# Manufacturer's Manual of Ncstudio Phoenix Wood Three-Axis Engraving System

Version: 2021.12.01 1st Version
Author: Product Testing Department



## Contents

# 1 System Introduction

## 1.1 Overview

This section gives you a brief introduction of the hardware and software of the **NcStudio Phoenix Three-axis Engraving System**.

**Hardware**

- Industrial computer: NC60A, NC65C, NC68A, etc.
- Lambda controller
  - Lambda 21B: Applicable to bus control systems
  - Lambda 21A: Applicable to non-bus control systems
  - Lambda 20A and so on
- EX series terminal board: EX31A

For the hardware connection diagram, see section 1.2 Hardware Connection Diagram.

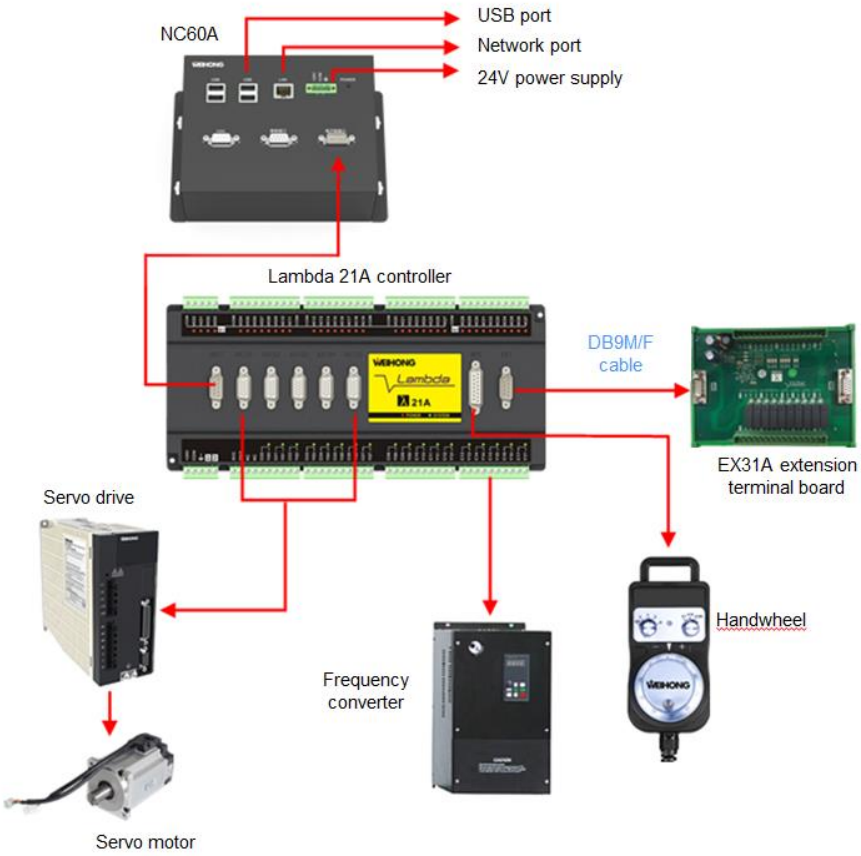
**Software**

For introduction of the **NcStudio Phoenix Three-axis Engraving System** software interface, see section 1.3 Software Interface.

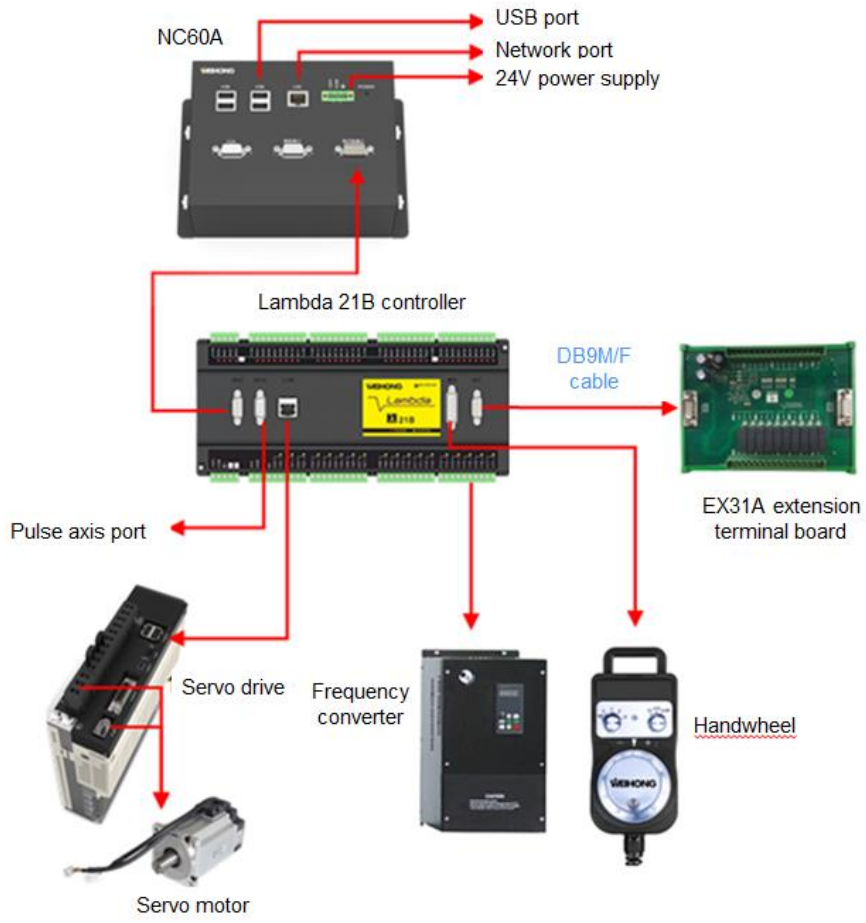
## 1.2 Hardware Connection Diagram

Hardware connection method varies based on the control system type. Example of a non-bus control system and a bus control system using NC60A are shown below.

