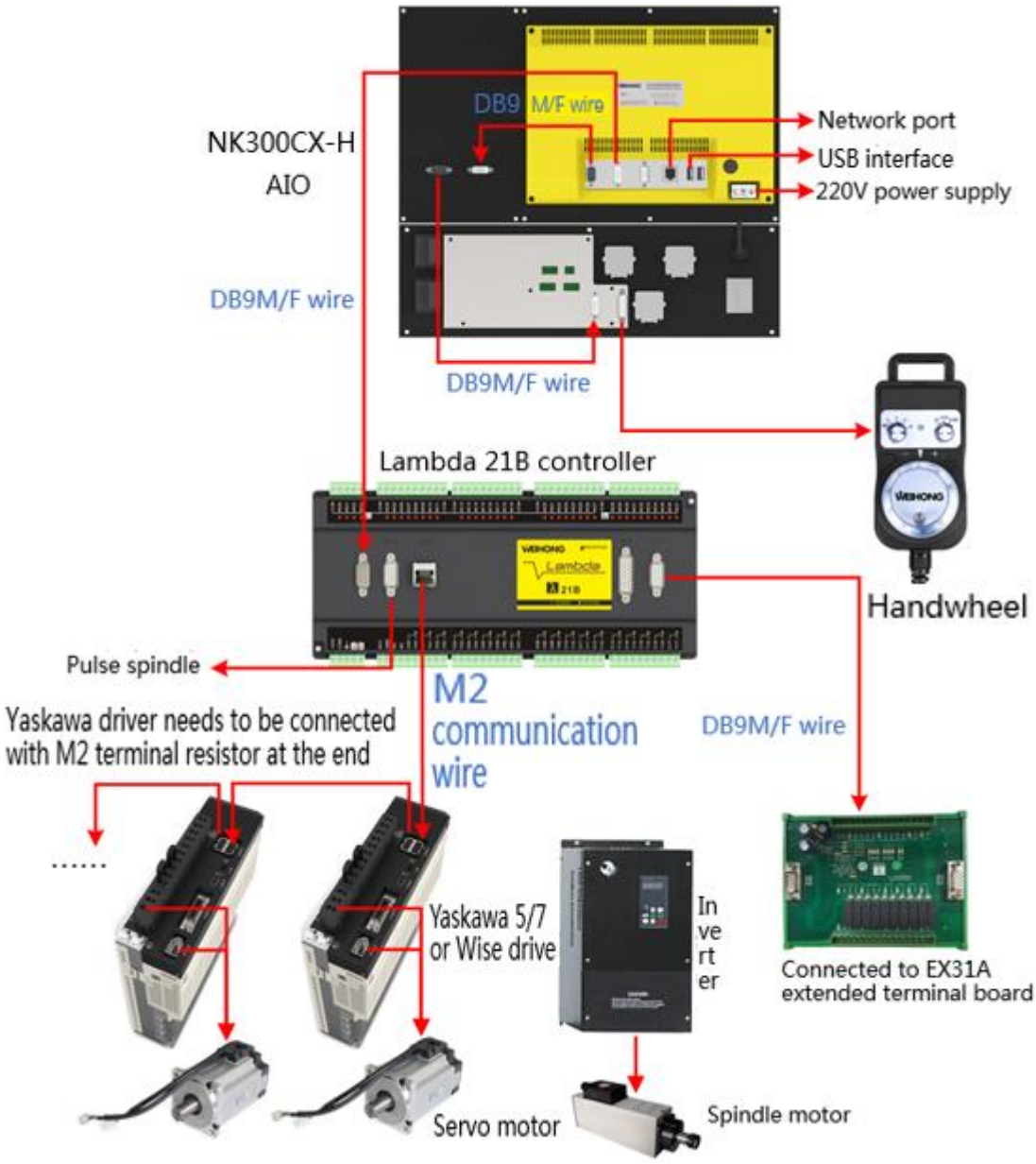
- Connection diagram of non-bus control system
- Connection diagram of bus control system

This article takes the connection diagram of common bus control system as an example.

- NK300CX-H Connection Diagram
- NK300CX-V Connection Diagram

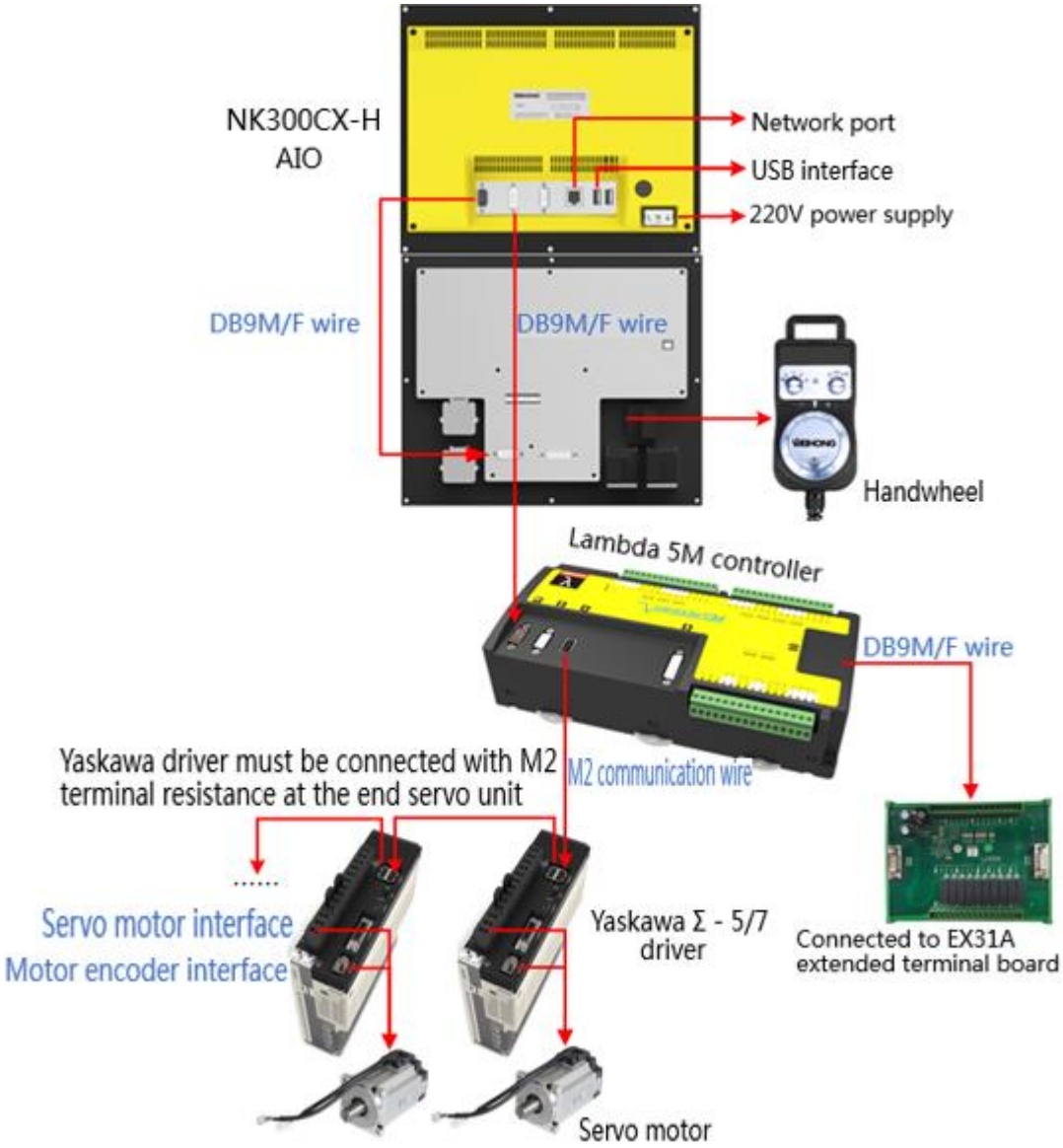
## 1.5.1 NK300CX-H Connection Diagram

The connection diagram of NK300CX-H bus control system is as follows:



## 1.5.2 **NK300CX-V Connection Diagram**

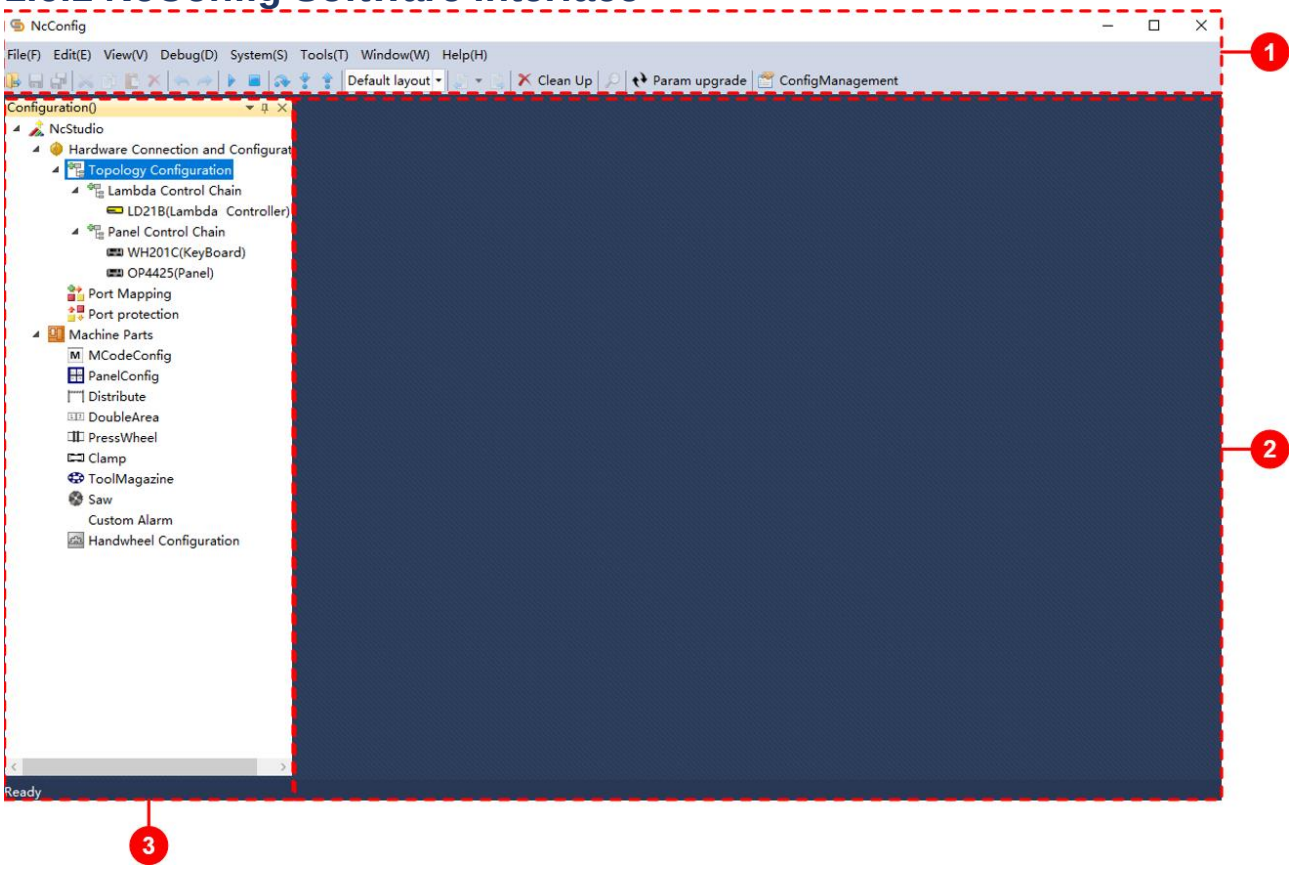
The connection diagram of NK300CX-V bus control system is as follows:



# 1.6 Software Main Interface

Now let's introduce the main interface of NcCongfig, NcTune and NcStudio software.

## 1.6.1 NcConfig Software Interface




1. Menu bar 2. Function display page 3. Configuration page

### 1.6.1.1 Function Display Page

After selecting the target function on the **Configuration Page**, the details will be displayed in the **Function Display Area**.

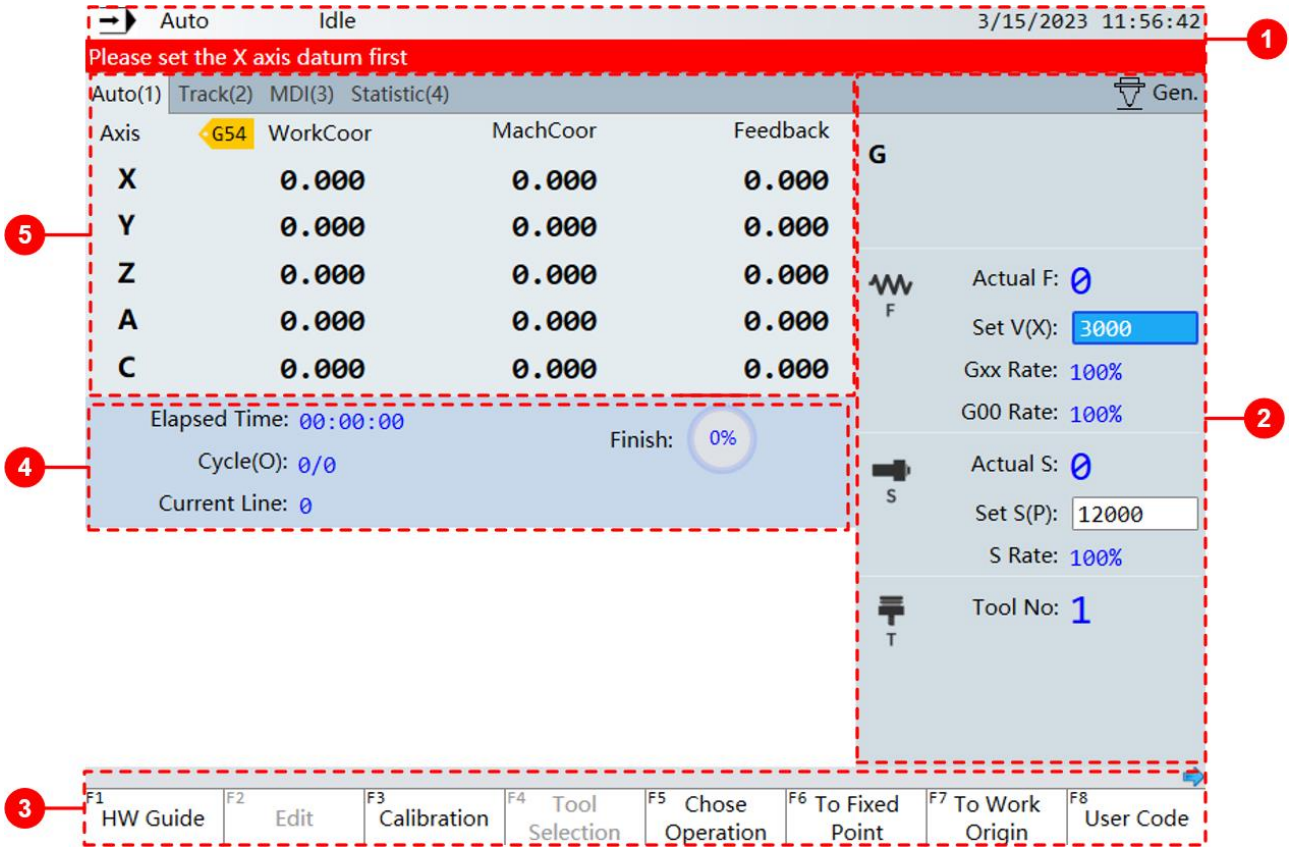
### 1.6.1.2 Configuration Page

Configuration page includes:

- Configure Device
- Setting Machine Tool Part
- Configure Port Mapping
- Set Port Protection

## 1.6.2 NcSudio Software Interface

The UI layout of the NK300CX integration software is as follows:



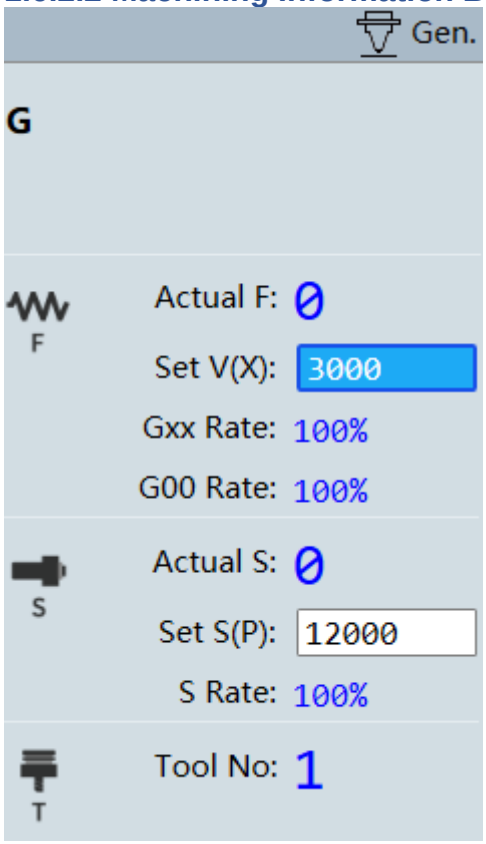

1. Status information bar 2. Machining information bar 3. Operation button bar 4. Machining process information area 5. Function window switching area

### 1.6.2.1 Status Information Bar


Display the following information:

- Current mode: auto, jog, reference point, etc.
- Current status of machine tool: idle, running, etc.
- Current date: MM/DD/YY.
- Running time of the software since this startup: hour/minute/second.
- Log information: display warnings, errors and other information during software operation.

### 1.6.2.2 Machining Information Bar




Display the processing information of the machine tool, including:

- Current page: machining, advanced, program, system, parameter, diagnosis log.
- Current speed: machining speed during machining, G00 speed during G00 movement.
- Set machining speed.
- Feed rate, G00 rate, spindle rate.
- Current spindle speed, set spindle speed:

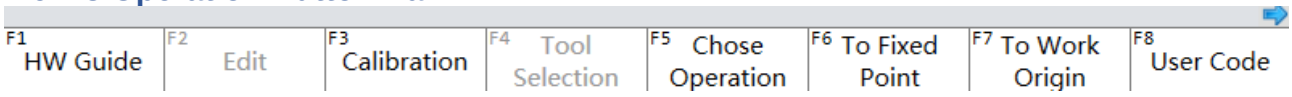
  Actual machining speed= Machining speed × Feed rate

  Actual G00 speed= G00 speed × G00 rate

  Actual spindle speed= Set speed × Spindle rate

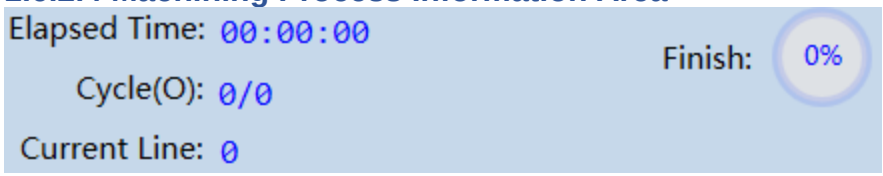
- The tool No. of the currently used tool.

### 1.6.2.3 Operation Button Bar




Display the meanings corresponding to the **F1** ~ **F8** operation buttons, and the meanings vary with the sub-function interface.
When there are more than eight operation buttons, a blue page turning arrow appears in the upper right corner.

### 1.6.2.4 Machining Process Information Area



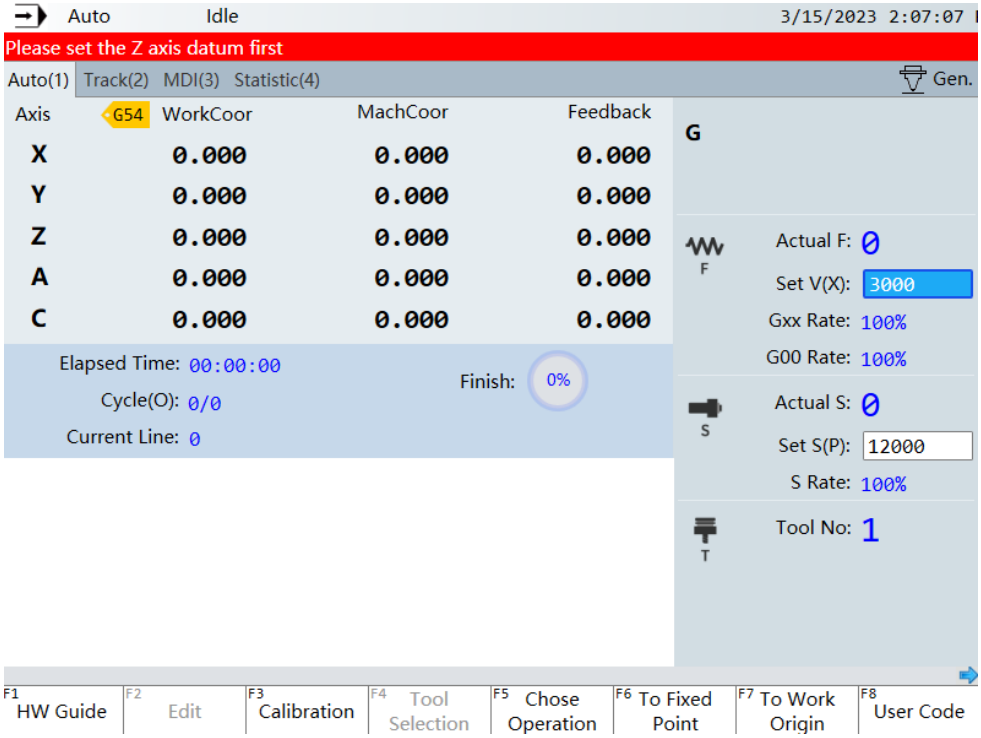
Displays information about the machining process, including:

- Elapsed time
- Cycle
- Current line
- Finish

### 1.6.2.5 Function Page Switching Area

Press 1~4 on the panel to switch to display the corresponding function page information.

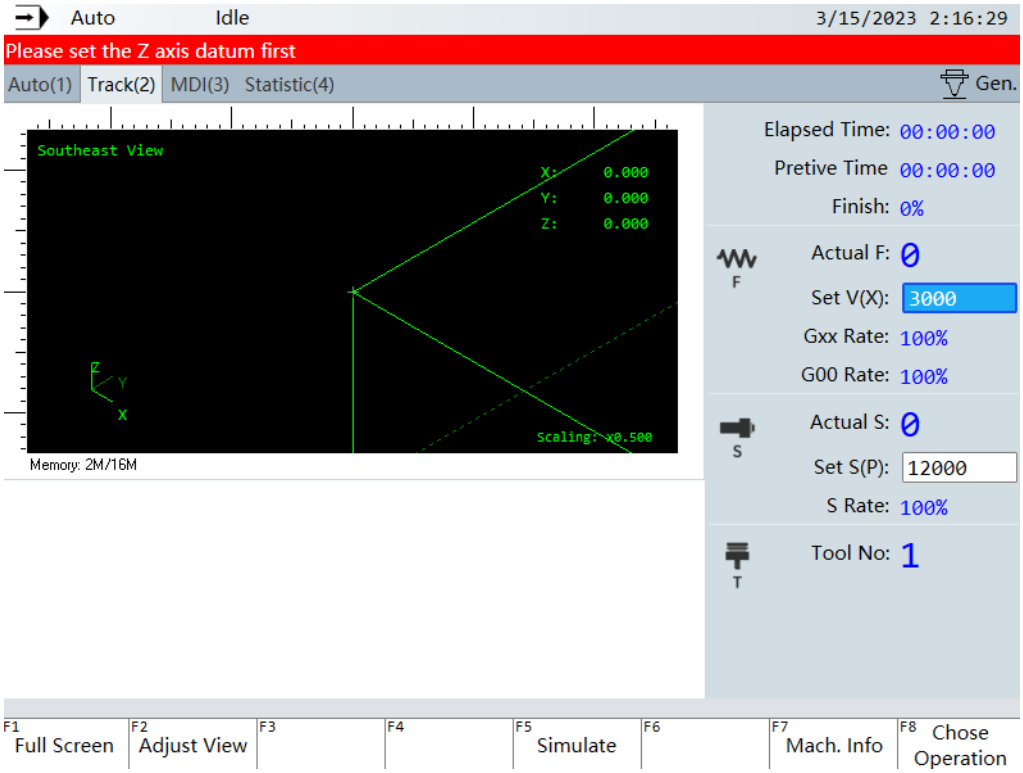
- Coordinate page




Display the workpiece, mechanical and feedback coordinates of the axis.

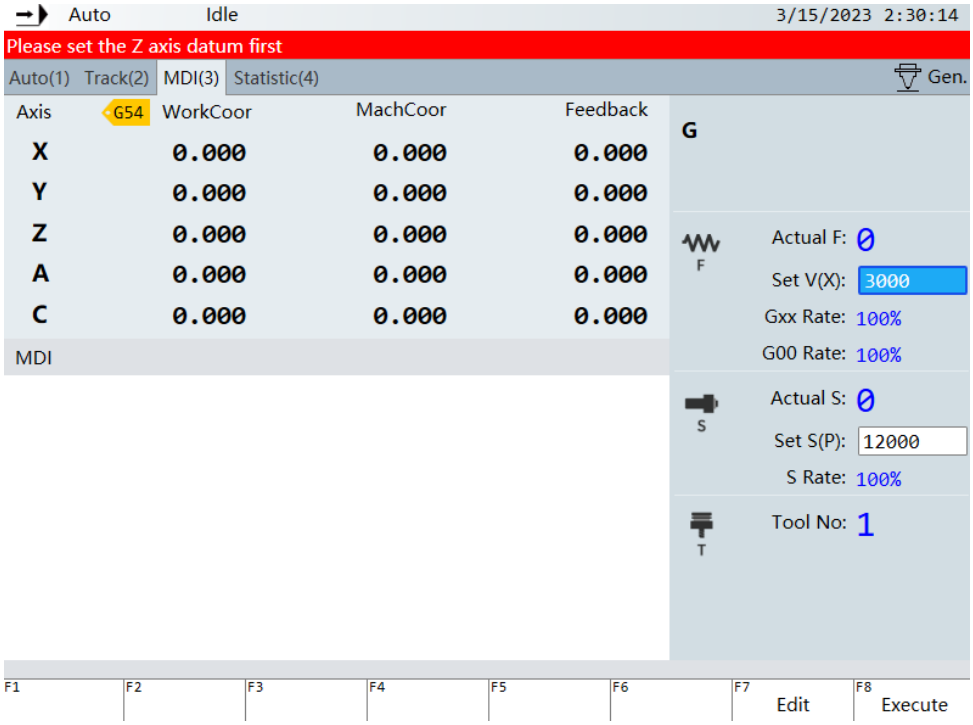
 sign appears in front of the corresponding axis after returning to the mechanical origin.

- Track information page




Display the machining track.

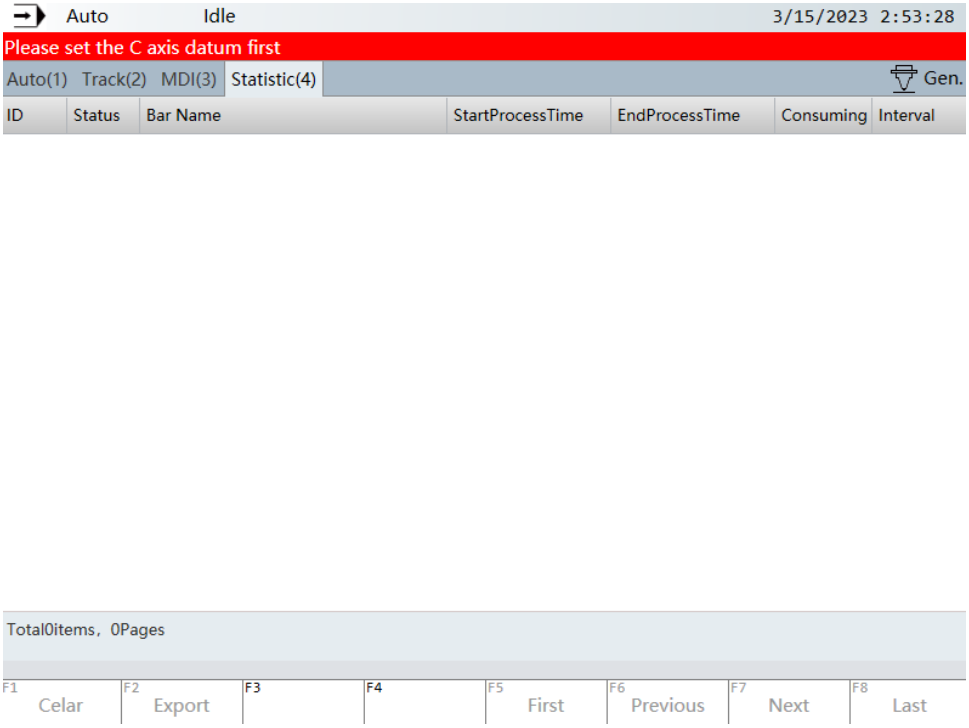
- MDI page




Custom command page, including the following functions:

- Edit: add or delete custom commands.
- Execute: Execute the selected command.

- Statistic page



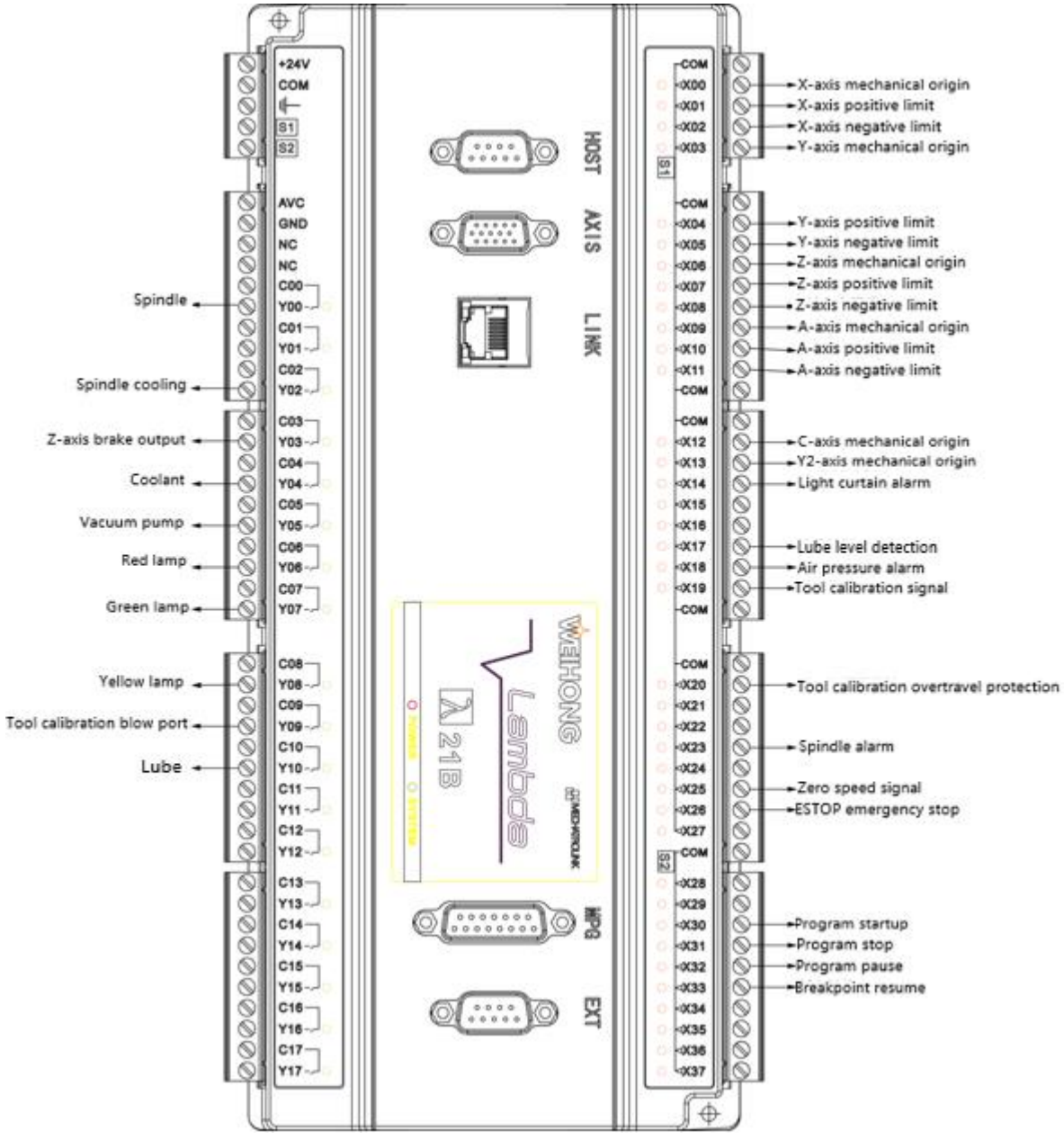

Show machining history.

# 2 Wiring

Through this section, you can quickly understand the signal types and port definitions supported by the NK300CX integrated CNC system, which is helpful for wiring.

## 2.1 Port Definition

Lambda 21B port and wiring diagram are as follows:




**Note**

The figure shows the default port definition of the software, which can be configured by configuring port mapping.

## 2.2 Signal Type

Support the following signal types:

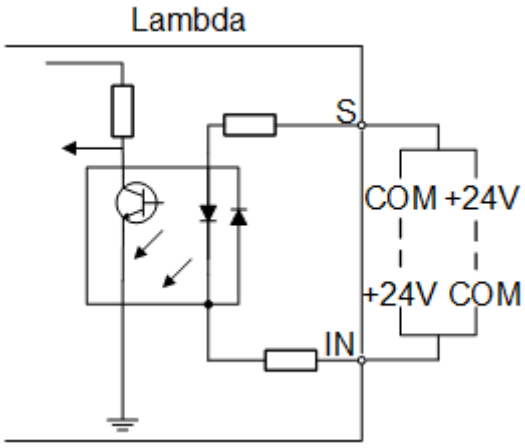
- Switch Value Input Signal
- Relay Output Signal
- Analog Output Signal
- Command Pulse Signal

### 2.2.1 Switch Value Input Signal

Switch input signal supports high and low level valid:

- Connecting to the COM port in NO mode means to receive single.
- Unconnecting with the COM port in NC mode means to receive single.

The diagram is as follows:




Input port supports high/low level valid:

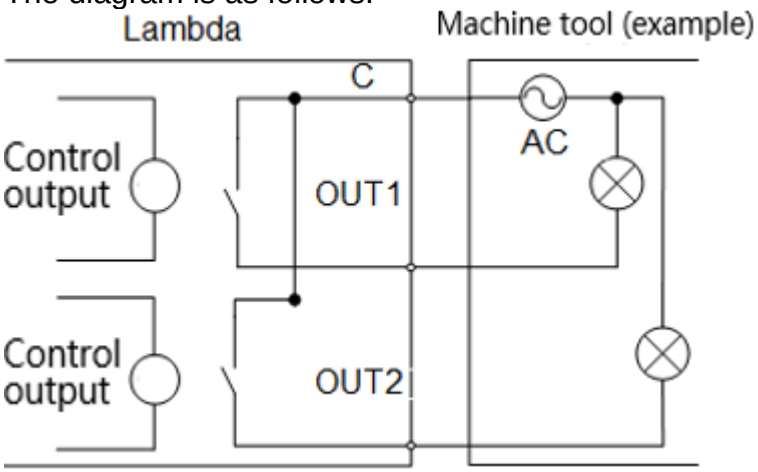
- When the common port S of Lambda controller is connected to COM, the input port is valid at high level.
- When the common port S of Lambda controller is connected to+24V, the input port is valid at low level.

### 2.2.2 Relay Output Signal

The output type of Lambda controller is relay output.

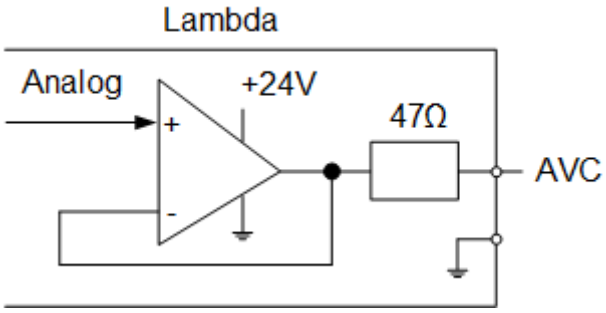
Load capacity of relay contacts: AC 7A/250V, DC 7A/30V. If connected to high-power load, can connection contactor.

The diagram is as follows:



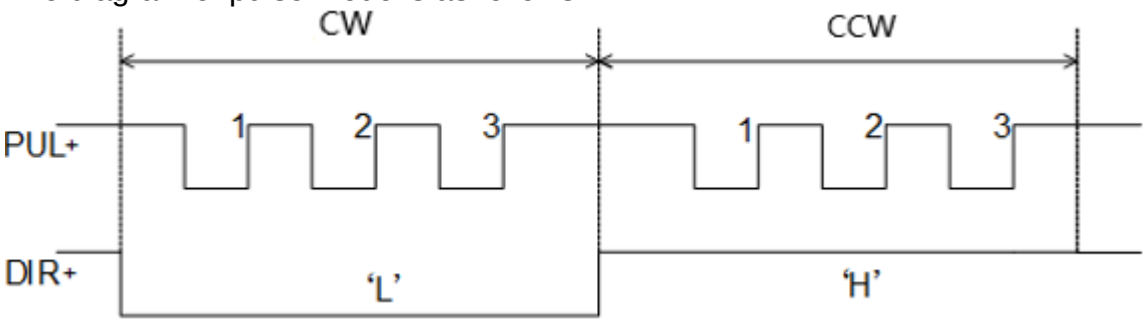
## 2.2.3 Analog Output Signal

AVC is a controlled voltage output of 0V~10V, which is connected to the analog voltage frequency command input port of the external inverter. The frequency of the inverter is controlled by changing the voltage to control the spindle speed.
The diagram is as follows:

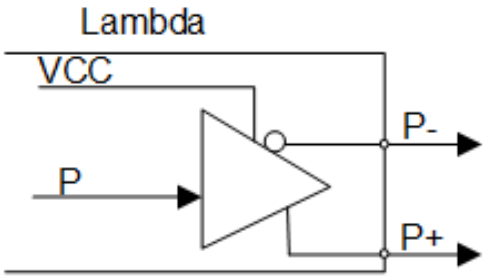



## 2.2.4 Command Pulse Signal

The pulse command that controls the drive movement is pulse+direction, negative logic. Maximum pulse frequency 1MHz.
The diagram of pulse mode is as follows:

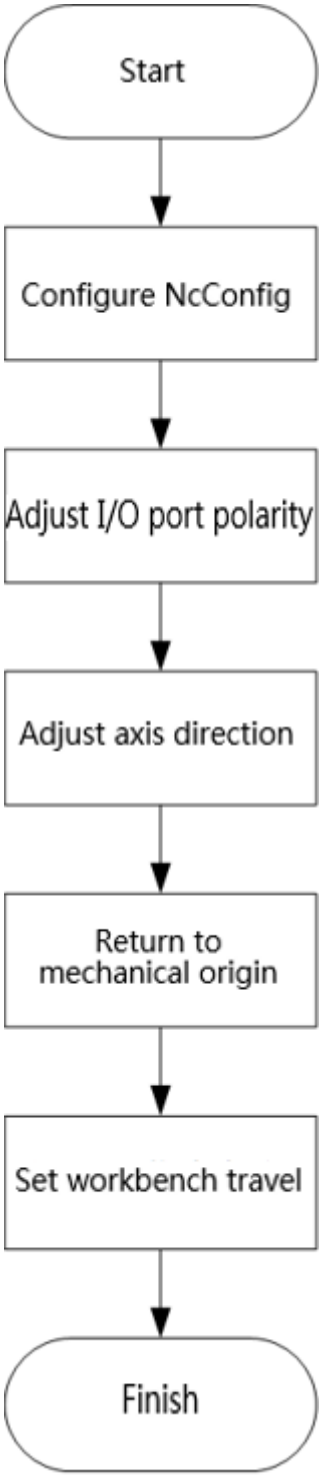



The diagram of pulse command output is as follows:



# 3 Machine Tool Debug

Through this section, you can quickly understand how to debug the NK300CX-Phoenix integrated CNC system.
The basic debug process is as follows:



# 3.1 NcConfig Configuration

Configure Lambda controller and extended terminal board, set machine tool part, port mapping, and port protection to improve the efficiency of project development.

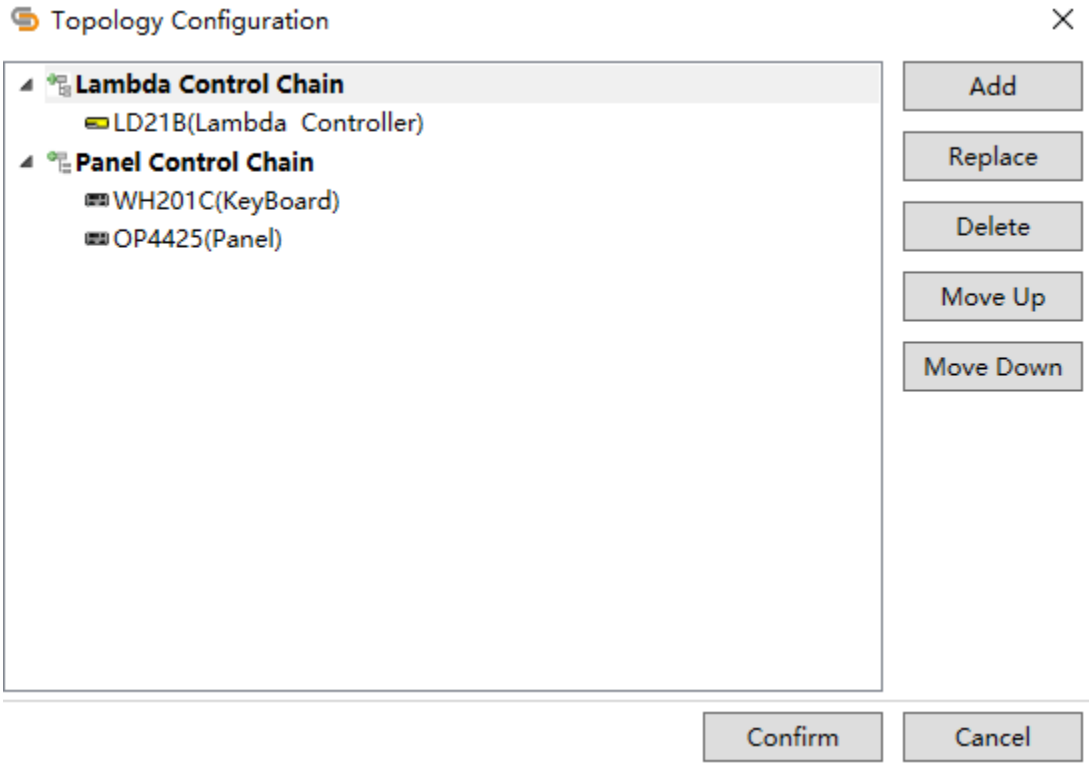
1. Select one of the following methods to open NcConfig:
   - In the following installation path, double-click **NcConfig.exe**.
     **C:\Program Files\Weihong\NcStudio\NcConfig\Bin**
   - In the start menu, Weihong folder, double-click **NcConfig.exe**.
2. According to the actual needs, perform the following operations:
   - Configure Device
   - Setting Machine Tool Part
   - Configure Port Mapping
   - Set Port Protection

## 3.1.1 Configure Device

Self-configurable operation panel, key panel, Lambda controller and extended terminal board.

**Operation Steps:**

1. (**Optional**): In the menu bar, click **View → Configuration**, and the **Configuration** is displayed on the left.
2. Double click the **Topology Configuration** in the left **Configuration** page to open the **Topology Configuration** dialog box:




3. Configure **Panel Control Chain** according to panel type:
   - Horizontal panel: select WH106C and WH201C
   - Vertical panel: select WH108C
   - Woodworking special type: WH201C and OP4425
4. Configure **Lambda Control Chain** according to the actual situation:

- If need to add a controller/ extended terminal board, click the **Lambda Control Chain**, click **Add**, and double-click to select the target controller/ extended terminal board in the displayed list.
- If need to replace the controller/ extended terminal board, click the target to be replaced, click **Replace**, and double-click to select the target controller/ extended terminal board in the displayed list.
- If need to delete the controller/ extended terminal board, click the target to be deleted, and then click **Delete.**
- If need to move the controller/ extended terminal board, click the target to be moved, and then click **Move Up/Down**.