**Non-bus control system**




**Bus control system**

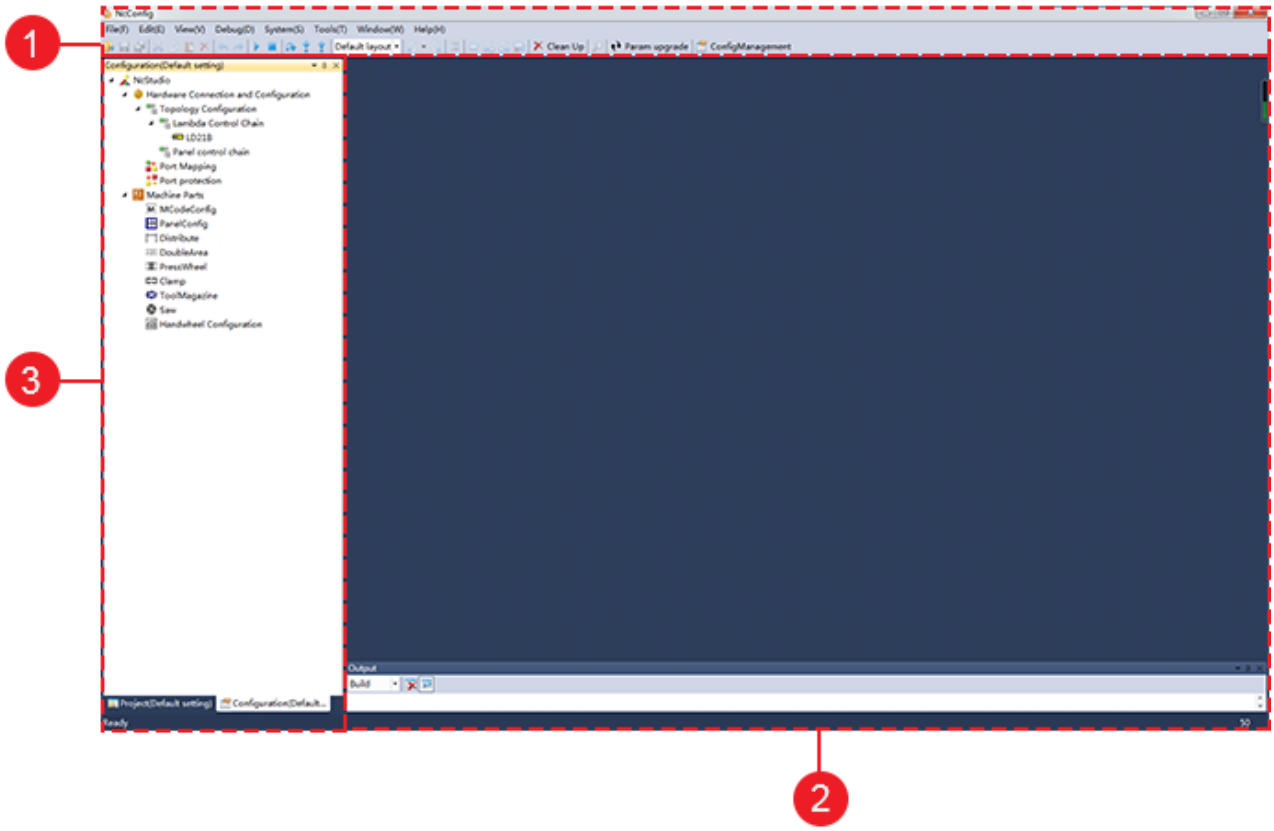



## 1.3 Software Interface

This section gives you a brief introduction of the software interface of the **NcStudio Phoenix Three-axis Engraving System**.

### 1.3.1 NcConfig

The homepage of NcConfig is shown below:




1. Menu bar
2. Function area
3. Project configuration area

#### *1.3.1.1 Function Area*

Target functions are selected in the **project configuration area** and debugged in the **function area**.

#### *1.3.1.2 Project Configuration Area*

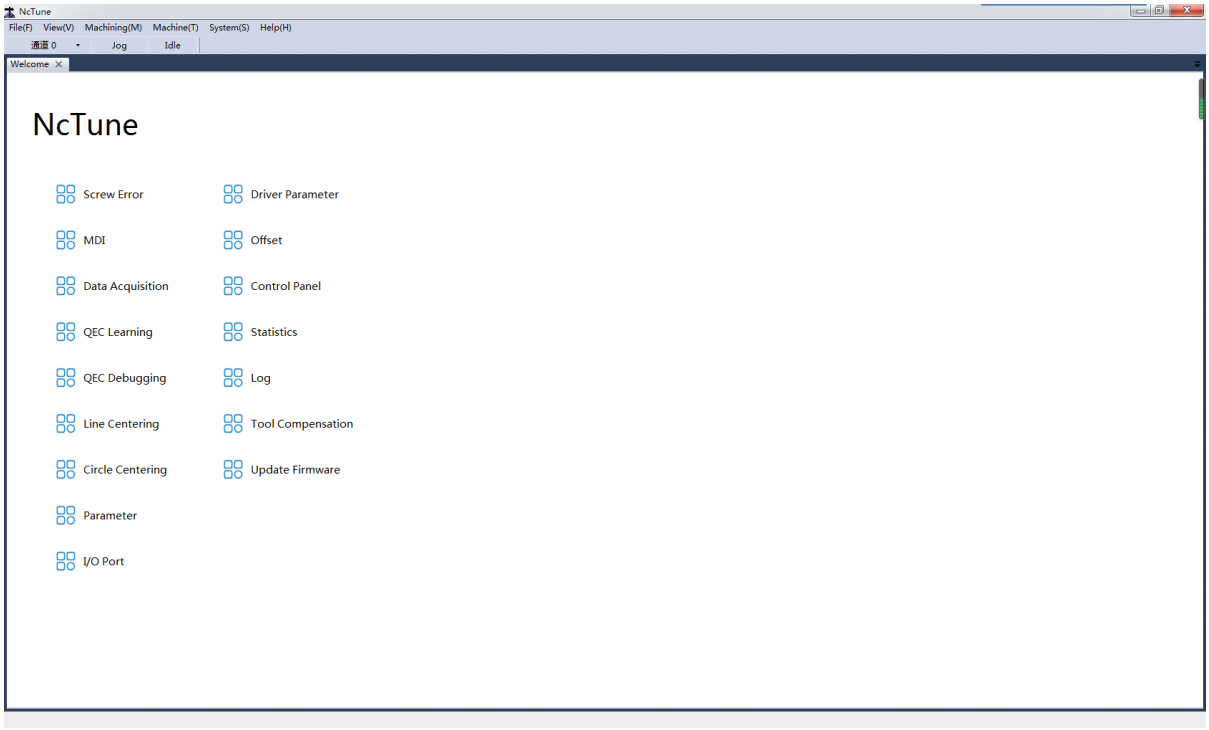
Click the **Project** or **Configuration** tab in the project configuration area's lower part to use different functions (If the **Project** tab is not displayed in this area, go to **View** > **Project** in the menu bar).