5. Click **Confirm**.

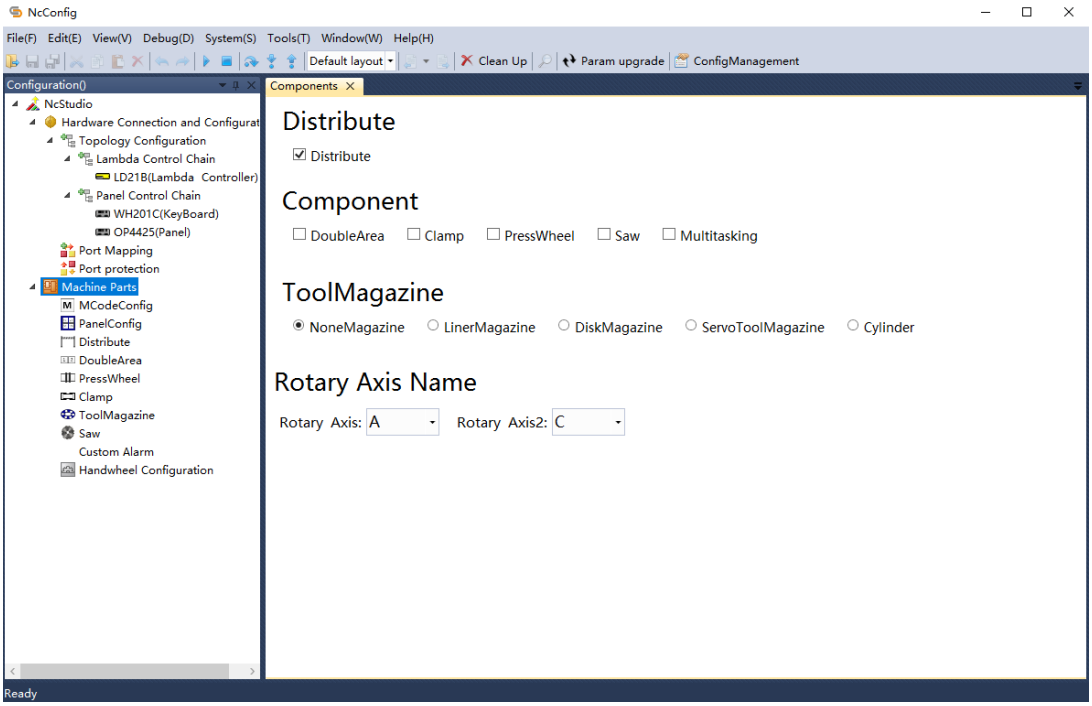

- The difference between bus and non-bus is that through configuration Lambda auto switch without any other operation.
- If Lambda chooses non-bus Lambda, such as 21A, 20A, etc., it can only support 5 axes at most, and the software cannot be started if it exceeds the number; If Lambda chooses bus, there is no limit on the number of axles.

## 3.1.2 Setting Machine Tool Part

Select and set the actual used machine tool part in NcConfig.
It can set M code config, panel config, distribute, double area, press wheel, clamp, tool magazine, saw and handwheel configuration.

**Operation Steps:**

1. Double click the **Machine Parts** in the left **Configuration** page to open the **Components** page：




2. Select rotary axis.
3. Check the required components, and the corresponding functions will appear on the corresponding software.
4. According to the actual needs, perform the following operations:
   - Configure M Command
   - Configure Panel
   - Configure Distribute
   - Configure Double Area
   - Configure Press Wheel
   - Configure Clamp
   - Set Tool Magazine
   - Set Saw
   - Configure Handwheel

### 3.1.2.1 Configure M Command

Set the M command configuration, modify the function command, and control the machine tool action and port status.

After selecting different machine parts and saving, the M command configuration window can display more configurable M codes.

**Note**

Configure the corresponding M code according to the currently selected machine part. Please do not configure it at will.

---

M command include:

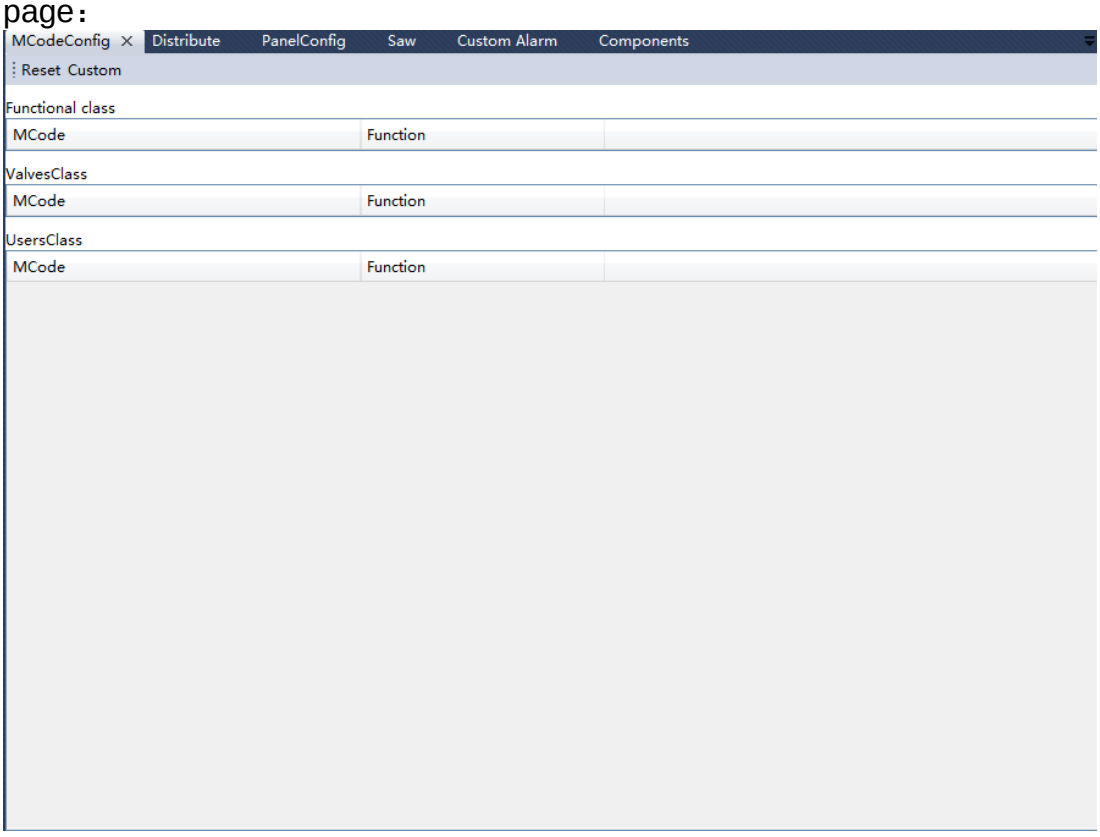
- Functional class: after the command is executed, the machine tool generates an action.
- Valves class: control port status.
- Users class: customize the added subprogram.

**Prerequisite:**

The components related to the M command to be configured have been checked in the **Components** page.

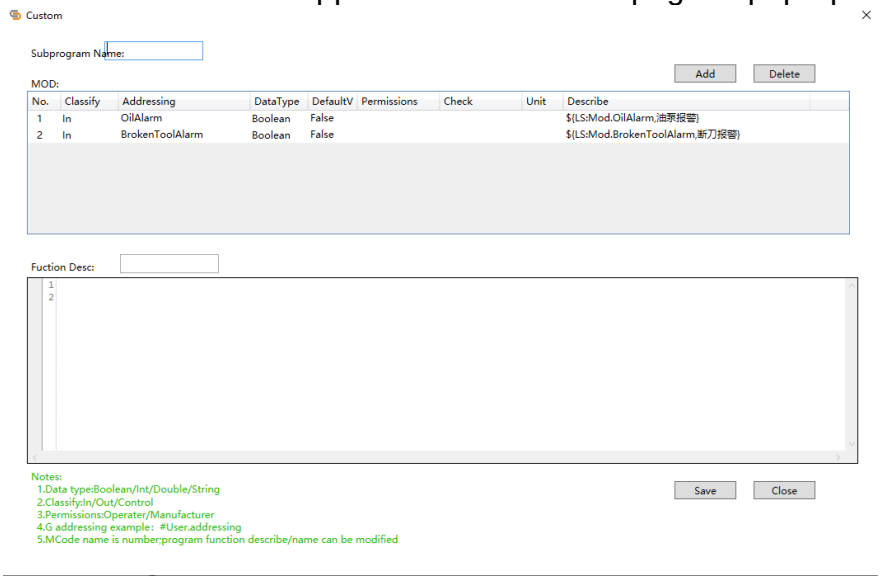
**Operation Steps:**

1. Click the **M Code Config** on the left **Configuration** to enter the **M Code Config** page：




2. (**Optional**): Add custom subprogram.

a. Click **Custom** in the upper left corner of the page to pop-up the **Custom** window:




b. According to the actual situation, enter the following information and add custom subprograms:
   - Subprogram name: subprogram function name.
     Usually English.
   - Subprogram function description.
     The default is the same as the subprogram name, which can be manually changed to the corresponding Chinese or other content.
   - Specific function: add in the box.

c. Click **Save**.

**Note**

The added subprogram also appears in the configurable functions of the panel.

3. Select the target function, double-click the corresponding **MCode** column cell, and modify the function command code.

### 3.1.2.2 Configure Panel

Through the configuration panel, you can configure the customized function keys (K1~K9) on the NK300CX panel.

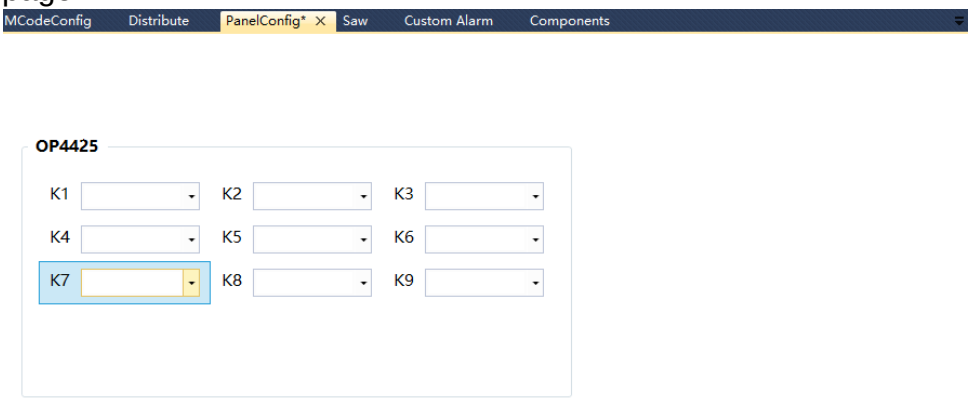
Configurable function buttons include: tool retraction, area 1 start, area 2 start, clamp page, zone 1 clamp, zone 2 clamp, press wheel switch, saw blade T31, saw blade T32, saw blade start and saw blade cylinder.

**Prerequisite:**

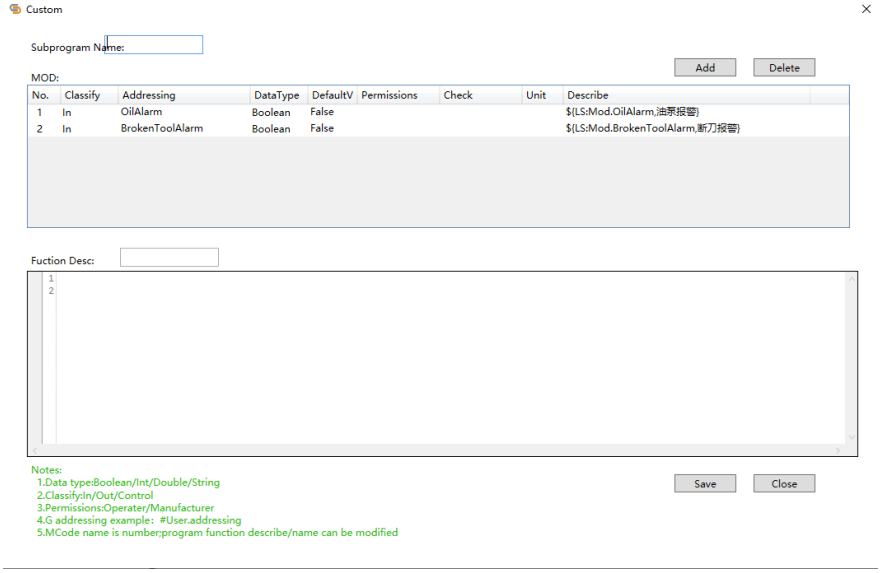
The components related to the function button to be configured have been checked in the **Components** page.

**Operation Steps:**

1. Double click **Panel Config** in the left **Configuration** to enter the **Panel Config** page：




2. (**Optional**): Customize the subprogram function and configure the panel button function that meets customer needs.
   a. Click the **M Code Config** on the left **Configuration** to enter the **M Code Config** page.
   b. Click **Custom** in the upper left corner of the page to pop-up the **Custom** window:




   d. According to the actual situation, enter the following information and add custom subprograms:
      - Subprogram name: subprogram function name. Usually English.
      - Subprogram function description.

The default is the same as the subprogram name, which can be manually changed to the corresponding Chinese or other content.

- Specific function: add in the box.

c. Click **Save**.

3. At the target position, click the drop-down box, select the target function, and configure the function for the corresponding custom key.
4. (**Optional**): Click the blank position of the first line in the target drop-down box to cancel the function configuration of the corresponding custom key.

### 3.1.2.3 Configure Distribute

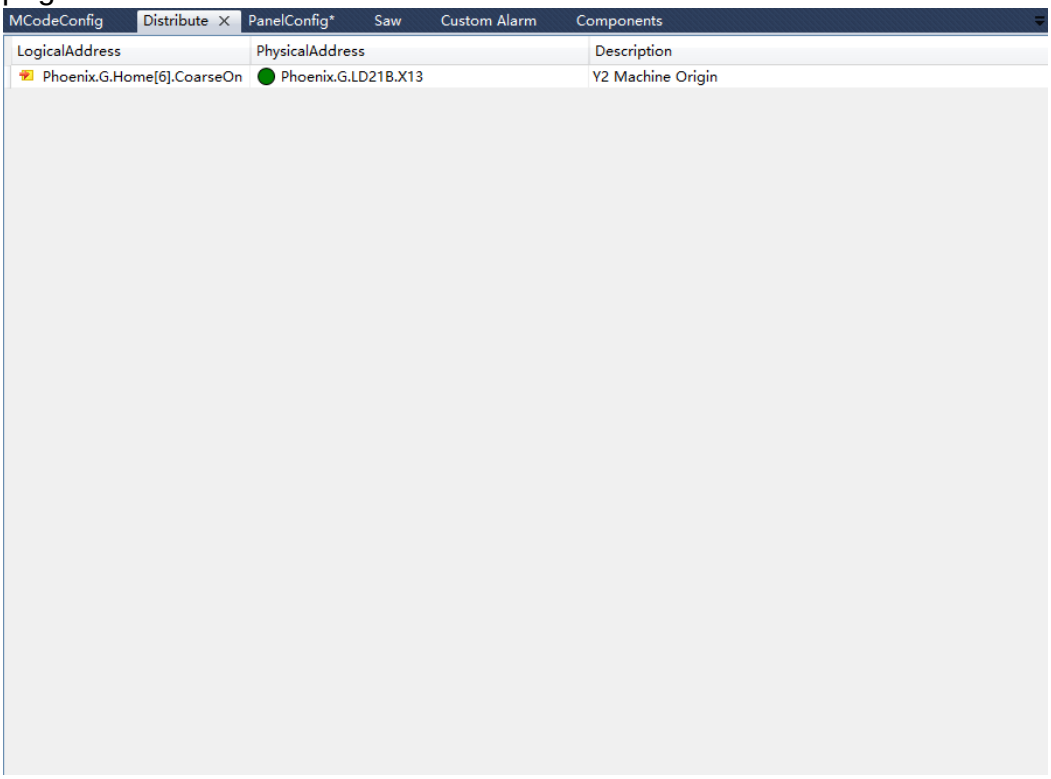
Set the distribute according to the actual structure of the machine tool.

**Prerequisite:**

**Distribute** has been checked on the **Components** page.

**Operation Steps:**

1. Double click the **Distribute** in the left **Configuration** page to enter the **Distribute** page：




2. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.

**Note**

If the Lunda control chain is configured as non-bus, auto changes the logic of double Y return to the origin according to the parameter **Enable Auto Calibration of Distribute Axis**: if the parameter is no, it can directly return to the origin and only detect one origin signal; If the parameter is yes, it is necessary to perform double Y origin detection before returning to the mechanical origin and detect two origin signals.

### 3.1.2.4 Configure Double Area

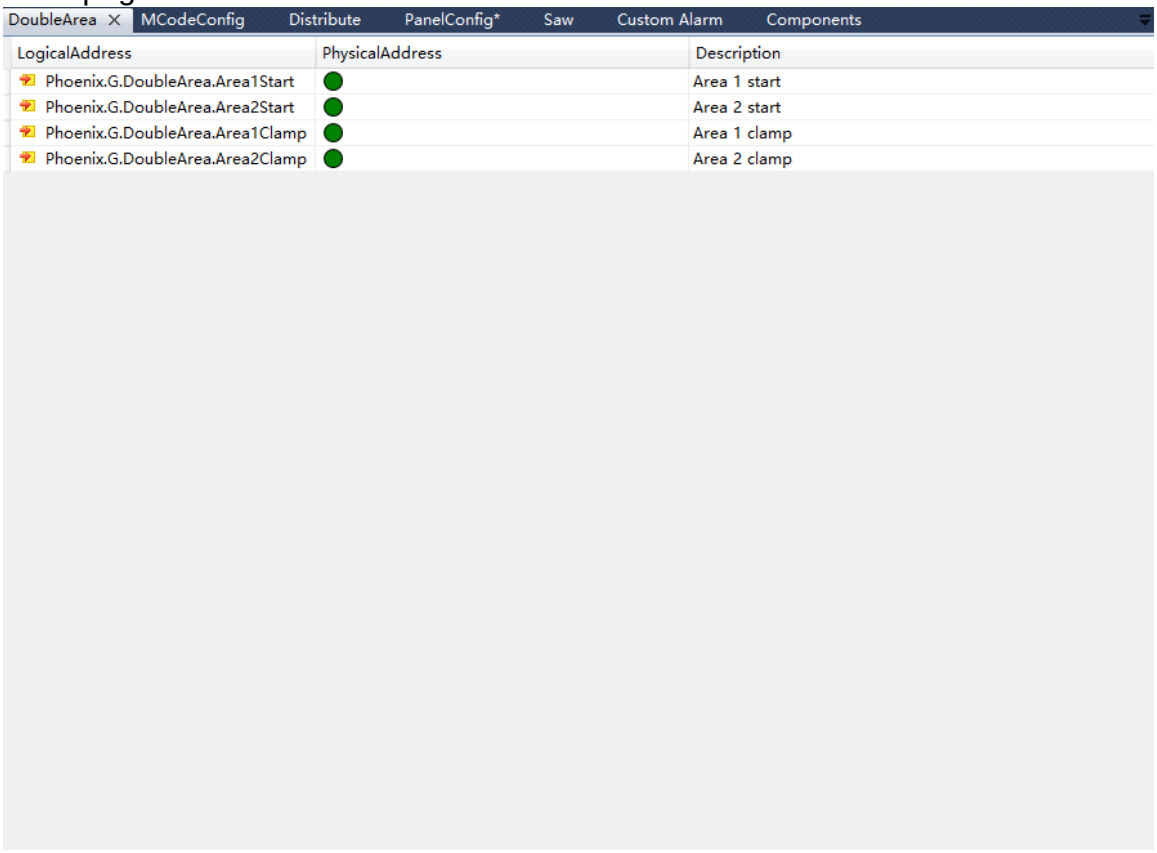
Switch from single area to double area. After the completion of machining, area 1 can directly process area 2, saving the time of loading and unloading, and improving the work efficiency.

**Prerequisite:**

**Double Area** has been checked on the **Components** page.

**Operation Steps:**

1. Double click the **Double Area** in the left **Configuration** page to enter the **Double Area** page：




2. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.

### 3.1.2.5 Configure Press Wheel

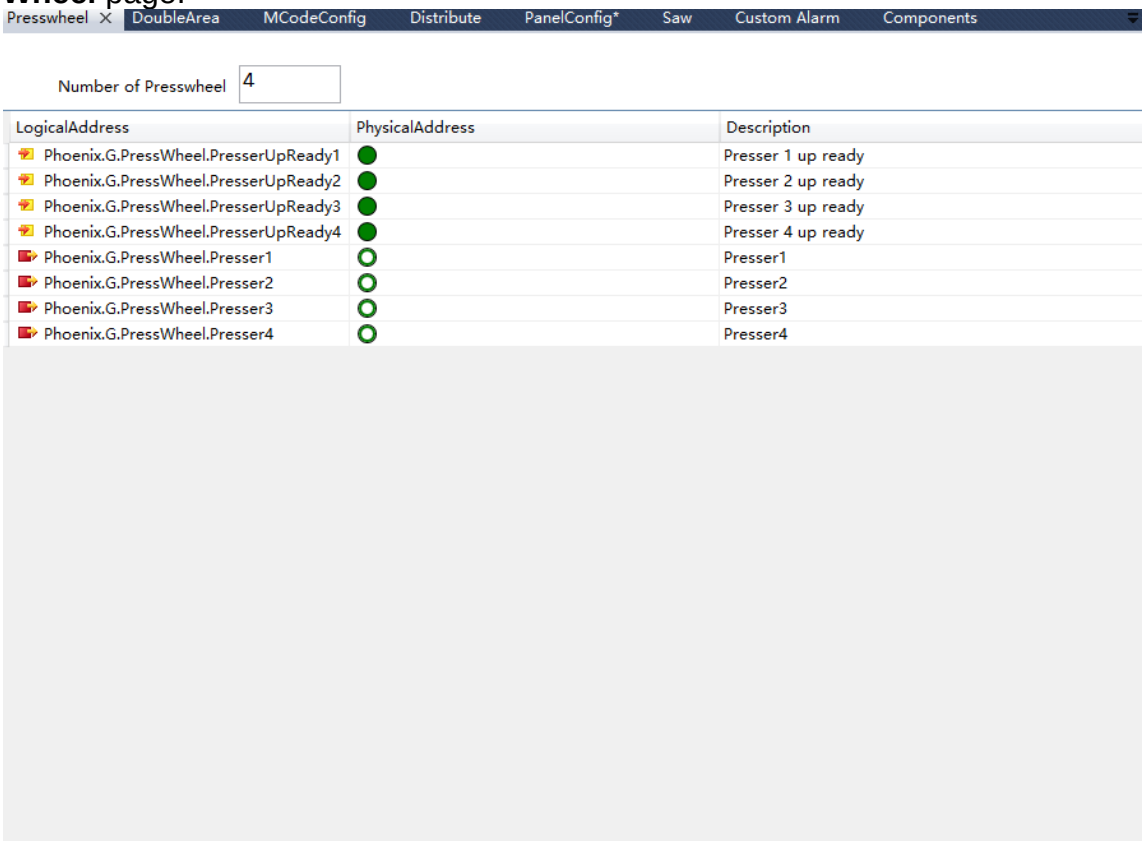
Configure press wheel can prevent the plate from displacement during machining.

**Prerequisite:**

**Press Wheel** has been checked on the **Components** page.

**Operation Steps:**

1. Double click the **Press Wheel** in the left **Configuration** page to enter the **Press Wheel** page:




2. Set the parameter **Number of Press Wheel**.
3. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.

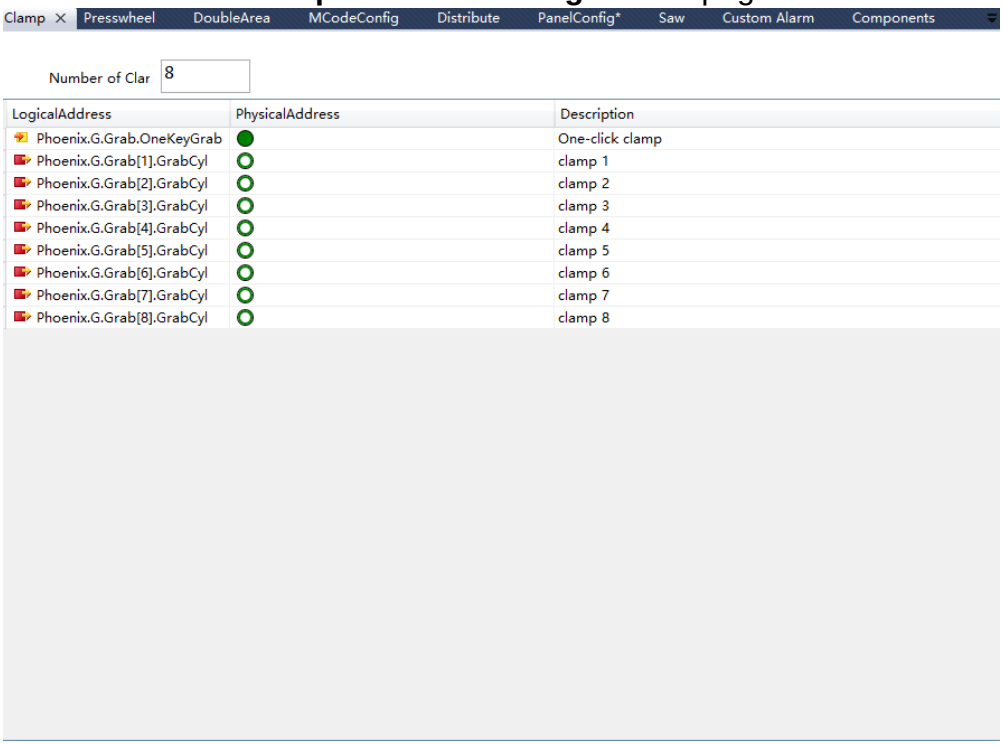
### 3.1.2.6 Configure Clamp

**Prerequisite:**

**Clamp** has been checked on the **Components** page.

**Operation Steps:**

1. Double click the **Clamp** in the left **Configuration** page to enter the **Clamp** page：




2. Set the parameter **Number of Clamp**.
3. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.

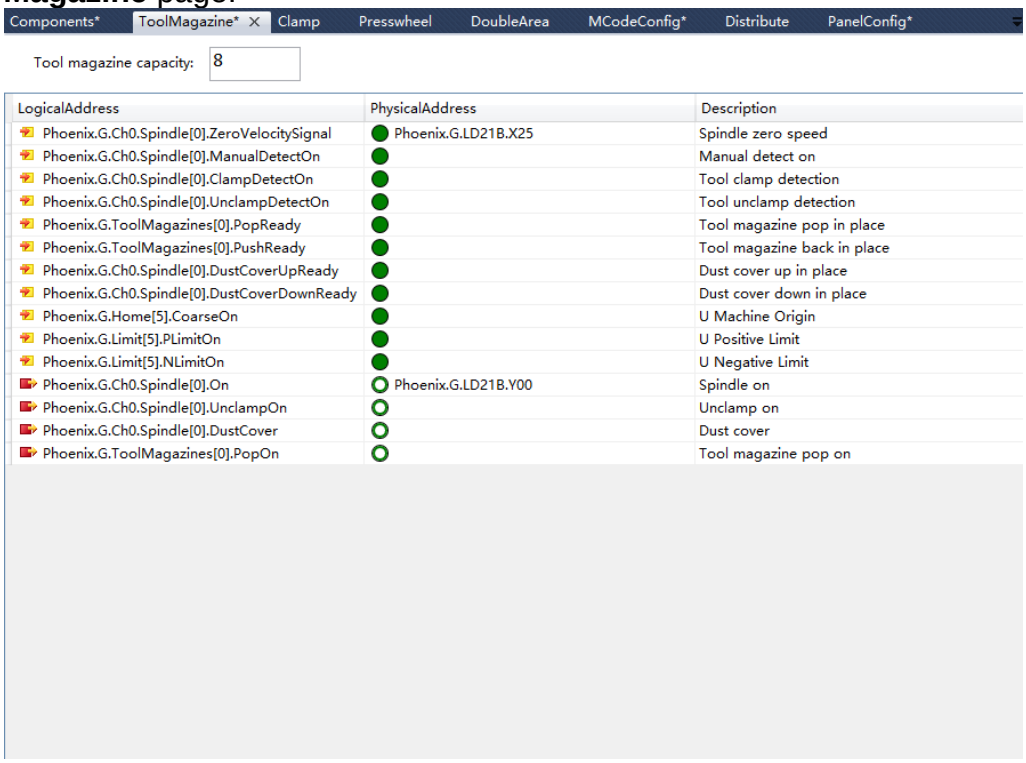
### 3.1.2.7 Set Tool Magazine

Support multiple mechanical structures, provide multi-cylinder single-frequency conversion and multi-cylinder multi-frequency conversion function combination customization and select tool magazine.

**Operation Steps:**

1. (**Optional**): Double click the **Machine Parts** in the left **Configuration** page to open the **Components** page.
2. In the **Tool Magazine** area, select the **Tool Magazine** type:
   - None magazine
   - Liner magazine
   - Disk magazine
   - Servo tool magazine
     Set servo axis index: the axis number of the servo axis.
   - Cylinder

3. Double click the **Tool Magazine** in the left **Configuration** page to enter the **Tool Magazine** page:




**Note**

Select different tool magazine types, and the tool magazine setting page may be different.

4. Set specific parameters according to the page display.

| Parameter | Description |
|---|---|
| Tool magazine capacity | The number of tools that the magazine can hold. |
| Count tool overtime | Maximum time to wait for tool counting signal. |
| Cylinder count | Number of cylinders. |
| Cylinder type | Multi-cylinder single-fre: one inverter controls ON/OFF of multiple spindles.<br>Multi-cylinder multi-fre: one Z-axis with two to four cylinders, and multiple inverters control the ON/OFF of multiple spindles.<br>Four-cylinder double-fre: one Z-axis with four cylinders, and two inverters control the ON/OFF of multiple spindles. |

5. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.

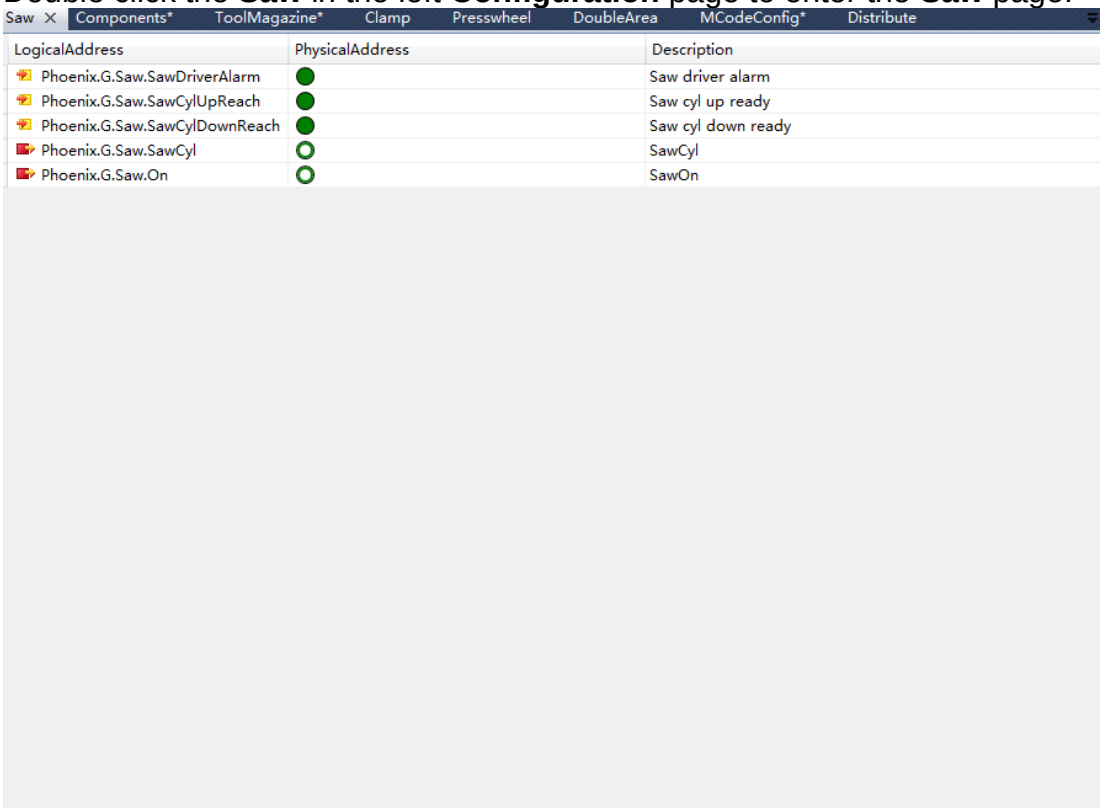
### 3.1.2.8 Set Saw

**Prerequisite:**

**Saw** has been checked in the **Components** page.

**Operation Steps:**

1. Double click the **Saw** in the left **Configuration** page to enter the **Saw** page:



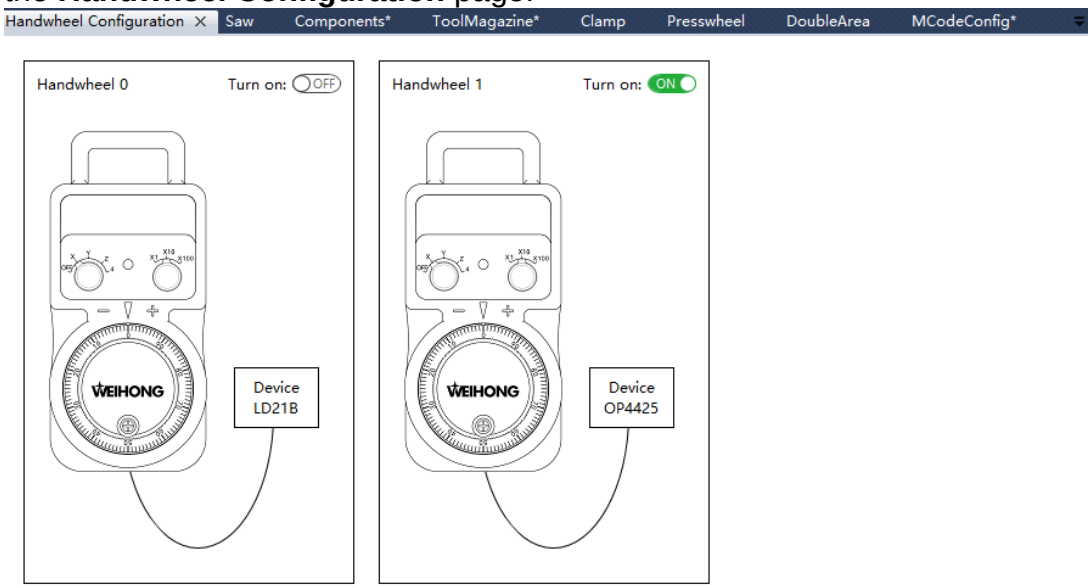

2. Set the parameter **Saw Blade Axis Index** and select the target saw blade axis.
3. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.

### 3.1.2.9 Configure Handwheel

The handwheel controls the machine tool movement or enables the handwheel guide function.

**Operation Steps:**

1. Double click the **Handwheel Configuration** in the left **Configuration** page to enter the **Handwheel Configuration** page:



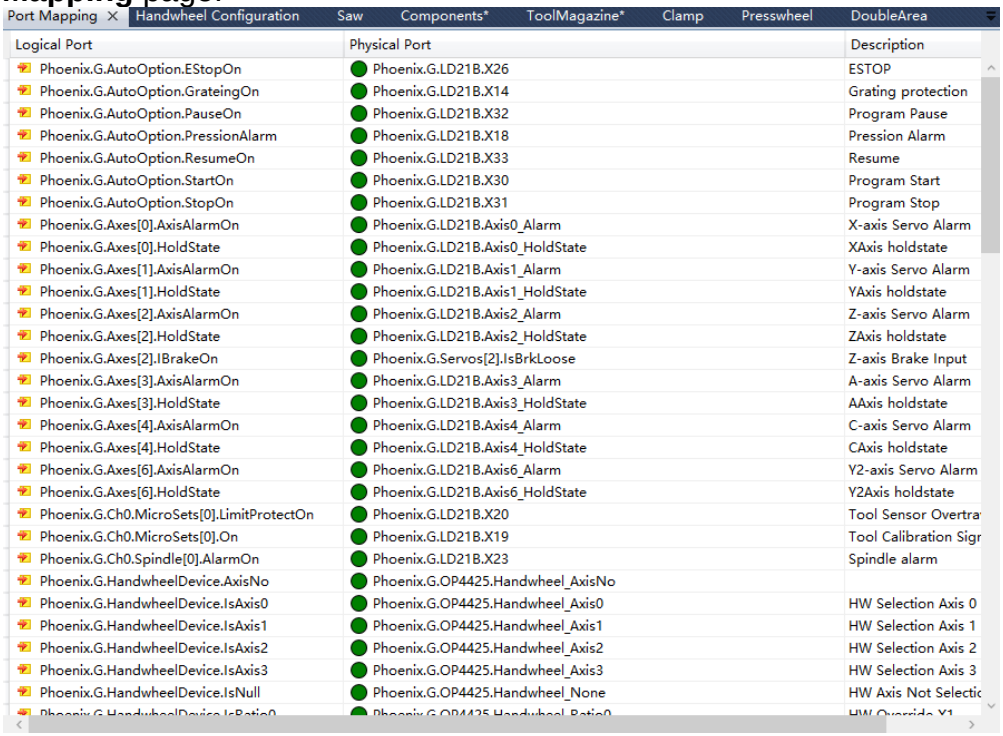

2. Set the **Turn On** to **ON**.

## 3.1.3 Configure Port Mapping

Modify the physical address of the port to match the logical address.

**Operation Steps:**

1. Double click the **Port Mapping** in the left **Configuration** page to enter the **Port Mapping** page:




| Logical Port | Physical Port | Description |
|---|---|---|
| Phoenix.G.AutoOption.EStopOn | Phoenix.G.LD21B.X26 | ESTOP |
| Phoenix.G.AutoOption.GrateingOn | Phoenix.G.LD21B.X14 | Grating protection |
| Phoenix.G.AutoOption.PauseOn | Phoenix.G.LD21B.X32 | Program Pause |
| Phoenix.G.AutoOption.PressionAlarm | Phoenix.G.LD21B.X18 | Pression Alarm |
| Phoenix.G.AutoOption.ResumeOn | Phoenix.G.LD21B.X33 | Resume |
| Phoenix.G.AutoOption.StartOn | Phoenix.G.LD21B.X30 | Program Start |
| Phoenix.G.AutoOption.StopOn | Phoenix.G.LD21B.X31 | Program Stop |
| Phoenix.G.Axes[0].AxisAlarmOn | Phoenix.G.LD21B.Axis0_Alarm | X-axis Servo Alarm |
| Phoenix.G.Axes[0].HoldState | Phoenix.G.LD21B.Axis0_HoldState | XAxis holdstate |
| Phoenix.G.Axes[1].AxisAlarmOn | Phoenix.G.LD21B.Axis1_Alarm | Y-axis Servo Alarm |
| Phoenix.G.Axes[1].HoldState | Phoenix.G.LD21B.Axis1_HoldState | YAxis holdstate |
| Phoenix.G.Axes[2].AxisAlarmOn | Phoenix.G.LD21B.Axis2_Alarm | Z-axis Servo Alarm |
| Phoenix.G.Axes[2].HoldState | Phoenix.G.LD21B.Axis2_HoldState | ZAxis holdstate |
| Phoenix.G.Axes[2].IBrakeOn | Phoenix.G.Servos[2].IsBrkLoose | Z-axis Brake Input |
| Phoenix.G.Axes[3].AxisAlarmOn | Phoenix.G.LD21B.Axis3_Alarm | A-axis Servo Alarm |
| Phoenix.G.Axes[3].HoldState | Phoenix.G.LD21B.Axis3_HoldState | AAxis holdstate |
| Phoenix.G.Axes[4].AxisAlarmOn | Phoenix.G.LD21B.Axis4_Alarm | C-axis Servo Alarm |
| Phoenix.G.Axes[4].HoldState | Phoenix.G.LD21B.Axis4_HoldState | CAxis holdstate |
| Phoenix.G.Axes[6].AxisAlarmOn | Phoenix.G.LD21B.Axis6_Alarm | Y2-axis Servo Alarm |
| Phoenix.G.Axes[6].HoldState | Phoenix.G.LD21B.Axis6_HoldState | Y2Axis holdstate |
| Phoenix.G.Ch0.MicroSets[0].LimitProtectOn | Phoenix.G.LD21B.X20 | Tool Sensor Overtra |
| Phoenix.G.Ch0.MicroSets[0].On | Phoenix.G.LD21B.X19 | Tool Calibration Sigr |
| Phoenix.G.Ch0.Spindle[0].AlarmOn | Phoenix.G.LD21B.X23 | Spindle alarm |
| Phoenix.G.HandwheelDevice.AxisNo | Phoenix.G.OP4425.Handwheel_AxisNo | |
| Phoenix.G.HandwheelDevice.IsAxis0 | Phoenix.G.OP4425.Handwheel_Axis0 | HW Selection Axis 0 |
| Phoenix.G.HandwheelDevice.IsAxis1 | Phoenix.G.OP4425.Handwheel_Axis1 | HW Selection Axis 1 |
| Phoenix.G.HandwheelDevice.IsAxis2 | Phoenix.G.OP4425.Handwheel_Axis2 | HW Selection Axis 2 |
| Phoenix.G.HandwheelDevice.IsAxis3 | Phoenix.G.OP4425.Handwheel_Axis3 | HW Selection Axis 3 |
| Phoenix.G.HandwheelDevice.IsNull | Phoenix.G.OP4425.Handwheel_None | HW Axis Not Selectic |

2. Double click the **Physical Address** cell of the target, select the controller and port in the pop-up drop-down box, and specify the physical address for the corresponding logical address.
3. After setting, click  to save the current page or click  to save all pages. If it is not saved after editing, the * mark will appear in the upper right corner of the corresponding page button, and it will disappear after saving.

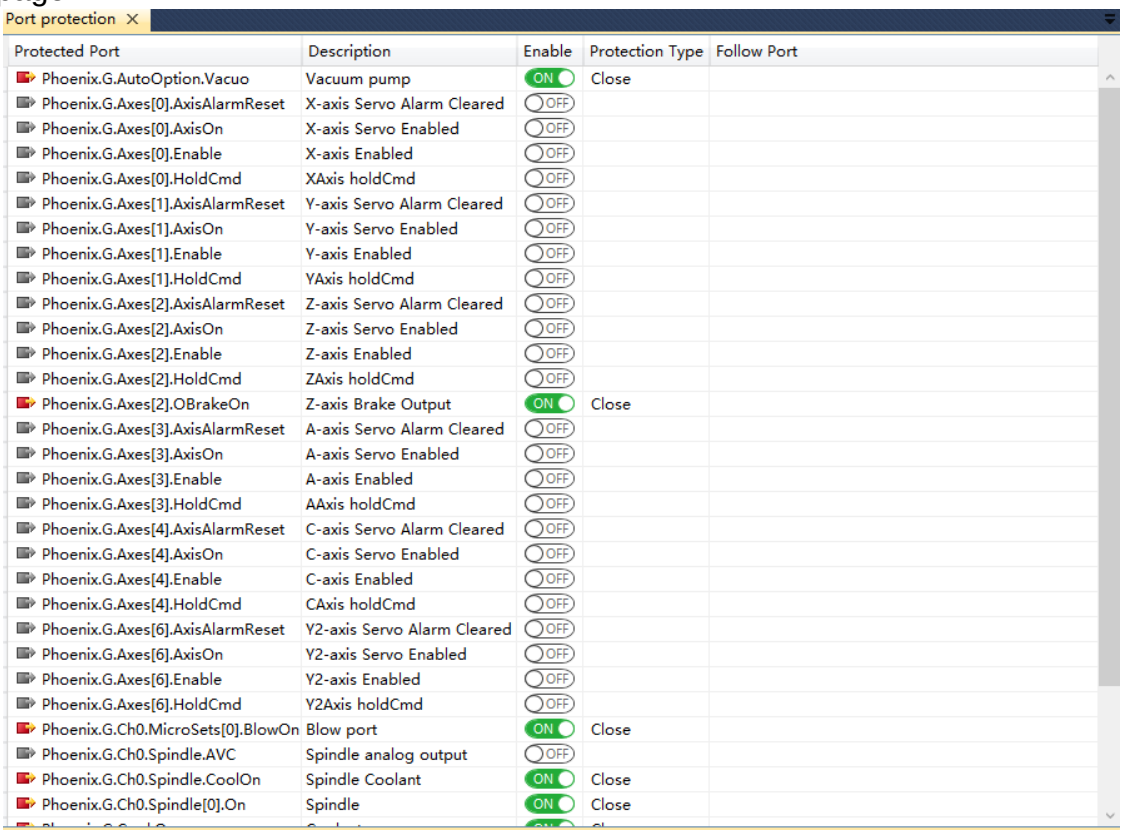
## 3.1.4 Set Port Protection

When the software is closed, the protected output port can be automatically close, open, follow or reverse according to the protection type.

| Protection Type | Description |
|---|---|
| Open | When opening the software, the port auto opens the signal. |
| Close | When the software is turned off, the port auto turns off the signal. |
| Follow | The following port status is consistent with the protected port status. |
| RFollow | The following port status and protected port status are reversed. |
| Hold | When the software is closed, the port keeps the current status. |

**Operation Steps:**

1. Click **Port Protection** in the left **Configuration** page to enter the **Port Protection** page：




2. Select the port to be protected and set the **Enable** cell to **ON**.
3. Select the target port, double click the **Protection Type** cell, and set the protection type according to the actual needs.
4. After setting, click  to save the current page or click  to save all pages.
   If it is not saved after editing, the * mark will appear in the upper right corner of the corresponding page button, and it will disappear after saving.