Functions on the **Configuration** page:

- Topology configuration
- Machine parts
- Port mapping
- Port protection

### 1.3.2 NcTune

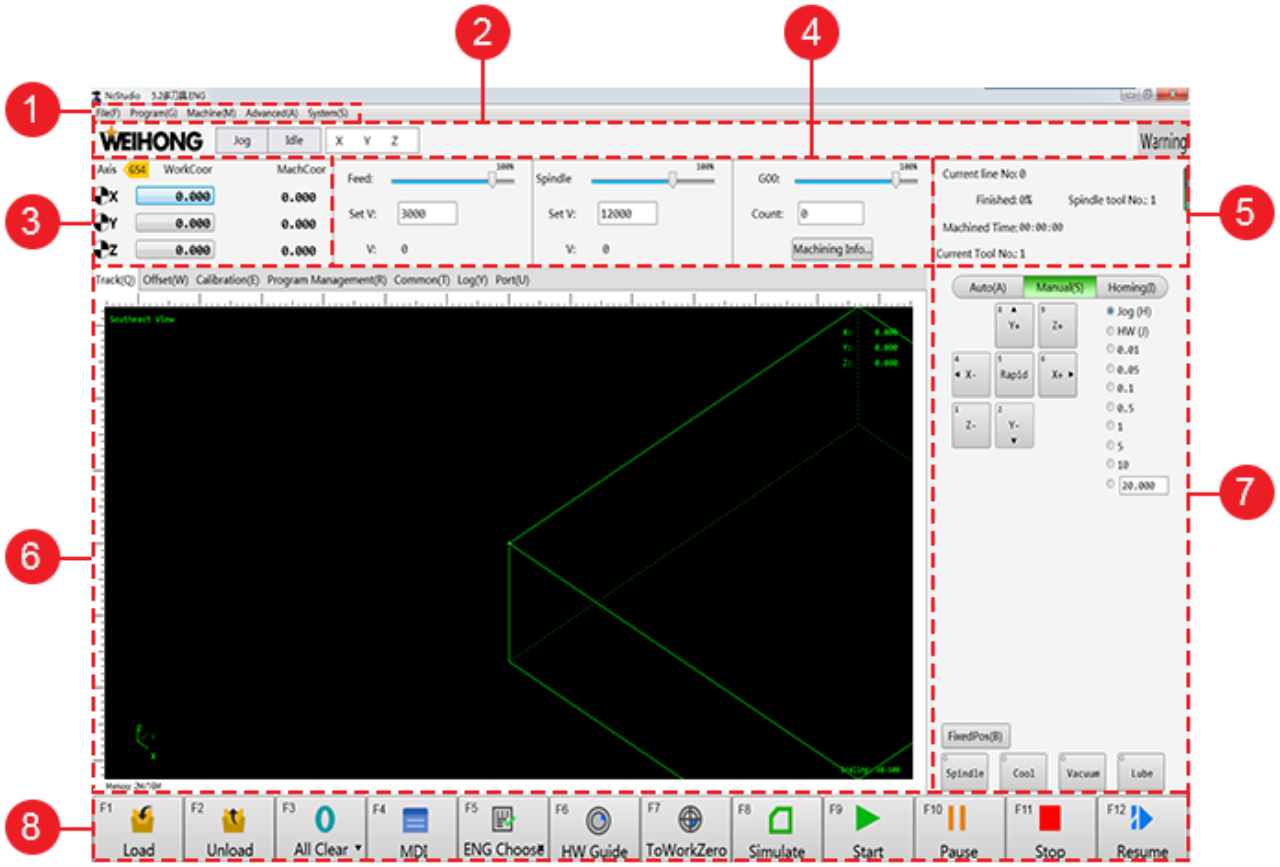
The homepage of NcTune is shown below:




1. Menu bar
2. Status bar
3. Welcome page

### 1.3.3 NcStudio

The homepage of NcStudio is shown below:




1. Menu bar
2. Status bar
3. Axis coordinate area
4. Override setting area
5. Machining information area
6. Function window
7. Operation mode area
8. Operation button bar

### *1.3.3.1 Menu Bar*

You can access the following functions in the menu bar:

- **File**: Open and Load, Load History File, Unload, Generate Installation Package, Multi-Config Pack Up, Restart software, Close System, Restart System, Show Desktop, and Exit
- **Program**: Machining Wizard, Machining Information, and Machining Statistics
- **Machine**: Single Block Execution, HW Guide, Current Point as Work Zero, Cycle Start, Cycle Pause, Cycle Stop, Simulate, Resume, Jiggle, Homing, To Work Zero, To Fixed Point, Centering, MDI, Datum Setting, Adjustment, Datum Cancel, Double Y Jiggle, Double Y Origin Detection, Clear Servo Alarm, Reconnect Drive, and Reconnect Controller
- **Advanced**: Selective Machining, ENG choose tool and row, AdvAuto, Cycle Setting, Timed Stop, and Nearby Resume
- **System**: Log, Global Parameters, Driver Parameters, Tool Compensation Parameters, Tool Offset, Change Password, Language, Multi-Z Mode Switching,

Data Backup, NcTune, Remote Assistance, NcCloud, M Instruction Table, and About

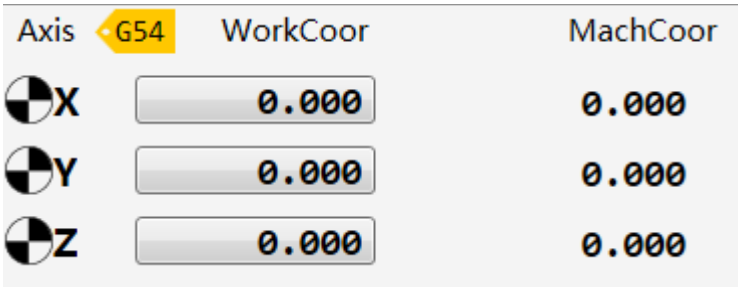
### *1.3.3.2 Status Bar*

You can access the following functions in the status bar:

- System modes: Auto, Manual, and Homing
- System status: Idle, running, paused, or emergency stop.
- Information/alarms: Current information and alarms
  Double-click the blank area to open the **Log** dialog box to view system alarms, errors and information.

### *1.3.3.3 Axis Coordinate Area*

The axis coordinate area is shown below:

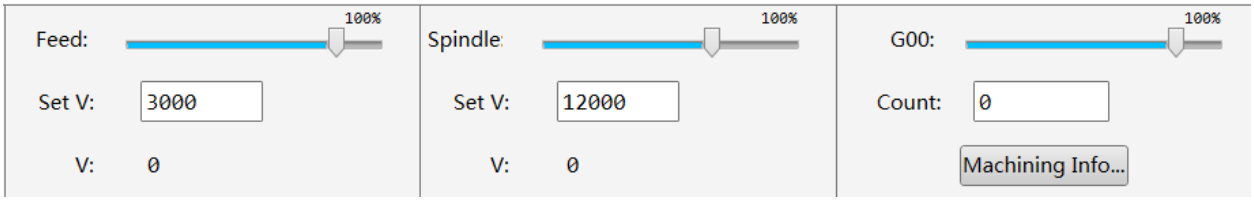



It shows the name of the current workpiece coordinate system, and the mechanical and workpiece coordinates of the axes.