## 3.2 Adjust I/O Port Polarity

This operation monitors the machine tool status by controlling the polarity of input and output ports.

The polarity of input and output ports in the software is set according to the type of switch:

- Normally closed switch polarity is set to NC.
- Normally open switch polarity is set to NO.

The relationship between machine tool status and input/output ports is as follows:

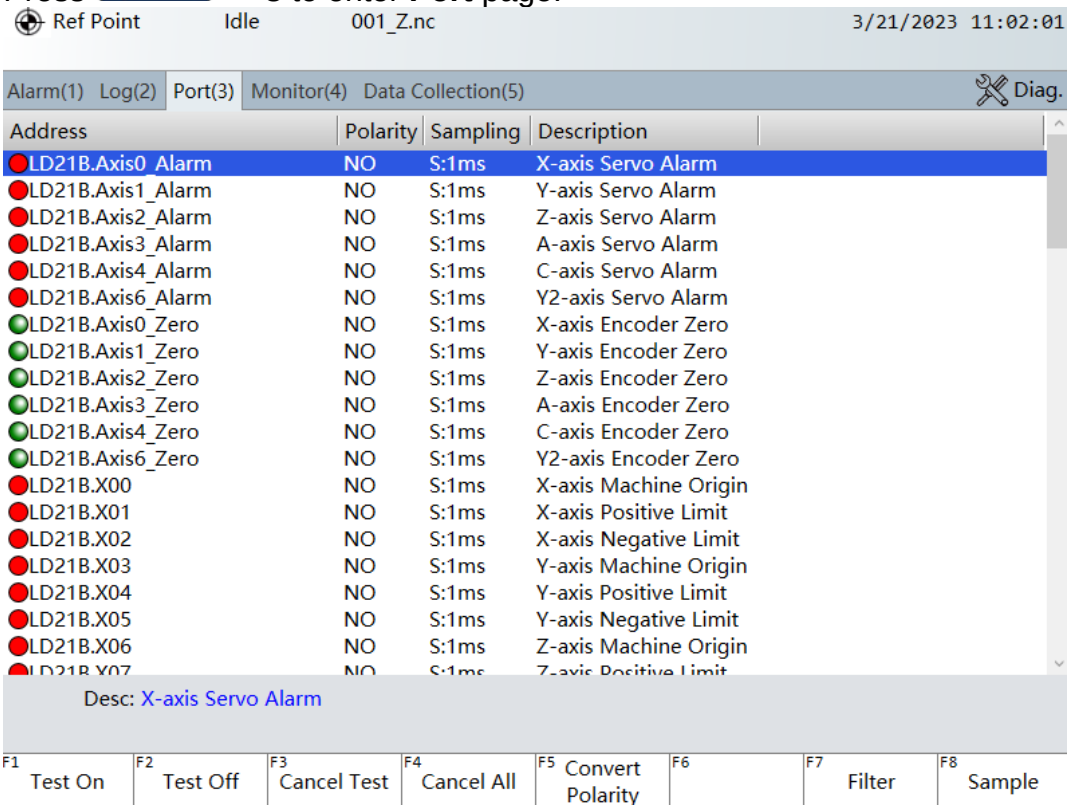
- Input port: 🔴 No signal; 🟢 Signal detected.
- Output port: ⭕ No signal;⭕ Signal detected.

**Prerequisite:**

The electrical lines have been correctly connected.

**Operation Steps:**

1. Press  → **3** to enter **Port** page:




2. Press ↑/↓ to select the target port, and press **F5** to modify the polarity of the port.
3. According to the actual needs, perform the following operations:
   - Press **F1** or **F2** to perform simulation test, and judge whether there is output through testing port signal.
     -  : The port is under test when there is no signal.
     -  : The port is under test when there is signal detected.
   - Press **F3** to deselect the port analog signal and simulation test, and replace the analog signal with the real hardware signal.
   - Press **F4** to cancel the simulation test of all ports.
   - Press **F5** to modify the port polarity.
   - Press **F7** to set the filtering duration, and the system will exclude the signal whose duration is small than the interval.
   - Press F8 to set the sampling interval to increase or decrease the filtering duration.

**Note**

- The smaller the sampling time of the input port, the higher the response of the system when receiving the signal.
- If the sampling time is too low, in some environments with poor grounding, it may cause false triggering of alarms or malfunctions.

## 3.3 Adjust the Axis Direction

This operation is used to ensure that the running direction of the machine tool is consistent with that specified in the **Right Hand Rule**, to avoid machine tool damage caused by incorrect axis direction during machine tool movement.

Take the X-axis as an example.

**Operation Steps:**

1. Press  → **F2** → **F8**, enter the manufacturer's password (123456) to view the value of axis mechanical specification parameter **Axis Direction (X)**:
   - 1：Positive direction
   - -1：Negative direction
2. Determine the positive direction of the X-axis according to the **Right Hand Rule**.
3. In the **Jog**/ **HandWheel**/ **Step** mode, press **X+** to move the X-axis, and observe whether the movement direction of the X axis is consistent with that determined by the **Right Hand Rule**:
   - Yes: the axis direction is correct.
   - No: modify the value of the parameter **Axis Direction (X)** to the opposite value.

## 3.4 Return to Mechanical Origin or Set Datum

This operation is used to adjust the coordinate position before machining. The machine tool coordinate system is the inherent coordinate system of the machine tool, and the origin of the machine tool coordinate system is also known as the mechanical origin or mechanical zero point. After the machine tool is designed, manufactured, and adjusted before leaving the factory, this origin point is determined. This operation is used to move the machine tool return to this origin.

According to the encoder type, this operation can be divided into:

- Return to Mechanical Origin: suitable for incremental encoder.
- Set Datum: suitable for absolute encoder.

### 3.4.1 Return to Mechanical Origin

Incremental encoder returns to mechanical origin, divided into two cases:

- Take the mechanical origin signal and encoder zero point signal
- Only take the mechanical origin signal, similar to step

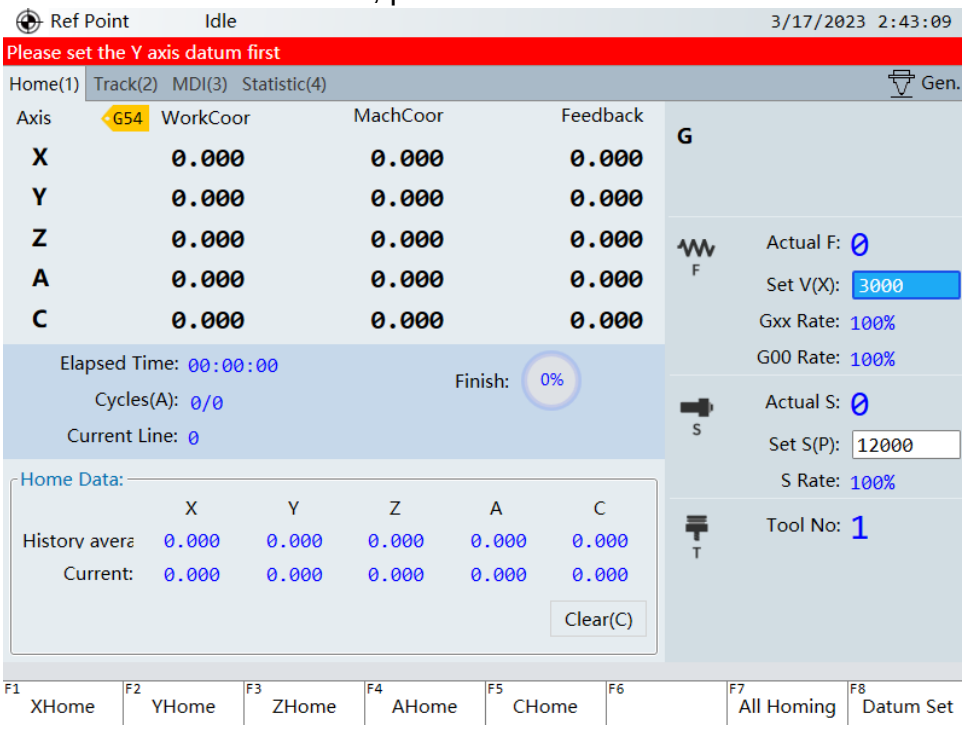
These two modes are switched by setting the origin parameter **Return to Mechanical Origin Using Encoder Feedback**.

**Prerequisite:**

- The Lambda terminal board configured in NcConfig is non-bus, such as 21A and 20A.
- The encoder used is an incremental encoder.
- The axis mechanical specification parameter **Encoder Type** of the software is set to 0.

**Operation Steps:**

1. In **Reference Point** mode, press  → **1** to enter the **Coordinate** page：




2. Select the corresponding axis to return to mechanical origin:
   - **F1**: X-axis return to mechanical origin.
   - **F2**: Y-axis return to mechanical origin.
   - **F3**: Z-axis return to mechanical origin.
   - **F4**: A-axis return to mechanical origin.
   - **F5**: C-axis return to mechanical origin.
   - **F7**: After Z-axis returns to the mechanical origin first, all other axes return at the same time.

After returning to the mechanical origin,  symbol appears in front of the corresponding axis name.


For safety, suggest to return to the Z axis first, and then to other axes.

## 3.4.2 Set Datum

This operation is used to set the zero point of the encoder, which is also known as datum. During this operation, there is no need to distinguish the order of axis return to the original point, and after saving or exporting datum settings, there is no need to repeat when software restart or update, power off, E-stop, etc., which can reduce the process and save time.

Take the X-axis as an example.

**Prerequisite:**

- The Lambda terminal board configured in NcConfig is bus type, such as 20B, 21B, and 21E.
- The encoder used is an absolute encoder.
- Software parameters axis mechanical specification parameter **Encoder Type** set to 1.

**Operation Steps:**

1. Manually move the X-axis to the machine tool mechanical origin position.
2. In **Reference Point** mode, press  → **F8** → **F1** to set X-axis datum.
   The system reads and records the mechanical coordinates of the X-axis. After setting successfully,  symbol appears before the X-axis name.
3. Restart the software to make the settings take effect.


There are two modes for bus return to mechanical origin:

- Directly to datum point, which is the position of mechanical coordinate 0 of each axis;
- Method of origin switch and encoder zero point signal: it is necessary to turn on the **Auxiliary Datum Setting** function in the origin parameter.

## 3.5 Set Workbench Travel

This operation sets the effective movement range of the machine tool in the X, Y, and Z directions by setting the upper and lower limits of the workbench travel, so as to protect the soft limit position.


If setting the workbench travel for the first time, it is necessary to confirm the actual effective range of machine tool movement before setting to prevent accidents.

**Operation Steps:**

1. Pass  → **F8** → **F2** to enter **Feed Axis Parameter** page.
2. Set the value of the parameter **Check Workbench Travel Range** to **Yes** to enable workbench travel.
3. According to the actual situation, set the following parameters:

| Parameter | Description |
|---|---|
| Workbench travel lower limit | The allowable mechanical coordinate value of the workbench lower limit when the parameter **Check Workbench Travel Range** is valid. |
| Workbench travel upper limit | The allowable mechanical coordinate value of the workbench upper limit when the parameter **Check Workbench Travel Range** is valid. |

# 4 Quick Start

Through this section, you can quickly familiarize yourself with the machining process of the NK300XC.

Complete the following operations to quickly start machining:

- Manage Program Files
- Tool Calibration
- Adjust the Workpiece Coordinate System
- Determine the Workpiece Origin

## 4.1 Manage Program File

This operation is used to manage machining files.

Press  , enter **Program** function area to manage the following types of files:

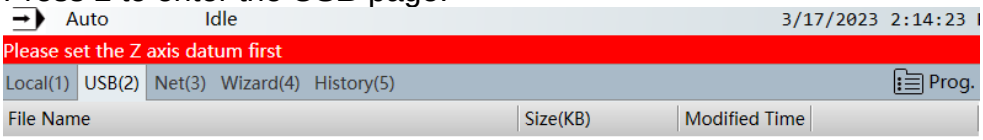
- Local program
- USB program
- Network program

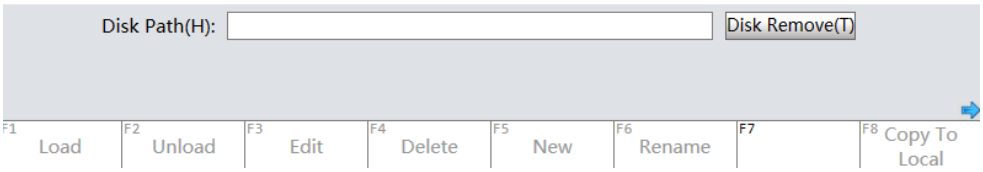
## 4.1.1 Manage Local/USB Program

This operation is used to manage the folders and saved program files displayed in the AIO root directory **D\NcFiles**, or the program files displayed in the folders and subdirectories under the USB flash disk root directory, including hidden files and folders.
Take the USB program as an example.

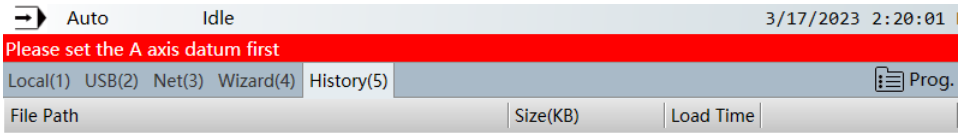
**Operation Steps:**

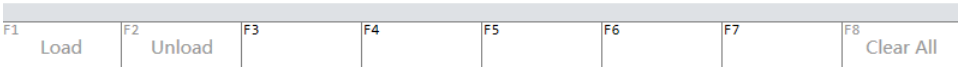
1. Press 2 to enter the **USB** page:




2. Press **H** to select the disk path.
3. Press ↑ / ↓ to select the target program file.
4. Press **F1** to load the target program file for machining.
5. (**Optional**): According to the actual situation, perform the following operations：
   - **F2**: Unload the loaded program file.
   - **F3**: Edit the target program file.
   - **F4**: Delete the target program file.
   - **F5**: Create a new program file.
   - **F6**: Rename the target program file.
   - **F8**: Copy the target program file to the AIO.
6. (**Optional**): Press 5 to view the loaded historical file information, and perform the following operations as required:




- F1: Load the selected file.
- F2: Unload the currently loaded file.
- F8: Clear all history.

## 4.1.2 Manage Network Path Program

Through operation establishes a correct connection between the **NK300CX** and the PC (personal computer), achieving file transfer between the AIO and the PC, so as to access shared files on the PC or other AIOs.

**Operation Steps:**

1. Set IP Address
2. Verify Connection
3. Access Shared Program File

### 4.1.2.1 Set IP Address

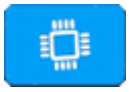
This operation is used to establish a network connection channel between the PC and AIO. The IP address can be obtained automatically or set manually. Automatic obtain is the default method of the system, without operation.

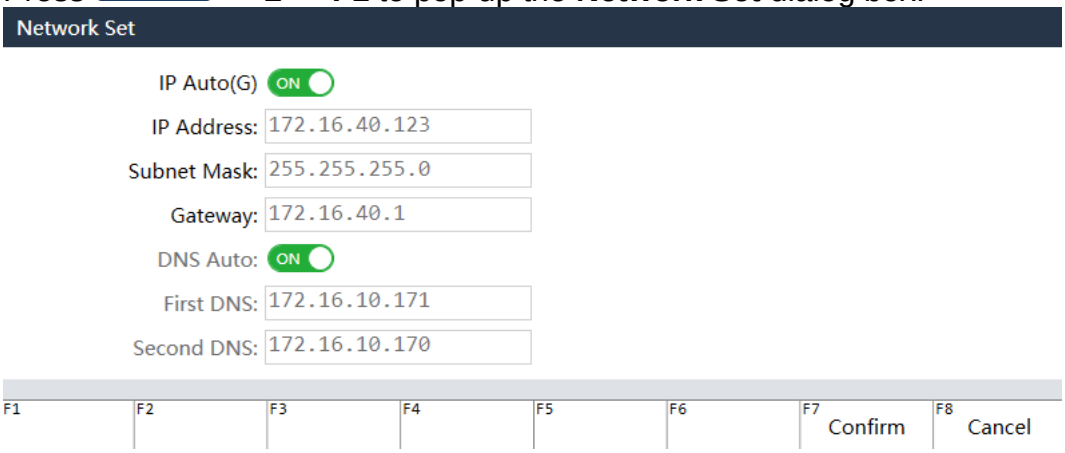
Take manually setting the IP address as an example.

**Prerequisite:**

PC and AIO have been placed on the same subnet.

**Operation Steps:**

1. Press  → **2** → **F1** to pop-up the **Network Set** dialog box:



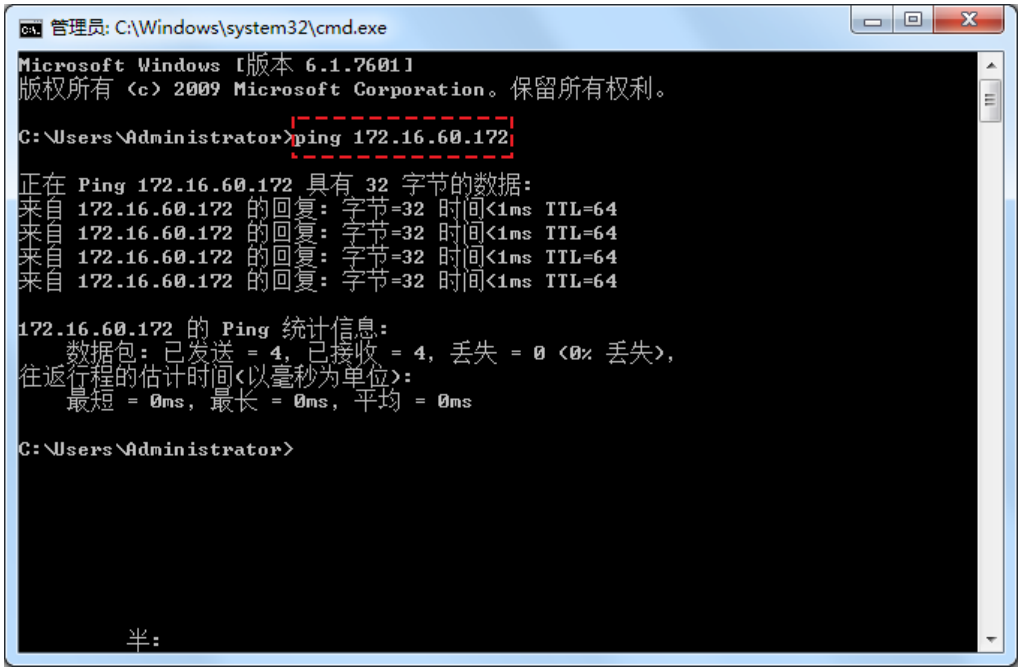

2. Press the shortcut key **G** to turn off the automatic obtain IP address.
3. Press ↑ / ↓ to select the corresponding input box and set the following address:
   - IP address: The first three groups are consistent with the PC terminal settings, and the last group is inconsistent with the PC terminal settings.
   - Subnet mask: consistent with PC.
   - Default gateway: consistent with PC.
4. Press **F7** to confirm, exit the **Network Set** dialog box and save the settings.
   The new network information is displayed in the **Network Connection Status** area of the **Computer Information** page.

#### 4.1.2.2 Verify Connection

After setting the IP address, use the ping command to verify whether the network connection channel between the PC and the AIO has been successfully established.

**Operation Steps:**

1. Press the **Win** + **R** keys at the same time on the PC, enter cmd in the **Run** dialog box, and click **OK** to call the command window.
2. Enter ping+IP address.
   For example: ping 172.16.60.172.
3. Press **Enter** to view the ping result:




Ping passed, network connection channel successfully established.

4. (**Optional**): If the connection fails, perform troubleshooting:
   - Check whether the network cable interface corresponding to the router has instructions.
   - Check whether the IP address setting of AIO is correct.

   For details, please see Set IP Address.

### 4.1.2.3 Access Shared Program File

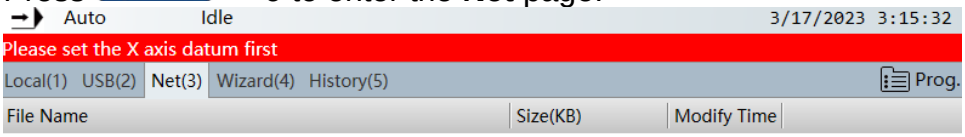
This operation is used to access shared files on the PC or other AIOs from AIO after successfully establishing a network connection channel.

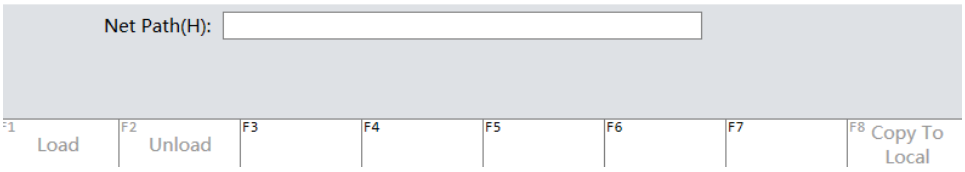
**Prerequisite:**

The target folder property has been set to **Shared Folder**.

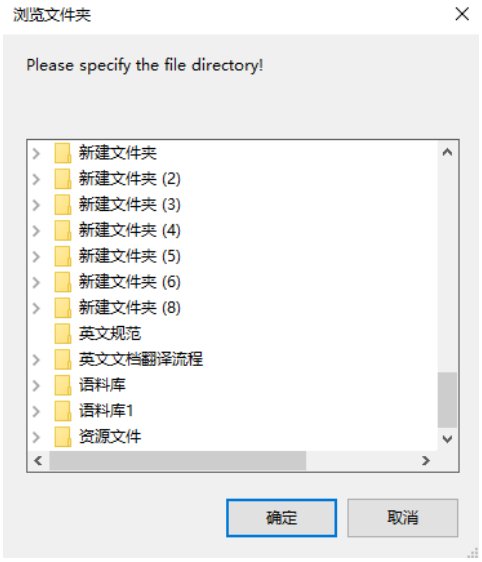
**Operation Steps:**

1. Press  → 3 to enter the **Net** page:




2. Press **H** to pop-up the **Browse Folder** dialog box:




3. Select **Network** → target PC name, find the folder you want to share, and click **OK.** The list of files in the shared file is displayed on the **Network Program** page.

4. Press ↑ / ↓ to select the target program file, and press **F1** to load the target program file for machining.
5. (**Optional**): According to actual needs, perform the following operations on the program file:
   - Press **F2**: Unload the loaded program file.
   - Press **F8** to copy the program file to AIO.

## 4.2 Tool Calibration

This operation is used to measure the selected tool to ensure that the selected tool can be machined normally on the blank.

The tool calibration type is divided into：

- Floating tool calibration
- Fixed Tool Calibration
- Tool Calibration for the First Time/After Tool Change: default method.

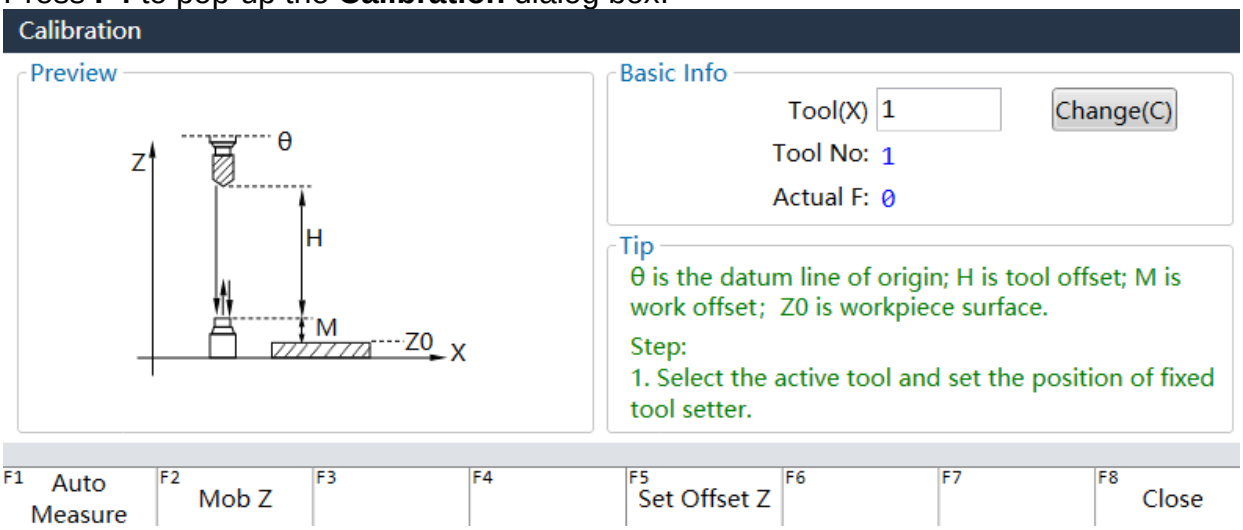
This operation mainly introduces the last two commonly used tool calibration methods.

### 4.2.1 Fixed Tool Calibration

When replacing a tool during machining due to tool wear or borke, the length and clamping position of the tool can change. Set the tool at a fixed position on the machine tool to make sure the tool length is offset to achieve accurate machining. Commonly used in machine tools with tool magazine.

**Operation Steps:**

1. Press  → **F5** → **F8**, enter the manufacturer's password to enter the **Tool Setting** page.
2. Set the following parameters:
   - Tool calibration type: set to 1.
   - Fixed tool calibration presetter position: Set parameters according to the actual position of the tool calibration presetter.
3. In **Auto**/ **Jog**/ **Handwheel**/ **Step** mode, press  → 1 to enter **Coordinate** page.
4. Press **F4** to pop-up the **Calibration** dialog box:

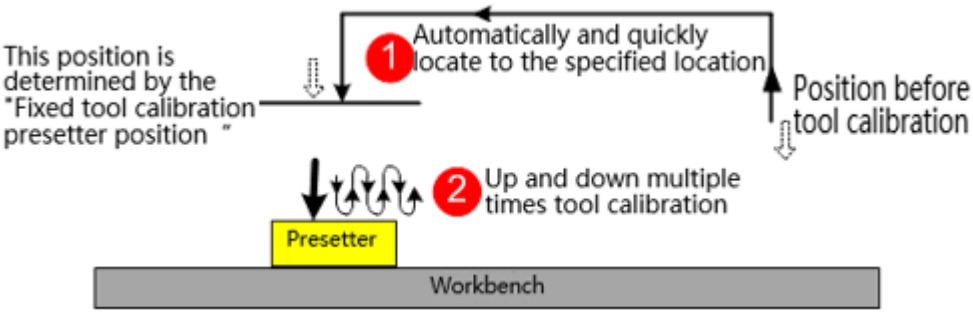



5. Press **F1** to fix and calibrate the selected tool.

**Note**

When the entered tool number is different from the current spindle tool number, the system will automatically change the tool before performing fixed tool calibration.

The system automatically performs tool calibration according to the following process, and saves the results to the tool offset.



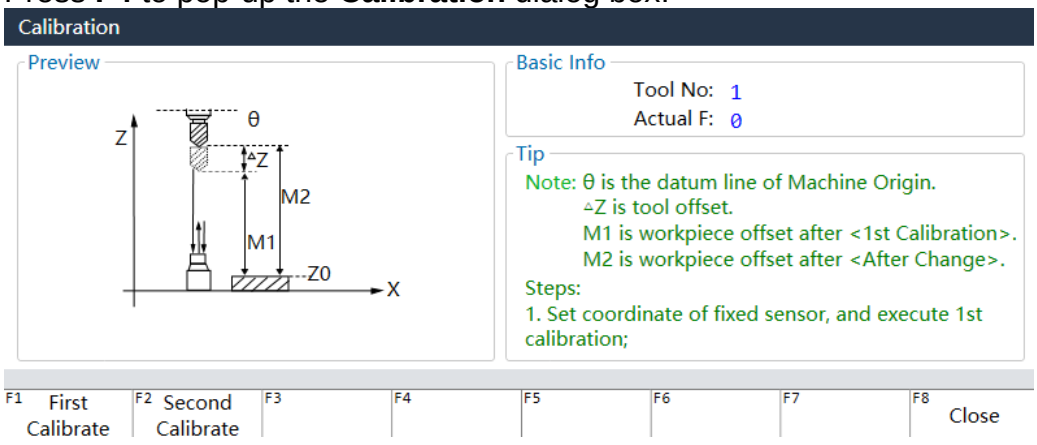

After the fixed tool calibration is completed, it is necessary to move the tool to the workpiece surface for Clearing.

## 4.2.2 Tool Calibration for the First time/After Tool Change

This operation is used to compensate the tool difference into the workpiece offset.

**Operation Steps:**

1. Press  → **F5** → **F8**, enter the manufacturer's password to enter the **Tool Setting** page.
2. Set the parameter **Tool Calibration Type** to 2.
3. Manually move the Z-axis to the workpiece surface and determine the workpiece origin through manual Clearing .
4. In **Auto/ Jog/ Handwheel/ Step** mode, press  → 1 to enter the **Coordinate** page.
5. Press **F4** to pop-up the **Calibration** dialog box:




6. According to the actual situation, select the following operations:

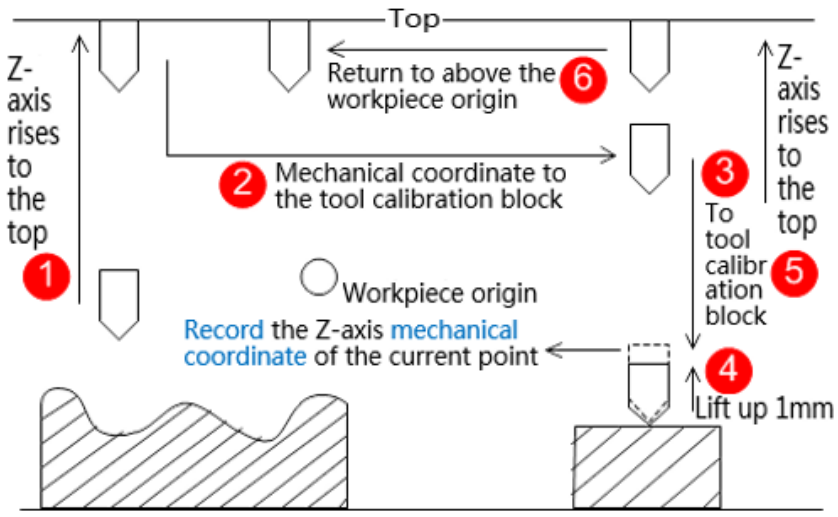
- If it is the first calibration, press **F1** to perform the first tool calibration, and the system will automatically record the mechanical coordinate value of the Z-axis at this time.
- If it is tool calibration after tool change or broke, press **F2** to perform tool calibration after tool change, and the system will automatically restore the workpiece coordinate value of the current point Z-axis.

**Note**

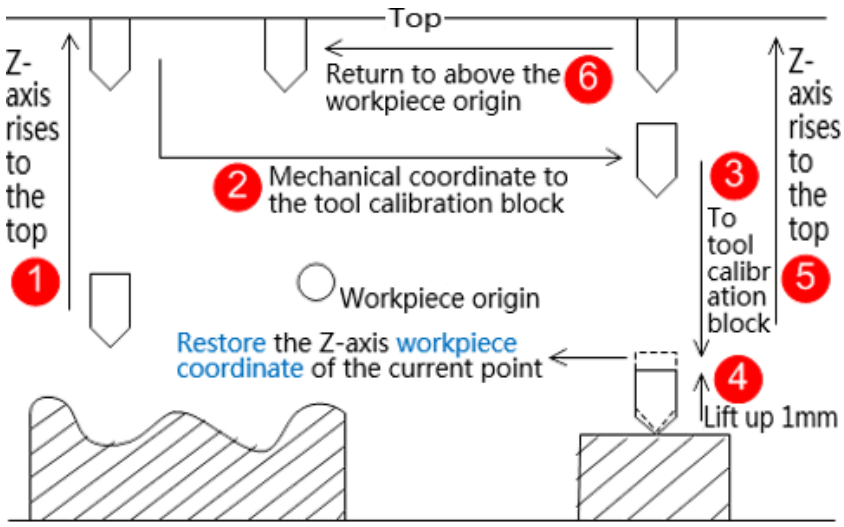
The tool calibration after tool change can only be performed after the first tool calibration is completed.

---

- First tool calibration process




- Tool calibration process after tool changse

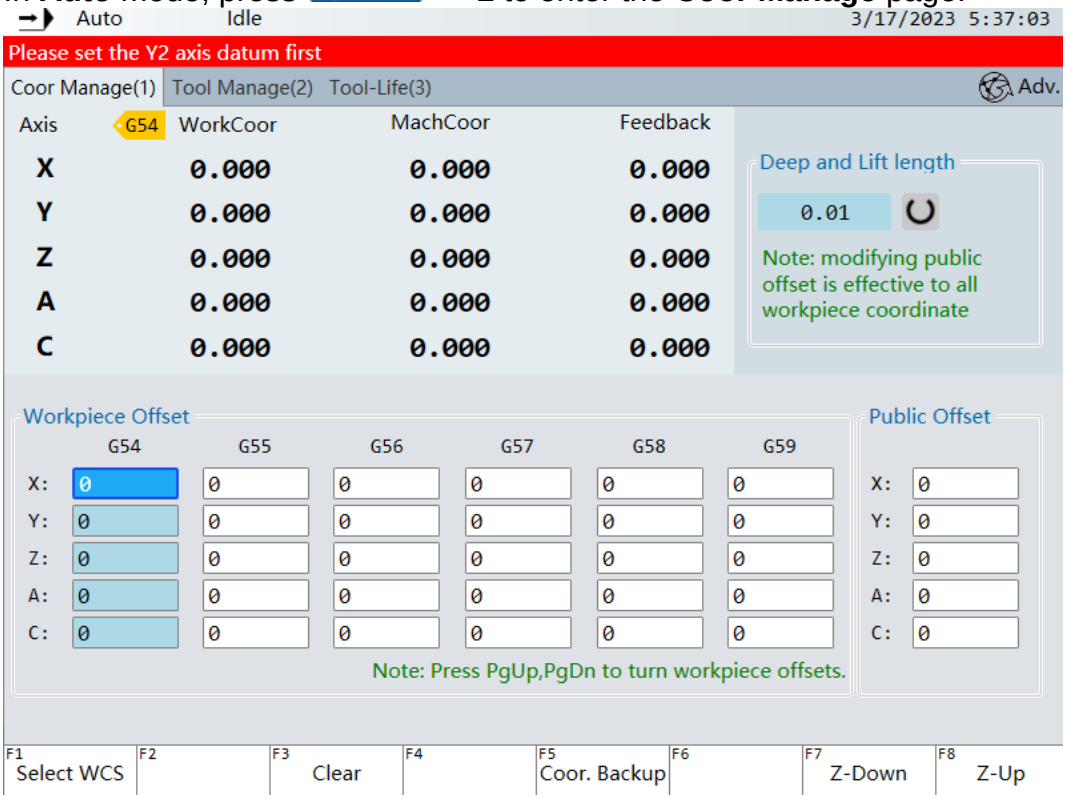


# 4.3 Adjust the Workpiece Coordinate System

During programming, the programmer selects a known point on the workpiece as the workpiece coordinate system origin (also known as the program origin), and establishes a new coordinate system called the workpiece coordinate system. This operation is used to adjust the workpiece coordinate system.

**Operation Steps:**

1. In **Auto** mode, press  → **1** to enter the **Coor Manage** page:




2. (**Optional**): Select the Workpiece Coordinate System.
3. Set Workpiece Offset and Common Offset.
4. (**Optional**): Access Coordinate.

## 4.3.1 Select the Workpiece Coordinate System

This operation is used to select the workpiece coordinate system for machining from G54 to G59.

**Operation Steps:**

1. Press ← / → to move the cursor to the target workpiece coordinate system bar. The coordinate system is displayed in blue.
2. Press **F1** to switch to the target workpiece coordinate system. The corresponding **GXX** character is highlighted in the workpiece offset area. If the current coordinate system is the workpiece coordinate system of except G54, the **Coordinate** interface of the **Machining** function area and the **Coordinate Manage** interface of the **Advanced** function area also highlight the coordinate system.

### 4.3.2 Set Workpiece Offset and Common Offset

Set the workpiece offset for a certain workpiece coordinate system, which is used to set the offset of the workpiece origin relative to the mechanical origin; Set the common offset for all coordinate systems to adjust the workpiece origin for each axis.

**Operation Steps:**

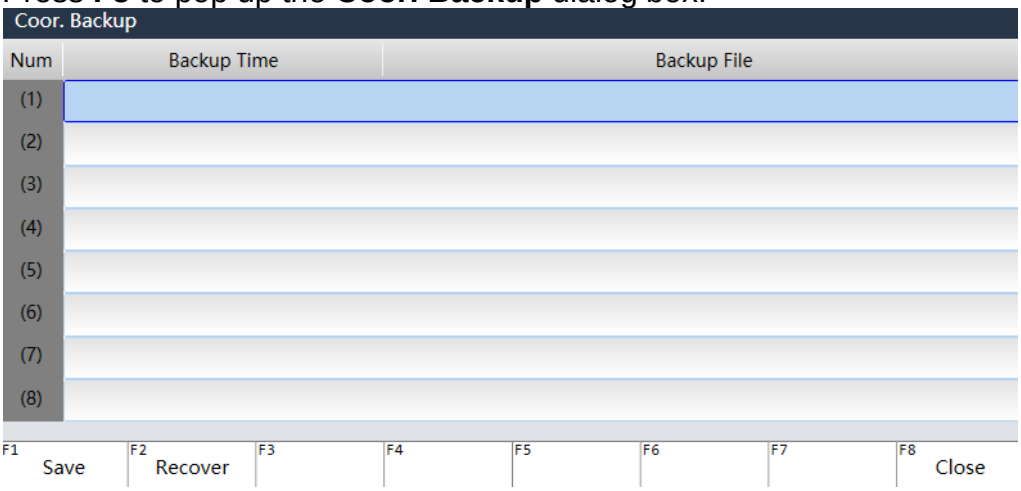
1. Set workpiece offset:
   a. Press ↑ / ↓ / ← / → to select the input box for the workpiece offset of the corresponding axis.
   b. Press **Enter** to enter the modified value in the pop-up input box.
2. Set common offset:
   a. Press ↑ / ↓ / ← / → to select the input box for the common offset of the corresponding axis.
   b. Press **Enter** to enter the modified value in the pop-up input box.
   c. (**Optional**): Press **F7** or **F8** to enter an adjustment value for the Z-axis feed rate, deepen (decrease) or raise (increase) the common offset of the Z-axis.
   The value itself does not distinguish between positive and negative and is an additive value, which is only valid for the common offset of the Z-axis.

### 4.3.3 Access Coordinate

This operation is generally used to backup the position of the workpiece origin corresponding to different workpiece/tool paths, enabling quick positioning when easily changing the tool path.

**Operation Steps:**

1. Press **F5** to pop up the **Coor. Backup** dialog box:



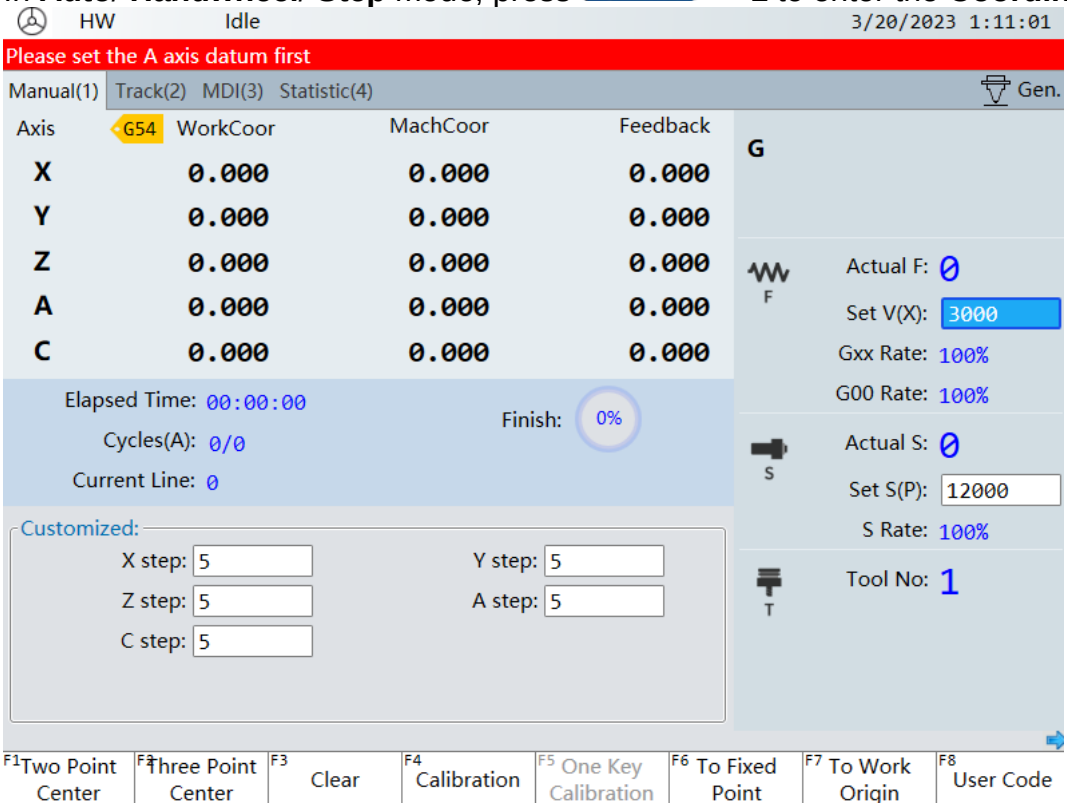

2. Perform the following operations according to actual needs:
   - Press ↑/↓ to select the target line, and press **F1** to save the current coordinate position information.
     If the target line already has data, the current data will overwrite the original data.
   - Press ↑/↓ to select the target line, and press **F2** to return the tool to the saved coordinate position.

## 4.4 Determine the Workpiece Origin

When programming, the programmer selects a known point on the workpiece as the workpiece origin, also known as the program origin. Generally, the selection of workpiece origin meets the conditions of simple programming, simple size conversion, and small machining errors caused. This point can float relative to the mechanical origin.

**Operation Steps:**

1. In **Auto/ Handwheel/ Step** mode, press  → **1** to enter the **Coordinate** page:




2. Select any of the following methods to determine the workpiece origin:
   - Clearing.
   - Centering: Suggest in **Handwheel** mode.

## 4.4.1 Clearing

This operation is used to reset the current workpiece coordinates of each axis and set the mechanical coordinate values of the corresponding axis or all axes to the corresponding workpiece offset values.