After an axis has gone to the machine origin, the icon will be displayed before the axis in this area.

### *1.3.3.4 Override Setting Area*

The override setting area is shown below:




- In this area, you can set the spindle rotational speed, feed rate and G00 speed parameters by entering values in the corresponding field.
- You can adjust the spindle rotational speed, feed rate and G00 override/multiplier by dragging the corresponding slider.

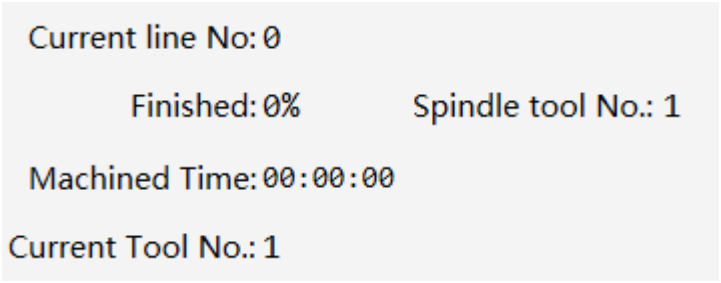
Actual spindle rotational speed = Spindle rotational speed multiplier * Spindle rotational speed setting

Actual machining speed = Feed rate multiplier * Feed rate setting

Actual G00 speed = G00 multiplier * G00 setting

### *1.3.3.5 Machining Information Area*

The machining information area is shown below:




It shows the current machining progress, current program line number, machining duration and current tool number.

#### *1.3.3.6 Function Window*

Click the following tabs in the function window to access different functions:

- **Track**: Displays the machining or dry running track.
- **Offset:** Setting of the workpiece offset and public offset, Z-axis lowing and lifting
- **Calibration**: Setting of the tool sensor position and tool calibration method (**Auto Measure**, **Z Mobile Cali**, **Z Manual Set**)
- **Program Management**: Loading, editing, deleting, renaming and creating programs
- **Common**: Viewing and setting of common parameters
- **Log**: Viewing of information, alarm, and error logs.
- **Port**: Viewing of port status and polarity, testing of ports, modification of port polarity, and setting of the filter

#### *1.3.3.7 Operation Mode Area*

The system supports the following operation modes:

- **Auto**
  - The loaded program is displayed in this mode.
  - Common function: Going to the fixed point
  - Common ports: **Spindle, Cool, Vacuum**, and **Lube**. The common ports can be modified in NcConfig.
- **Manual**
  - Axis buttons: control of the axis movement in each direction
    - **Jog**
      - Click and hold an axis button to make the target axis move in the target direction continuously at the relatively lower speed.
      - Click and hold multiple axis buttons at the same time to make the target axes move in the target directions continuously at the relatively lower speed.
        Mainly used to move the X axis and Y axis at the same time.
      - Click and hold an axis button and the **Rapid** button at the same time to make the target axis move in the target direction continuously at the relatively higher speed.
      - Click and hold multiple axis buttons and the **Rapid** button at the same time to make the target axes move in the target directions continuously at the relatively higher speed.
    - **HW**: In this mode, you use the handwheel to control the machine movement. Select the axis direction and multiplier on the handwheel

and turn the handwheel knob by a certain degree to make the target axis move in the target direction.

- **Stepping**: Select **0.01, 0.05, 0.1, 0.5, 1, 5**, or **10** (the unit is mm), or enter a value in the blank field. Click and release an axis button to make the target axis move in the target position by the set step.
  **Note**: Please do not set the step to a value that's too large to avoid damage due to misoperation, and do not click an axis button too frequently in short time.

- Common function: Going to the fixed point
- Common ports: **Spindle, Cool, Vacuum, Lube**, etc. The common ports can be modified in NcConfig.

- **Homing**
  - Tick **Execute homing when software starts** to make the axes go to the mechanical origin upon software start.
  - You can also check and clear the homing average values in this area.

#### *1.3.3.8 Operation Button Bar*

You can use the following functions with buttons in the operation button bar:

In **Auto** and **Manual** modes:

- Loading and unloading of programs
- Axis clearing
- MDI, selective machining, going to the workpiece origin, and simulation
- Using the handwheel to control machining movement
- Starting, pausing, or stopping machining; resuming machining from the interrupted point

In **Homing** mode:

- Making the axes go to the mechanical origin
- Starting, pausing, or stopping machining; resuming machining from the interrupted point

# 2 Wiring

This section gives you a brief introduction of the signal types supported by the **NcStudio Phoenix Three-axis Engraving System** and the port definition.

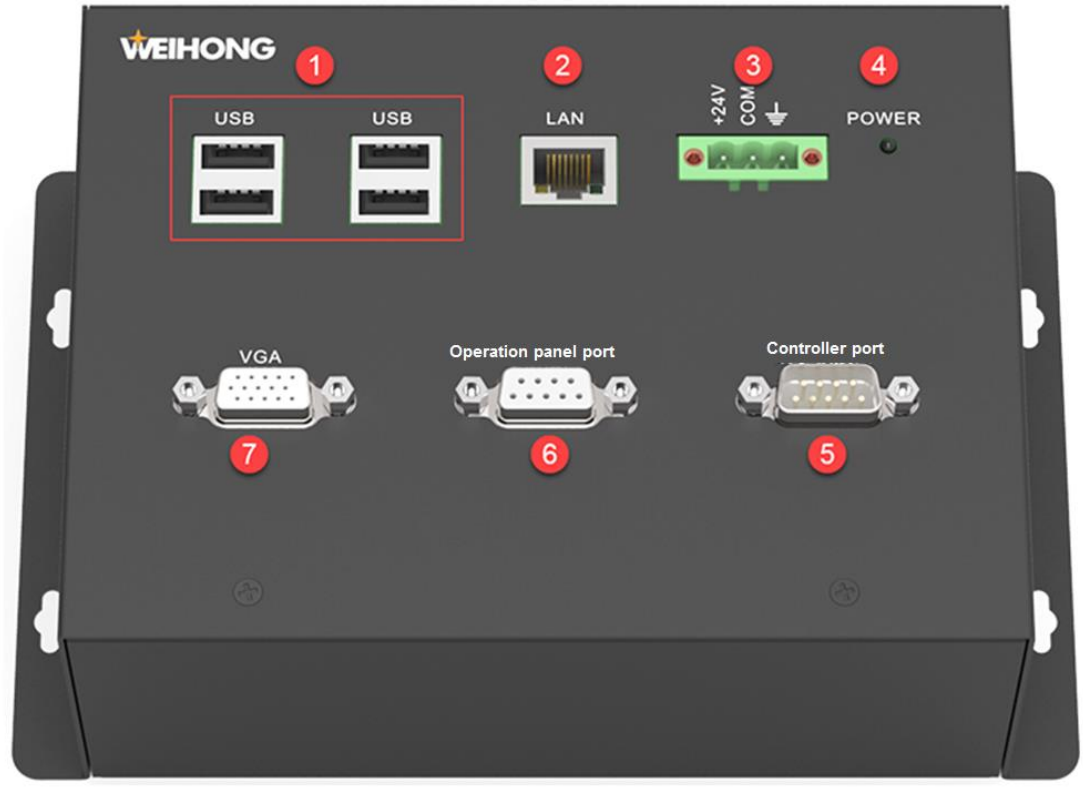
## 2.1 Hardware

This section gives introduction of the following hardware:

- NC60A
- Lambda controller
- EX31A

### 2.1.1 NC60A

Definitions of the NC60A ports are shown below:




1. USB 2.0 port: For USB data transmission
2. Network port: For network connection
3. 24V DC power supply port: For connection to a 24V DC power supply
4. LED indicator: Shows the current system status
    - ON: The system is powered on
    - OFF: The system is not powered on
5. Lambda controller port: Phoenix bus RS485 port. For connection to the Lambda controller.
6. Operation panel port: Phoenix bus RS485 port. For connection to the operation panel.
7. VGA display port: For connection to the display

### 2.1.2 Lambda Controller

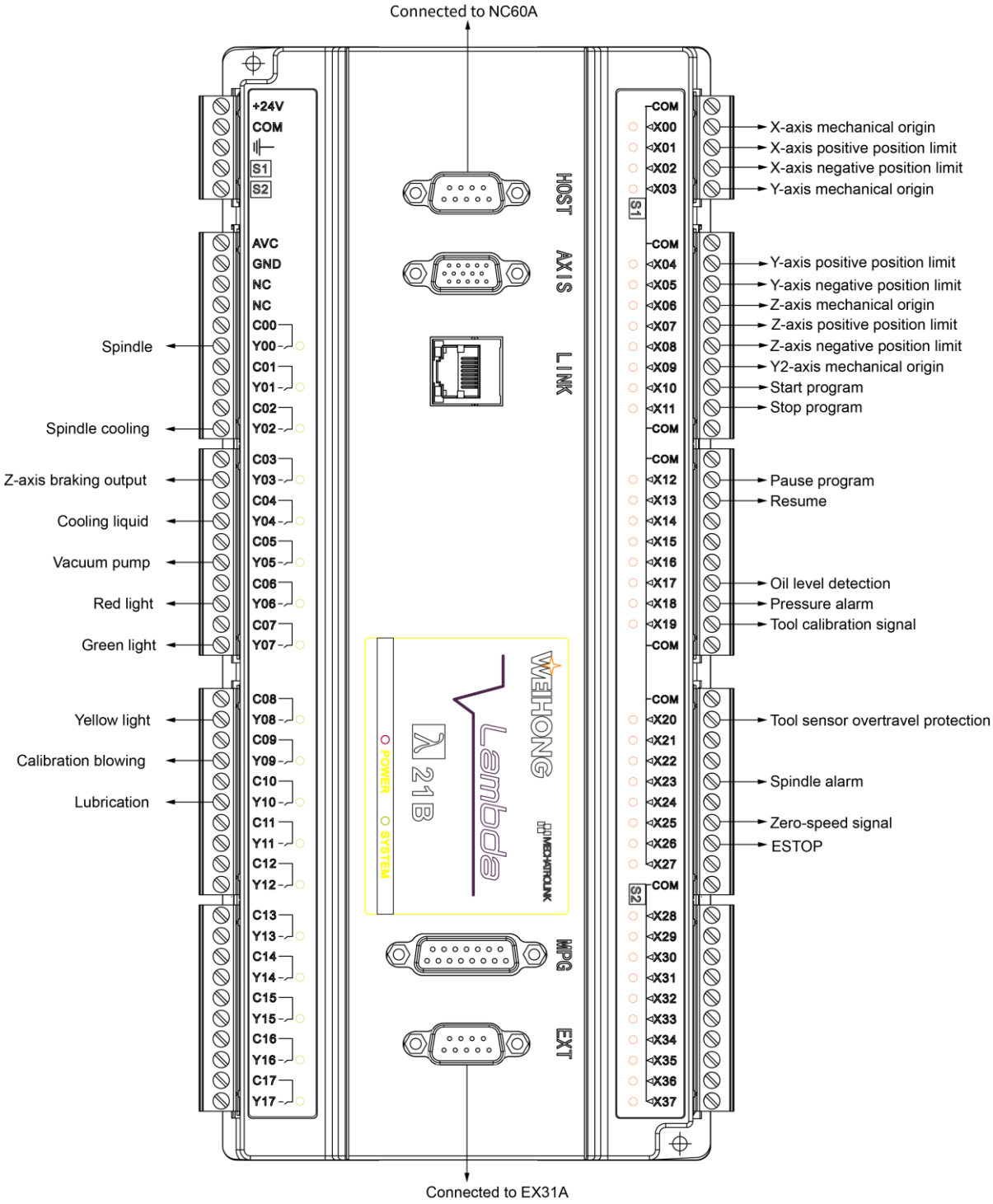
Use different Lambda controllers based on the type of the control system:

- Lambda 21B: Applicable to bus control systems
- Lambda 21A: Applicable to non-bus control systems

Definitions for Lambda 21B and Lambda 21A ports are basically the same; therefore, the following content introduces port definitions for the Lambda 21B controller as an example.

Multiple EX31A extension terminal boards can be connected to the Lambda 21B controller to meet your needs.

The port definitions are shown below:



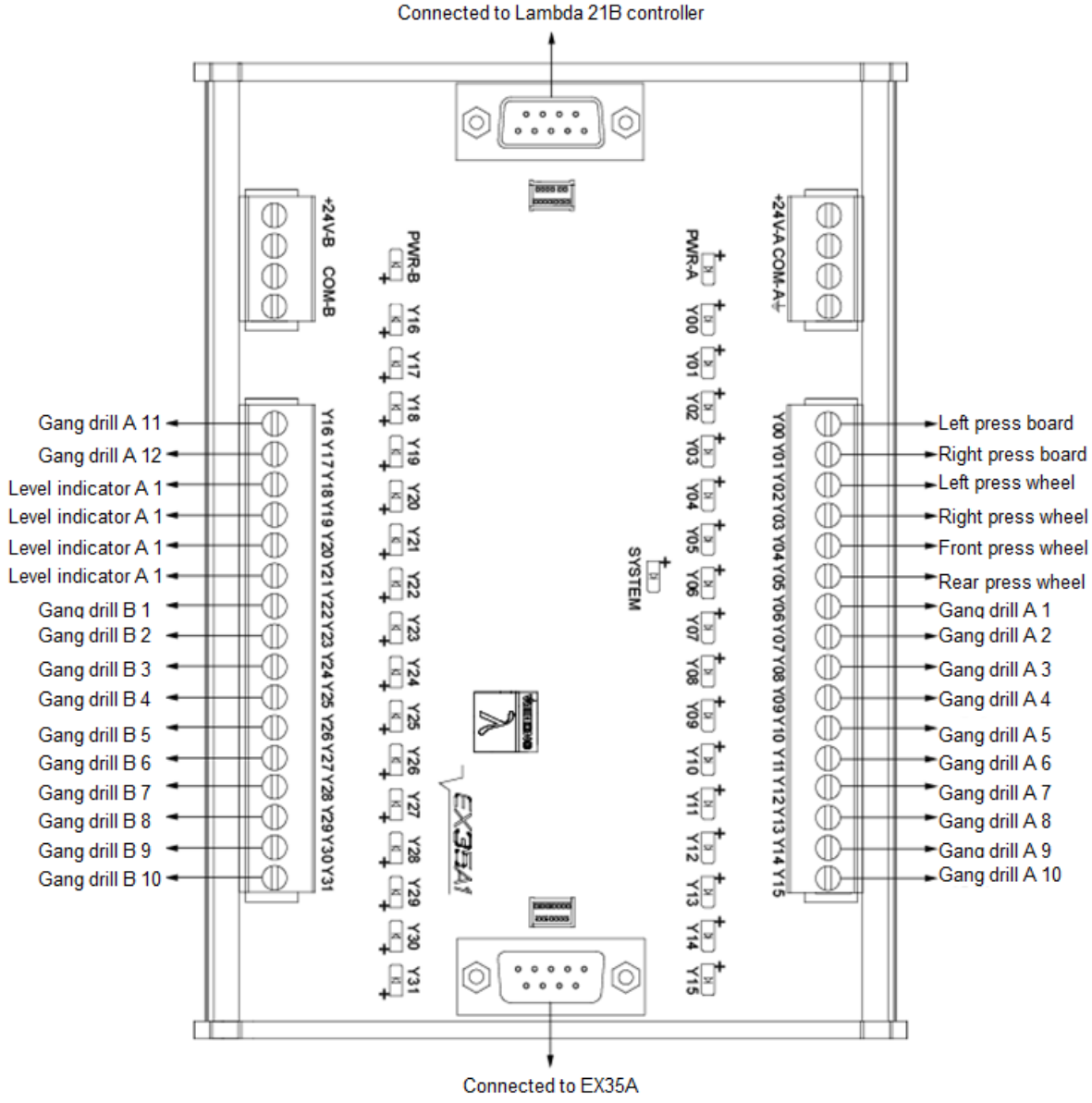

### 2.1.3 EX31A

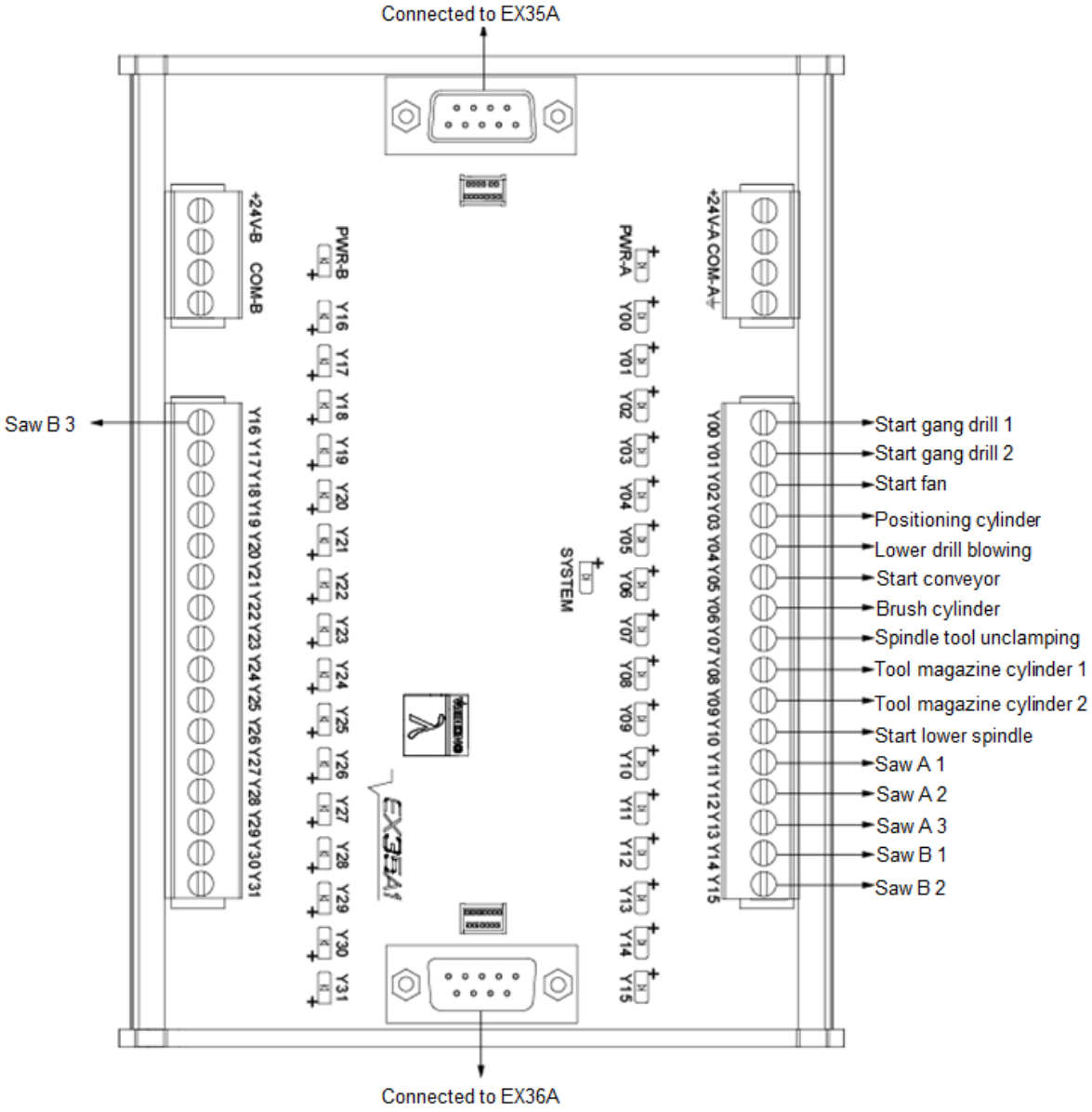
Select extension terminal boards based on your needs.