**Operation Steps:**

1. Move the machine tool to the position to be set as the workpiece origin.
2. Press **F3** to switch the operation button to clearing related functions.
3. Select the axis to perform the clearing operation as needed:
   - **F1**: Clearing the workpiece coordinate of the current X-axis.
   - **F2**: Clearing the workpiece coordinate of the current Y-axis.
   - **F3**: Clearing the workpiece coordinate of the current Z-axis.
   - **F6**: Clearing the workpiece coordinate of the current X-axis and Y-axis.
   - **F8**: Clearing the workpiece coordinates of all current axes.

## 4.4.2 Centering

This operation is used when the workpiece origin cannot be directly determined but is known to be in the middle or at the center of the circle. Apply to the workpiece origin of regular workpiece (regular rectangle/circle) and set the workpiece offset.
Include:

- Two point center: By recording the two point coordinates of the regular rectangle blank, the center point coordinates are automatically calculated and used as the workpiece origin.
- Three point center: By recording the three point coordinates on the circumference of a circular blank, the center point coordinates are automatically calculated and used as the workpiece origin.

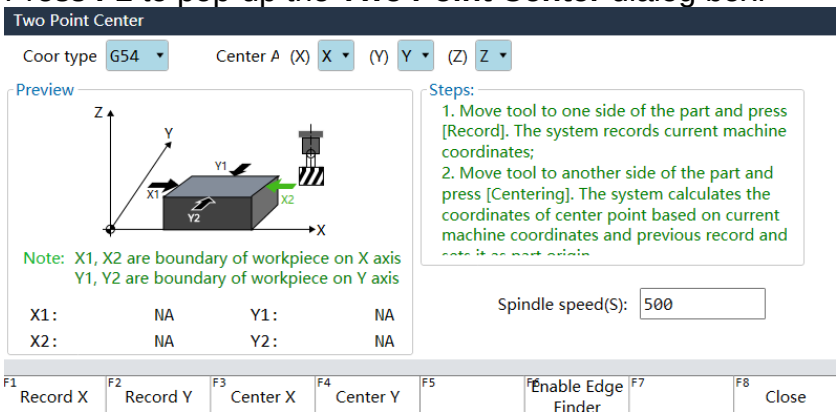
The two methods of operation are similar. This operation takes the two-point center X-axis as an example.

**Note**

When centering an axis, it is necessary to note that other axes hold still.

**Operation Steps:**

1. Press **F1** to pop-up the **Two Point Center** dialog box:




2. Press **G** to select the target workpiece coordinate system.
   The **Coordinate** page and **Coor. Manage** page are synchronously updated to the selected workpiece coordinate system.
3. (**Optional**): Press **F6** to use the **Edge Finder** for accurate positioning:
   - Use: **Spindle Speed** is valid during centering. Proceed to the next step.
   - Not used: **Spindle Speed** is invalid during centering.
4. Start the spindle, and the spindle speed is the set value in the software or the set value in the program file.
5. (**Optional**): In the input box for **Spindle Speed** during centering, set the spindle speed during centering.
   This value should not be too large. Default is 500 RPM.
6. Move the X-axis to the side of the workpiece, press **F1**, and record the mechanical coordinates of the X-axis.
7. Move the X-axis to the other side of the workpiece and press **F2.**
   The system calculates the center point coordinate of the X-axis based on the current position coordinates and the coordinates recorded in the previous step, and sets it as the workpiece origin.

# 5 Common Operation

This section describes how to use common operations.
Include the following:



## 5.1 Handwheel Guide

This operation is used to manually control the execution speed of the processing program during automatic processing, which can prevent appearance of tool damage caused by incorrect program installation or improper program.
This article takes the Weihong handwheel as an example to perform the operation instruction. The diagram of the Weihong handwheel is as follows:


**Operation Steps:**

1. In **Auto/ Jog/ Reference Point** mode, select one of the following methods to enable handwheel guide:
   - Press  .
   - Press  → **1** → **F1**.
2. Press  to start using the handwheel to control machining.

3. Rotate the handwheel clockwise to control the rotary disk and execute the machining program.
   The machining speed changes with the shaking speed of the handwheel. When the handwheel stops, the program machining stops, and the machine tool stops moving within 300ms.
4. (**Optional**): Set the parameter **Reverse Machining Buffer**.

   **Note**

   Only the parts within the buffer range can support reverse machining. When the parameter range is exceeded, rotate the handwheel counterclockwise to stop machining and the machine tool will not move.

## 5.2 Single Block

This operation sets the machining task to be executed as single block, easy to error diagnosis and fault recovery. When single program block is executed, the program stops every time the speed of each axis is 0.

**Operation Steps:**

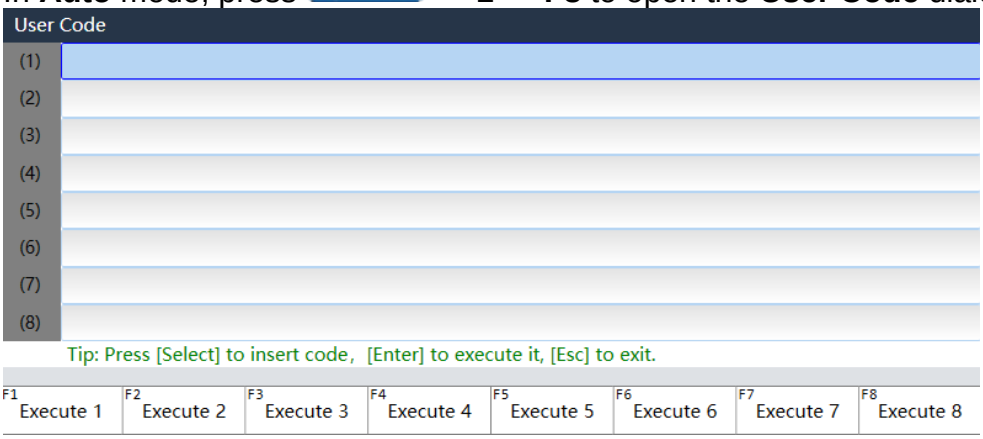
1. Press  to enable the single block function.
2. Press  , The software enters the pause state after executing a program.
3. Repeat step 2 until the whole machining program is executed.

## 5.3 MDI

This operation is used to input and execute up to 7 programming commands to achieve rapid movement, change the system state or perform simple machining, etc.

**Operation Steps:**

1. In **Auto** mode, press  → **1** → **F8** to open the **User Code** dialog box:



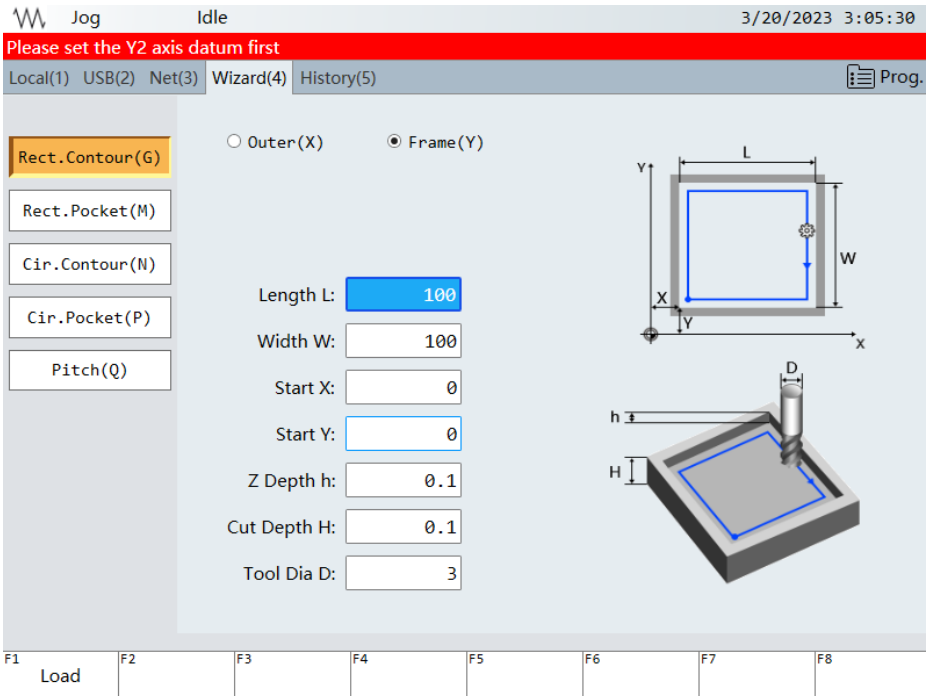

2. Press ↑ / ↓ to move to the target row.
3. Enter the command in the input box and use a semicolon to break the line.
4. Press **F1** ~ **F7** to execute the corresponding command.

# 5.4 Use the Machining Wizard

This operation quickly generates a machining program by using the machining wizard function.

**Operation Steps:**

1. Press  → **4** to switch to the **Wizard** page:




2. Press the corresponding shortcut key to select the wizard program:
   - G: Rect. contour
   - M: Rect. pocket
   - N: Cir. contour
   - P: Cir. pocket
   - Q: Pitch
3. Set relevant parameters according to the diagram.
4. Press **F1** to load the generated tool path file into the system and wait for machining.

For details of the wizard program, please see Use the Machining Wizard.

# 5.5 Jiggle Tool Head

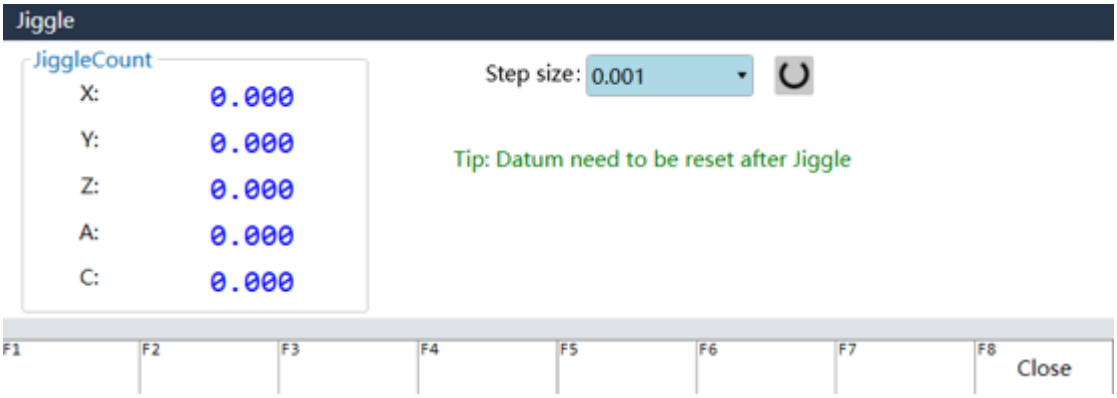
This operation is used to adjust the tool head to find the optimal cutting distance when the machining process is not in place. The jiggle result is only valid for this machining task, and becomes invalid after the machining is stopped.

**Prerequisite:**

- Machining mode is automatic mode.
- The system is in machining or paused state.

**Operation Steps:**

1. Press  → **1** →  → **F6** to open the **Jiggle** dialog box:




2. Press 

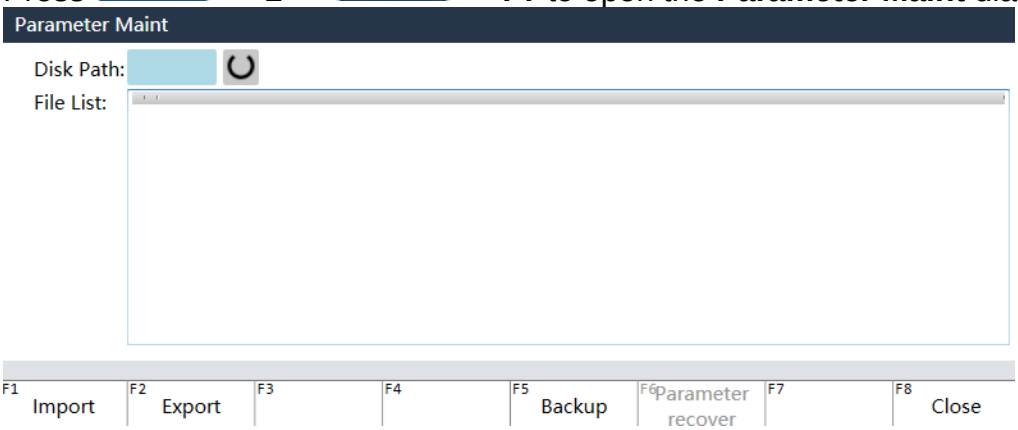
to select the appropriate step size value.
The system provides seven step values of 0.001, 0.01, 0.05, 0.1, 0.2, 0.5, and 1.
3. Select the following mothods to start jiggle:
   - Press the corresponding axis direction key on the panel.
   - Use the handwheel to adjust the corresponding axis.
     For details of handwheel operation, please see Handwheel Guide.
4. If the current state is paused, after the jiggle is completed, press  to continue machining.

## 5.6 Restore Parameter Backup

This operation is used to restore the parameters of the system's automatic backup. When the user forgets to save the parameters after setting them, this operation can be used to restore the parameter settings from the factory date to the last time the system was shut down.

**Operation Steps:**

1. Press  → **1** →  → **F7** to open the **Parameter Maint** dialog box:



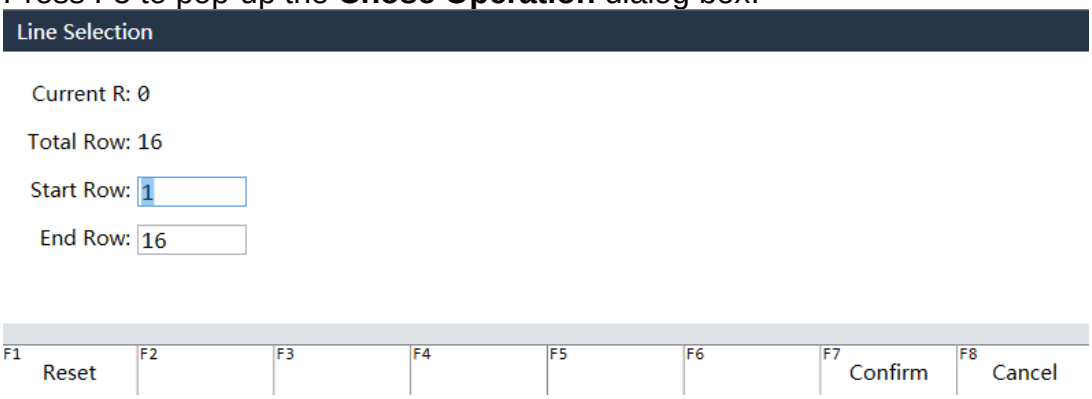

2. Press ↑ / ↓ to select valid backup parameters.
3. Press **F1** to restore the selected backup parameters.
4. (Optional) Press **F2** to export parameters to USB disk.
5. (Optional) Press **F5** to perform parameter backup operation.
6. (Optional) Press **F6** to perform the parameter recovery operation. The recovered data is the data from the parameter backup.

## 5.7 Line Selection

This operation starts and ends the row number with a custom command or searches for the command line corresponding to the specified tool number to run the target command.

**Operation Steps:**

1. In **Auto** mode, press  → 1 to enter the **Coordinate** page.
2. Press F5 to pop-up the **Chose Operation** dialog box:



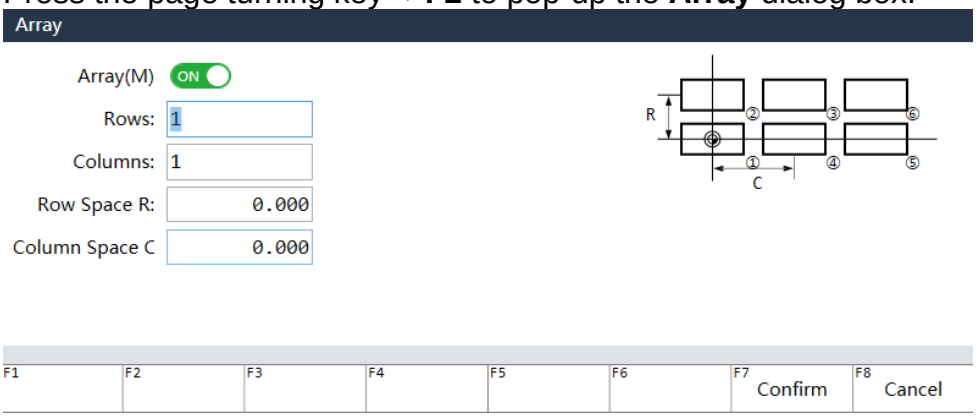

   - Current R: Displays the row number of the current machining program command.
   - Total row: Displays the total number of program commands in the current program file.
3. Specify the **Start Row** and **End Row** of the machining.
4. (**Optional**): Press **F1** to restore the default settings.
5. Press **F7** to apply the settings.
   Follow up machining according to the settings.

## 5.8 Array Machining

This operation is used to use the same tool path, machining multiple workpieces in different positions, simplifying user programming operations, etc.

**Operation Steps:**

1. In **Auto** mode, press  → 1 to enter the **Coordinate** page.
2. Press the page turning key→ **F2** to pop-up the **Array** dialog box:




3. Press **M** to enable array machining and set the following values:
   - Rows
   - Columns
   - Row space
   - Column space
4. Press **F7** to apply the settings.
   Follow up machining according to the settings.

## 5.9 Return to Workpiece Origin

This operation is used to control the spindle to automatically return to the workpiece origin from the current position.

**Operation Steps:**

Press  → **1** → **F7** to start returning to the workpiece origin.

# 5.10 Mirror and Rotate

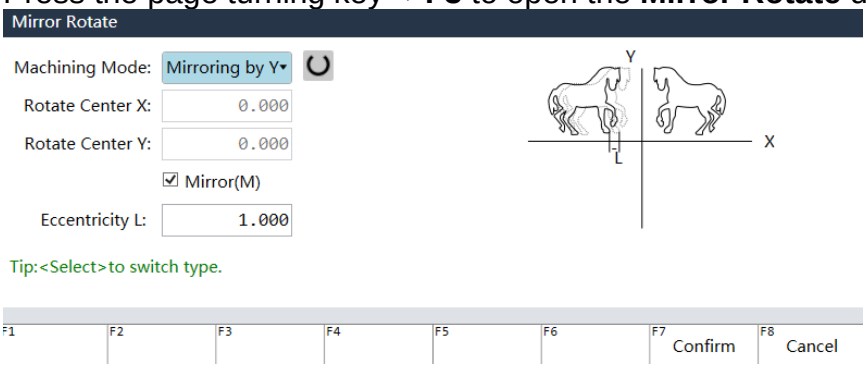
This operation is used to perform mirror rotation machining on the same machining program.

**Prerequisite:**

The system is in auto mode.

**Operation Steps:**

1. Press  → **1** to enter the **Auto** page.
2. Press the page turning key→ **F3** to open the **Mirror Rotate** dialog box:




3. Press 

to select the machining method:
   - Convention
   - Mirroring by X
   - Mirroring by Y
   - CW 90°
   - CW 180°
   - CCW 90°
4. (**Optional**): According to actual needs, perform the following operations:
   - Set the X and Y coordinates of the rotation center.

     If the rotation center is not specified, the software defaults to the current rotation center as the workpiece origin.

   - Press **M** to enable **Mirror** and set the **Eccentricity L**.
5. Press **F7** to confirm.

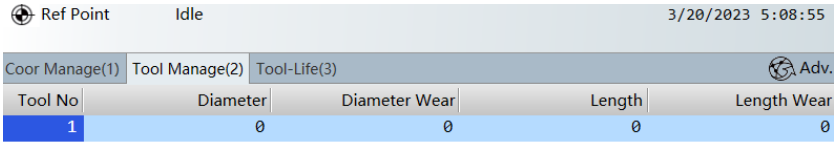
# 5.11 Execute Tool Compensation

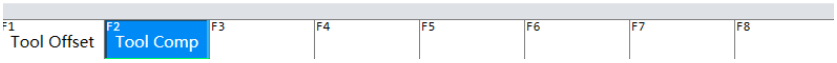
This operation is used to calculate the coordinate position of the tool center or related point of the tool holder based on the actual coordinate position of the tool tip or tool edge (that is, the actual coordinate position of the part contour), in order to achieve accurate machining.

When tool wear, sharpen, or change tools cause changes in the tool tip radius, changing the corresponding tool parameter values on the tool manage page can achieve accurate machining without modifying the machining program.

**Operation Steps:**

1. Set the parameter **Tool Radius Compensation Valid** to **Yes**.
2. (**Optional**): Set parameter **Number of Interference Detection Patterns**:
   Interference refers to over cutting caused by excessive tool radius.
   Perform interference detection between several adjacent graphics. When interference occurs, an interference alarm will be triggered. Generally, setting a smaller tool diameter can clear the alarm.
3. Measure and record the following values:
   - Diameter
   - Diameter wear
   - Length
   - Length wear
4. Press  → **2** to switch to the **Tool Manage** page, and then press **F2** to switch to enter the **Tool Comp** interface:




5. Press ↑ / ↓ / ← / → to select a cell and press **Enter** to set parameters based on the measured value.
   During machining, if there is a tool compensation command in the machining program, the system will automatically compensate. For details, please see NcStudio Programming Manual.

## 5.12 Lead Screw Error Compensation

This operation is used to compensate for pitch errors and errors generated by backlash to achieve high-precision machining.

Error compensation methods include:

- Only backlash compensation.