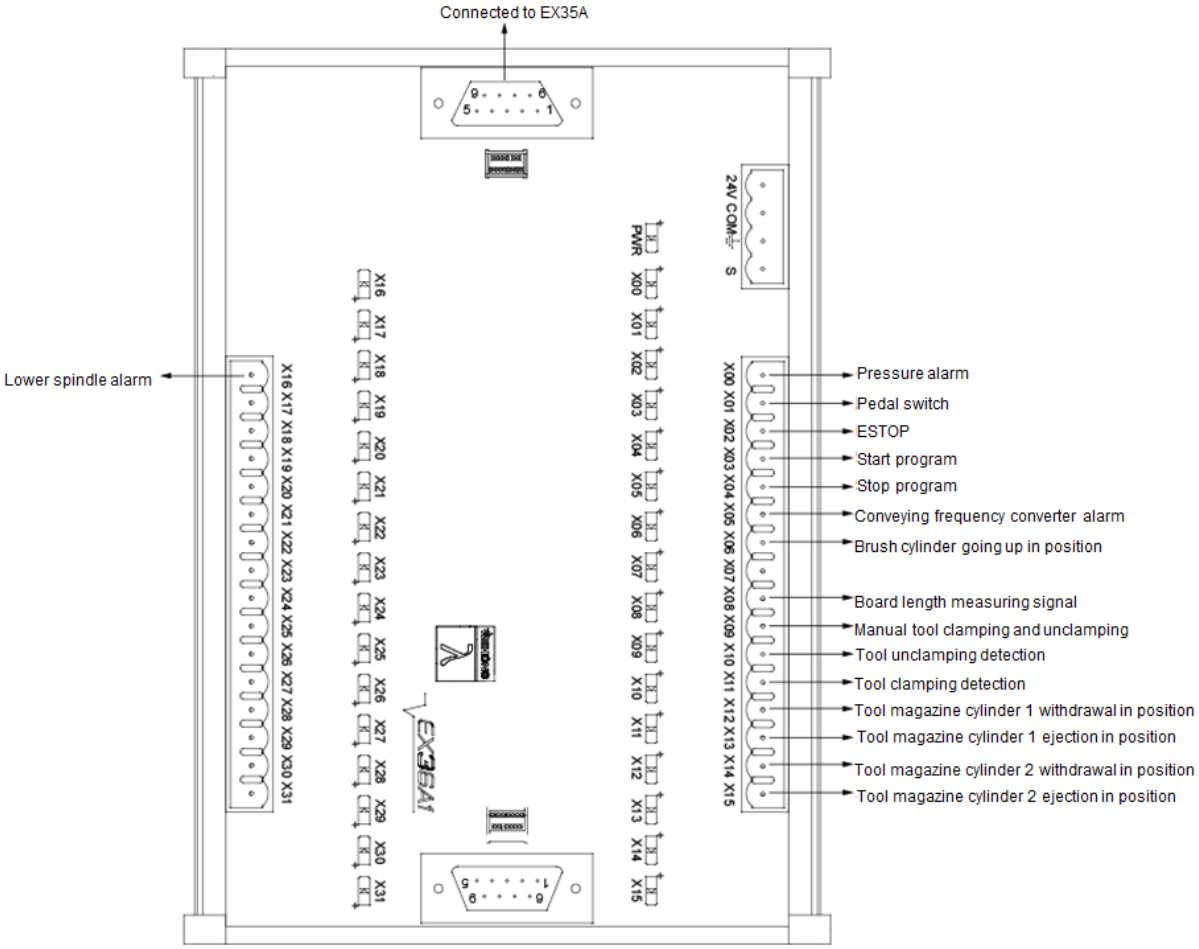
- EX31A: Supports 10-channel input and 10-channel output

- EX35A: Supports 32-channel output
- EX36A: Supports 32-channel input

Port definitions of the Lambda 21B connected to two EX35A terminal boards and one EX36A terminal board are shown below as an example:

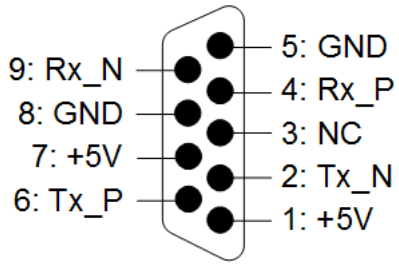



Connected to EX35A
+24V-B COM-B
PWR-B Y16 Y17 Y18 Y19 Y20 Y21 Y22 Y23 Y24 Y25 Y26 Y27 Y28 Y29 Y30 Y31
PWR-A Y00 Y01 Y02 Y03 Y04 Y05 Y06 Y07 Y08 Y09 Y10 Y11 Y12 Y13 Y14 Y15
SYSTEM
+24V-A COM-A
Saw B 3
Y16 Y17 Y18 Y19 Y20 Y21 Y22 Y23 Y24 Y25 Y26 Y27 Y28 Y29 Y30 Y31
Y00 Y01 Y02 Y03 Y04 Y05 Y06 Y07 Y08 Y09 Y10 Y11 Y12 Y13 Y14 Y15
Start gang drill 1
Start gang drill 2
Start fan
Positioning cylinder
Lower drill blowing
Start conveyor
Brush cylinder
Spindle tool unclamping
Tool magazine cylinder 1
Tool magazine cylinder 2
Start lower spindle
Saw A 1
Saw A 2
Saw A 3
Saw B 1
Saw B 2
Connected to EX36A




## 2.2 Operation Panel Ports

Descriptions of the operation panel ports are shown below:




- +5V: Provides power supply for the operation panel.
- Tx_N: Sends differential signal.
- NC: Needs no wire connection.
- Rx_P: Receives differential signal.
- GND: Connects to the ground.
- Tx_P: Sends differential signal.
- Rx_N: Receives differential signal.

## 2.3 Signal Type

The following signal types are supported:

- Switch Value Input Signal
- Relay Output Signal

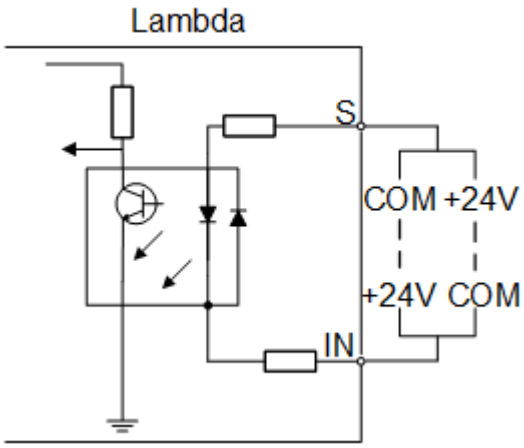
- Analog Output Signal
- Command Pulse Signal

### 2.3.1 Switch Value Input Signal

The switch value input signal supports active high and active low:

- Connecting to the COM port in NO mode means to receive signal.
- Disconnecting with the COM port in NC mode means to receive signal.