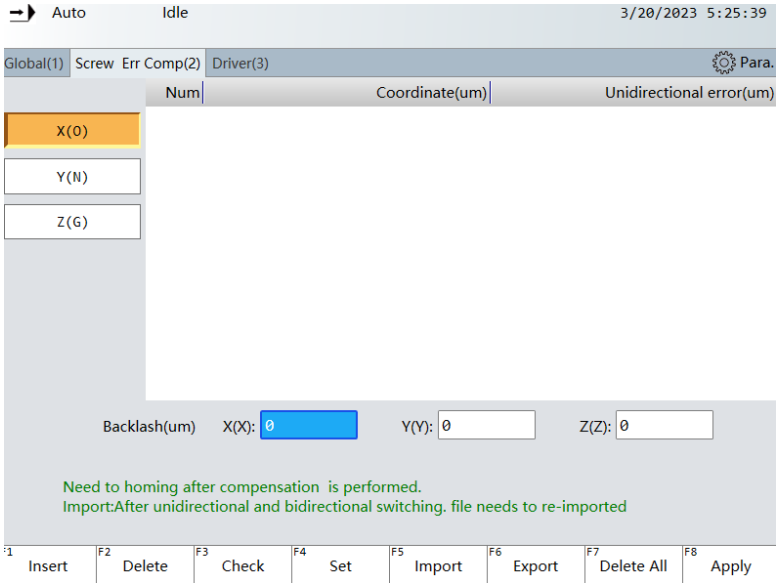
- Backlash and unidirectional compensation: Use the backlash value and unidirectional error data to compensate for stable errors in both directions of positive and negative movement.
- Bidirectional compensation: Use forward and reverse error data for comprehensive compensation, suitable for situations where there are large errors in both directions of positive and negative movement.

**Prerequisite:**

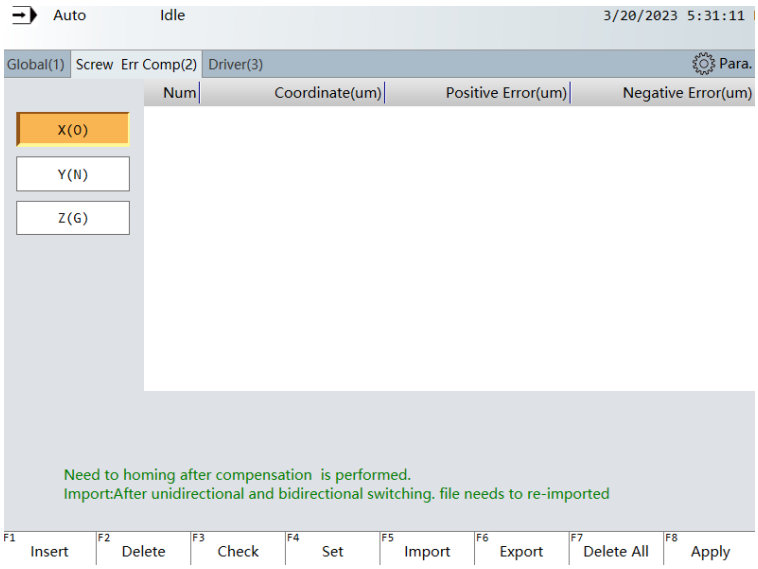
Parameter **Screw Err Comp** method has been set**.**

**Operation Steps:**

1. Press  → **2** to switch to the **Screw Err Comp** page:
   - Unidirectional error:




   - Bidirectional error:




2. Select one of the following methods to compensate for the lead screw error:
   - Through Lead Screw Error File Compensation
   - Through Parameter Setting Compensation

## 5.12.1 Through Lead Screw Error File Compensation

This operation is used to import compensation data files into the software to achieve lead screw error compensation.

**Prerequisite:**

Store the screw error compensation file named with the suffix (*.lin,*.rtl,*.REN,*.pos,*.cmp,*.dat,*.xml) in the U disk root directory.

**Operation Steps:**

1. Insert the U disk into the system host.
2. Press **F5** to pop-up the **Import** dialog box.
3. In the file list, press ↑ / ↓ to select a compensation file.
4. Press **F7** to import the file into the software.

**Note**

When switching between backing and unidirectional compensation, it is necessary to re import the used file and update the error value in the page.

5. Press **F8** to write the compensation data to the driver and save the compensation file to the axeserr.xml file on the D drive.
6. (**Optional**): If the screw error compensation file has been modified in the system, you can press **F6** to copy the compensation file to the USB disk.

**Note**

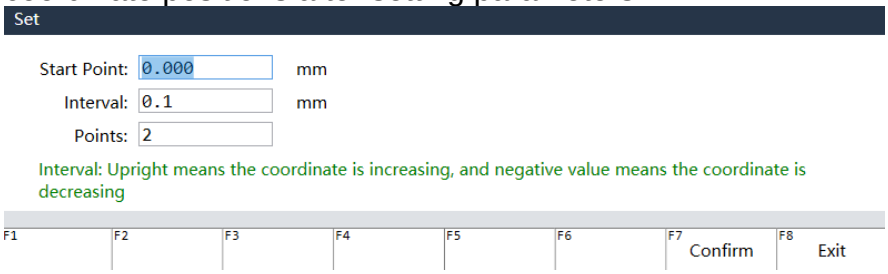
If a compensation file is modified in the system, the software needs to be restarted before the modified compensation file can take effect.

## 5.12.2 Through Parameter Setting Compensation

This operation is used to directly set compensation parameters to achieve lead screw error compensation.

**Operation Steps:**

1. Press **O** / **N** / **G** to select the axis to be compensated.
2. Select one of the following methods:
   - Press **F4** to pop up the **Set** dialog box, and generate a set of compensated coordinate positions after setting parameters:




   - Press **F1** to insert a blank row.

Multiple blank rows cannot be inserted consecutively. Insert one row and fill in the data before inserting it again.

3. Press ← / → to select the target parameter, and press **Enter** to set the data.
4. (**Optional**): For unidirectional compensation, press **X** / **Y** / **Z** to set the backlash of the corresponding axis.
5. Press **F3** to check whether the input compensation data meets the rules:
   - Legal: Proceed to the next step.
   - Illegal: Modify as prompted.
6. Press **F8** to execute the application and write the compensation data to the driver to make the settings effective.
   After executing the application, it must be executed back to the mechanical origin.
7. (**Optional**): Press **F6** to export the compensation data to the error compensation file on the USB disk.

## 5.13 Execute QEC

This operation is used to eliminate distortion (often sharp corners) that occurs at the transition from one quadrant to the other when the machine tool is machining an arc. Through setting parameters to achieve QEC, the settings in the positive and negative directions of X, Y, and Z are similar.

**Operation Steps:**

1. Press  → **1** to enter **Global** page.
2. Press F8 to enter the manufacturer's password (123456), find and set the following parameters:

| Parameter | Description |
|---|---|
| QEC is valid | Whether the system use the arc over quadrant sharp angle compensation function.<br>Yes: use; No: not use. |
| QEC positive/negative intensity | The intensity of QEC when the motion direction changes from positive to negative or from negative to positive during arc interpolation.<br>The higher the value, the greater the QEC effect.<br>Setting range: 0~3. |
| QEC forward/negative delay time | The delay time of QEC when the motion direction changes from positive to negative or from negative to positive during arc interpolation.<br>Setting range: 0~0.03. |

## 5.14 Use the Nc Cloud

NC Cloud is embedded in the system to provide data interfaces and communication protocols.

**Operation Steps:**

1. Press  → **1** to switch to the **System** page.
2. Press **F4** and follow the command in the pop-up window.

Please refer to the Nc Cloud Assistant User Manual for details.

# 6 System Maintenance and Management

This section describes operations related to operating system maintenance and NcStudio software maintenance.

Include:

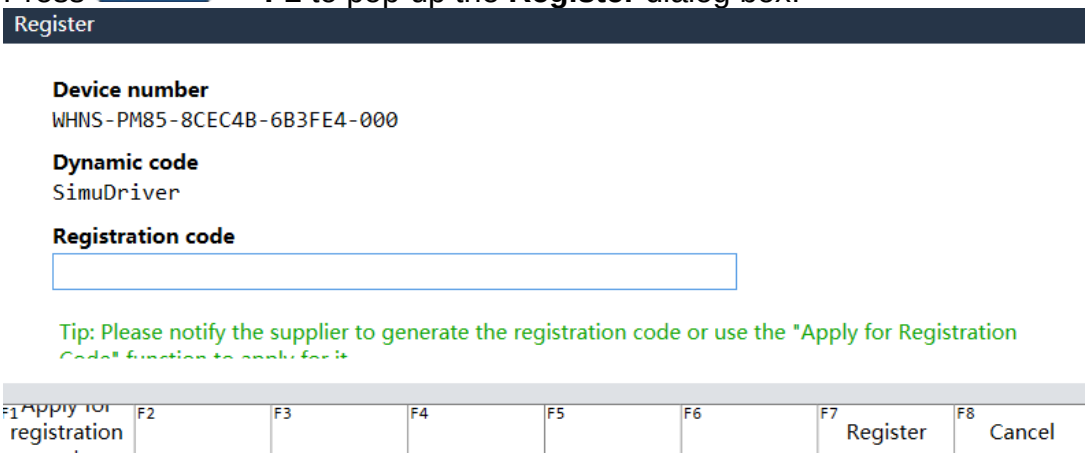


## 6.1 Register Software

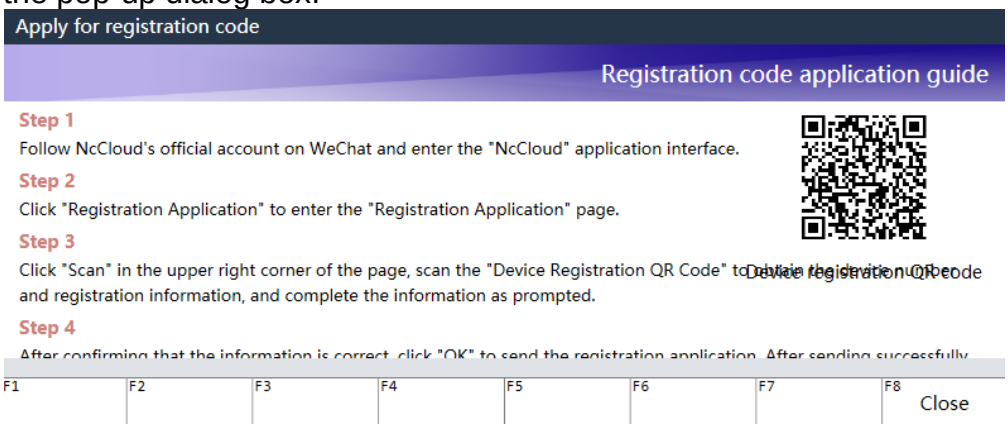
When the software is about to expire, has expired, and the customer's write number type does not match, it is necessary to register the software.

**Operation Steps:**

1. Press  → **F1** to pop-up the **Register** dialog box:




2. Select one of the following methods to obtain the registration code:
   - Send the device number to the machine tool manufacturer or developer, and the manufacturer or developer returns a registration code.
   - Click **Apply for Registration Code** and complete the operation according to the pop-up dialog box:



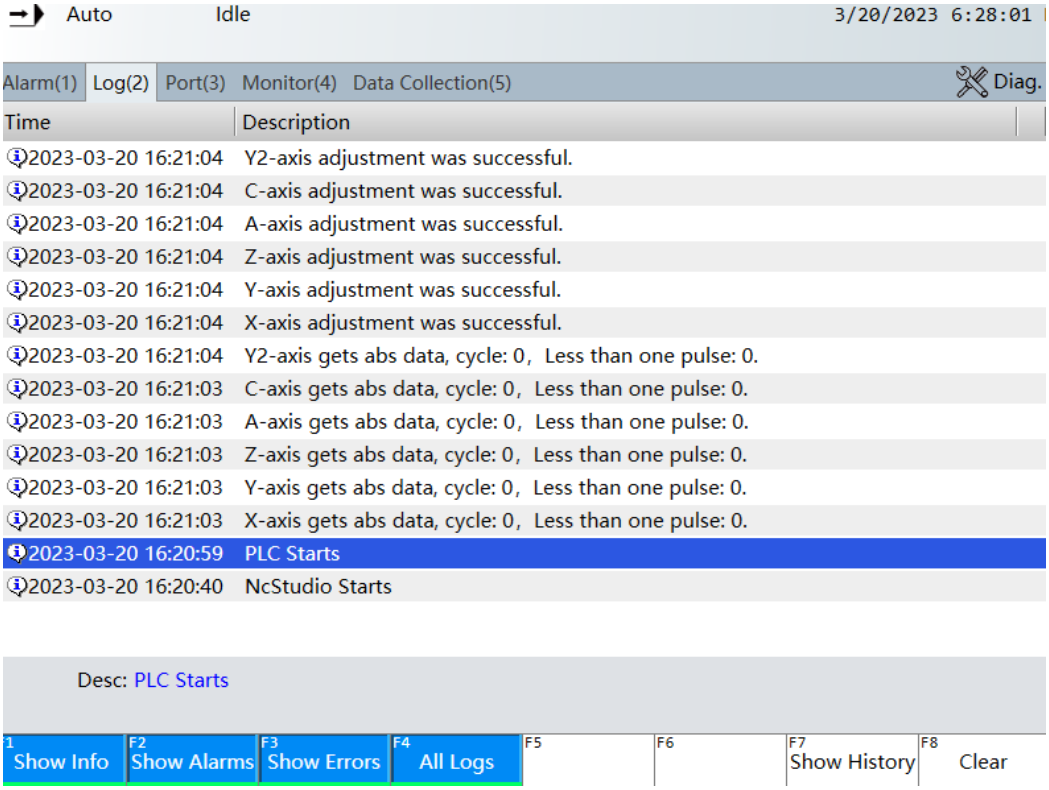

3. Press **F8** to return to the **Register** dialog box, enter the **Registration Code** in the registration code input box, and press **F7**.

## 6.2 View Log

The log records important user operations, system events, and time, including information and historical information after the system startup.

**Operation Steps:**

1. Press  → **2** to enter the **Log** page:




**Log** dialog box, which displays all logs after this startup by default.

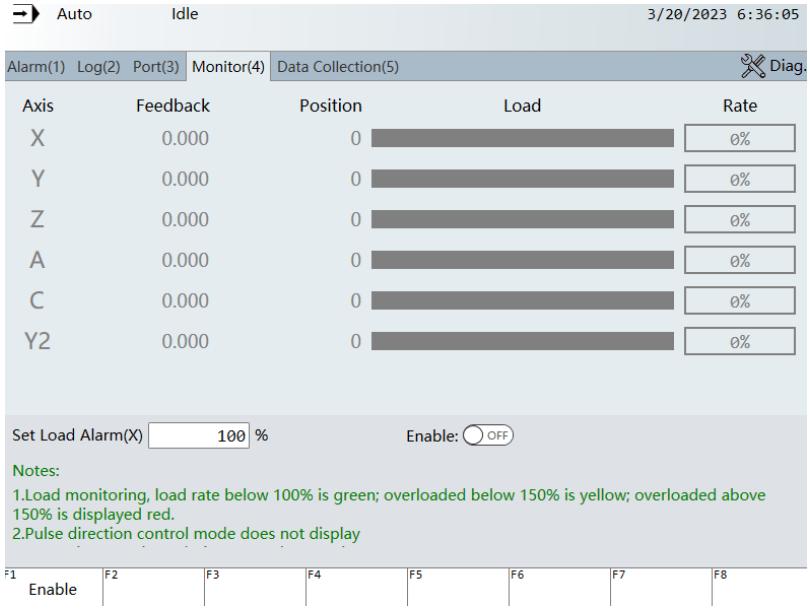
2. Select the type of log you want to view:
   - Press the **F1** button to display the software running status information with the icon.
   - Press the **F2** button to display the software warning information with the icon.
   - Press the **F3** button to display the software error fault information with the icon.
   - Press **F4** to display all corresponding log information since the system was started.
   - Press **F7** to display all logs of the software since installation.
3. (**Optional**): To delete all log information, press **F8**.

## 6.3 Monitoring Load

Check the load condition of each axle to avoid overload.

**Operation Steps:**

1. Press  → **4** to enter the **Monitor** page:



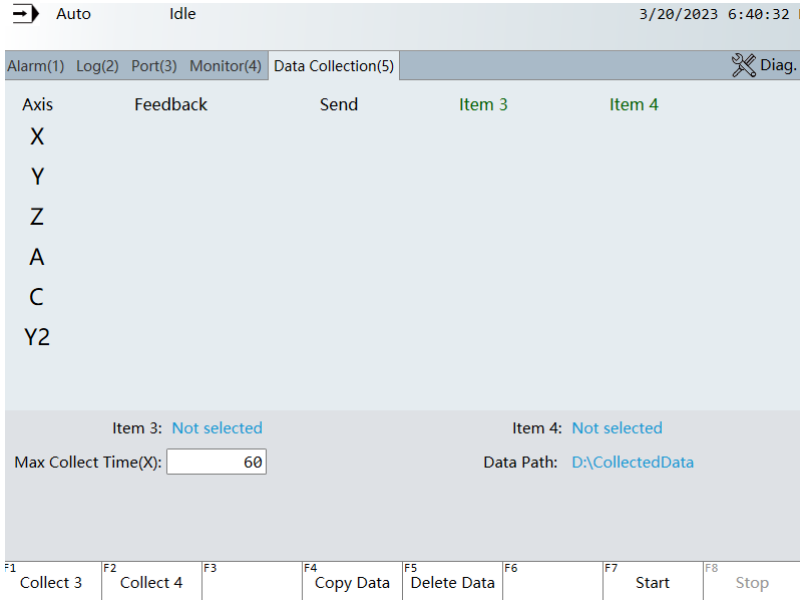

2. Press **F1** to enable.
3. Press **X** to set the load alarm threshold.

## 6.4 Collect Data

The data collection function is mainly used when the machining effect is not good, and further analysis is needed through the collected data. It is a software pulse transmission problem or a servo tracking problem. The main purpose of the collection item is to collect the feedback position of the mechanical coordinate system and the target position of the command coordinate system.

**Operation Steps:**

1. Press  → **5** to enter the **Data Collection** page:



2. (**Optional**): Press **F1**/ **F2** to set **Item 3**/ **Item 4**.
3. Press **X** to set the parameter **Max Collect Time**, and press **F7** to confirm.
4. (**Optional**): According to actual needs, perform the following operations:
   - Press **F4** to copy the collected data to the USB disk.
   - Press **F5** to delete the collected data.
5. Press **F7** to start collecting data.
6. (**Optional**): To stop collection, press **F8**.

## 6.5 Upgrade Software

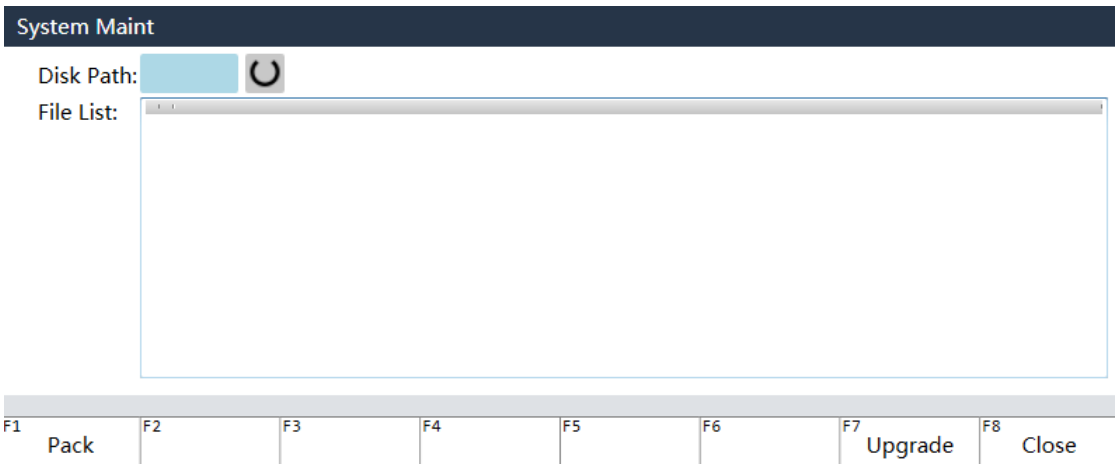
Used to update the software version, and maintain the current parameter settings after the upgrade.

**Prerequisite:**

- The system is in an idle state.
- The USB disk with the software upgrade package ready has been inserted into the USB slot of the system host.

**Operation Steps:**

1. Press  → **1** → **F2** to open the **System Maint** dialog box:




2. When the focus is on the **Disk Path**, press 

to select **Disk Path**.
3. (**Optional**): Press **F1** and the system will automatically package the current software and store the packaged backup software to the selected disk.
4. Press ↑ / ↓ to select the software to be installed in the file list.
5. Press **F7** to upgrade the software.

# Legal Notices

The term “Software Product” includes all copies of the licensed software and its documentation. This license agreement is a legal agreement between You (either an individual, a legal entity or any affiliated companies or other entities) and Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). By installing, copying, or otherwise using the Software Product, you agree to be bound by the terms and conditions of this license. Unless otherwise stated in this agreement, you shall not use, copy, revise, rent, or transfer the Software Product or any part of the Software Product for any other purposes.

## 6.6 Requirements of Use

1. You may install for use one copy of the Software Product on a single machine;
2. You may make a copy of the Software Product for archive or backup purposes and the copy is to be used on this machine only;
3. You may transfer the Software Product and the license agreement to a third party, provided that the third party accepts the terms and conditions stated in this agreement, with prior express permission from Weihong Company;
4. When transfer confirmed, you shall transfer all the copies of the original documents and the supplementary documents to the third party or destroy all the copies un-transferred;
5. You can use the Software Product on a network server or intranet server only if it is stipulated in explicit terms that you are allowed to use the Software Product on a network server or intranet server, or you have purchased license for each node and terminal using the Software Product;
6. You may NOT sublicense, assign or transfer the license agreement;
7. You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
8. You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
9. The license agreement shall be terminated automatically upon you transfer the Software Product or copies of all or part of the Software Product to a third party.

## 6.7 Intellectual Property Rights Notice

The Software Product and all intellectual property rights therein (including but not limited to any all copyrights, patents, trademarks, and publicity rights) are owned by Weihong Company. The Software Product is protected for Weihong Company on the basis of Chinese copyright law and international treaty provisions as well as on the basis of other laws and agreements regarding intellectual property. You are not allowed to remove the copyright statement made in the Software Product, and guarantee that you shall copy the copyright statement in all copies of the Software Product or of any part of the Software Product. You are obliged to stop any form of illegal copying of the Software Product and accompanying materials.

Weihong Company can update the Software Product at any time. You can pay real-time attention to Weihong Company's official website.

### 6.8 Termination

This license may be terminated by Weihong Company at any time once you violate any terms or conditions made in this agreement. Once the license is terminated, you are obliged to destroy all the copies of the Software Product or return them to Weihong Company.

Now, we affirm that you have already read through this agreement and understood it thoroughly and agreed to comply with all of the terms and conditions of this agreement strictly.

Weihong Electronic Technology Co., Ltd.