The figure is as follows:




The input port supports active high and active low:

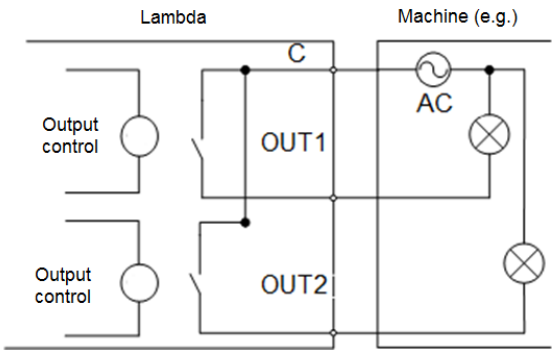
- When the common terminal board S of the Lambda controller is connected to the COM port, inputs are active in high voltage.
- When the common terminal board S of the Lambda controller is connected to the +24V port, inputs are active in low voltage.

### 2.3.2 Relay Output Signal

The Lambda controller outputs relay signals.

The load capacity of relay output contact points is AC 7A/250V or DC 7A/30V. If high power load is needed, a contactor should be connected as follows:

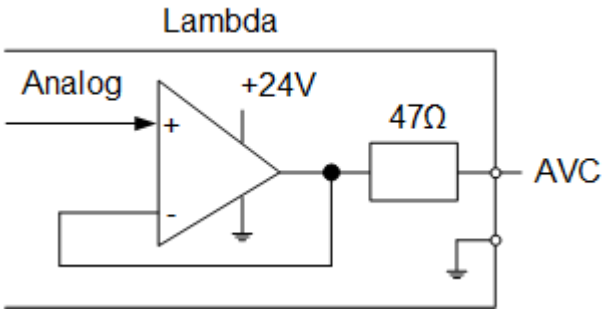
The figure is as follows:




### 2.3.3 Relay Output Signal

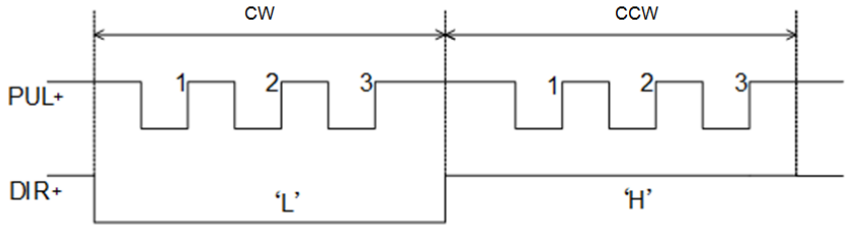
The AVC port, externally connected to the inverter's analog voltage frequency command input port, can output controllable voltage from 0V to 10V. By changing the voltage, it controls the inverter frequency and in turn the spindle speed.

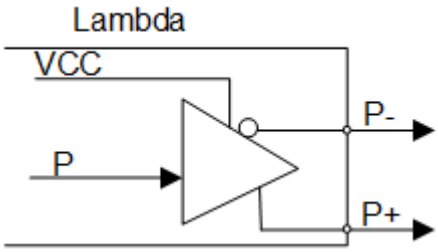
The figure is as follows:




### 2.3.4 Command Pulse Signal

Pulse commands that control the controller's motion consist of pulse + direction, negative logic, as shown below. The maximum pulse frequency is 1MHz.



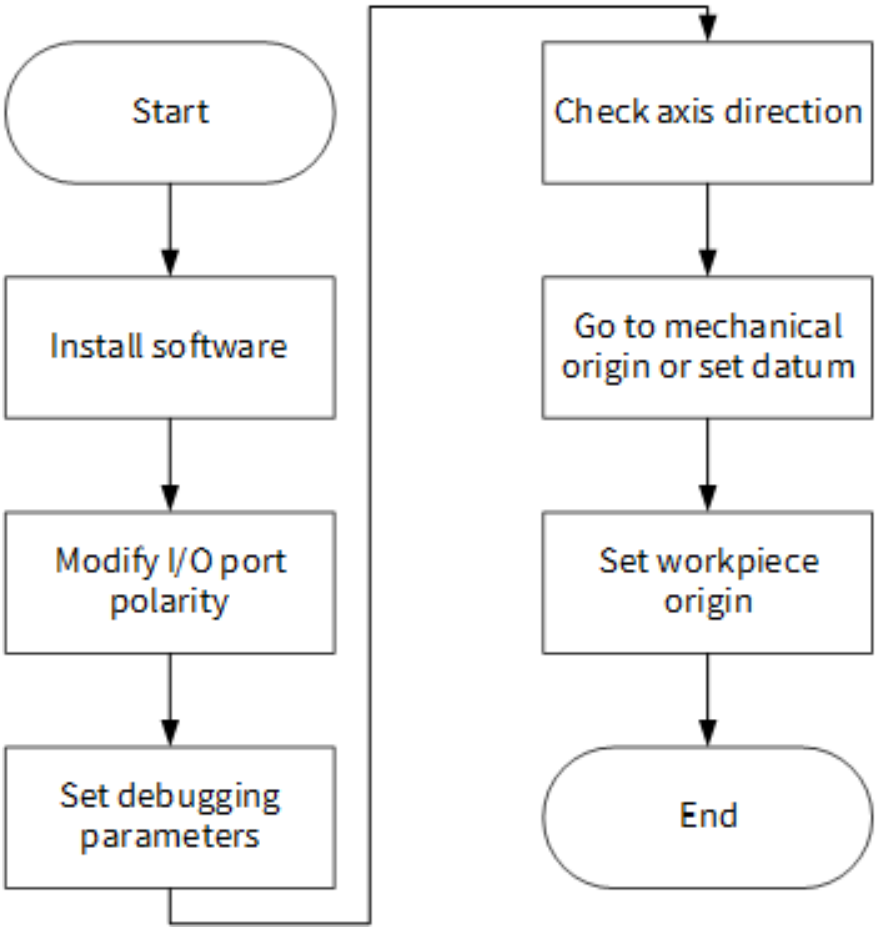



# 3 Machine Debugging

## 3.1 Overview

This section gives you a brief introduction of the basic debugging process for the **NcStudio Phoenix Three-axis Engraving System**.

The debugging process is shown below:




If the manufacturer's password is required during debugging, contact the manufacturer.

## 3.2 Install the Software

Before installing the software:

- Ensure that the machine can be powered on normally.
- The USB flash drive storing the installation package is connected to the USB port of the NC65C host.

Follow the steps below to install the software:

1. Double-click the installation package. The installation wizard window is displayed.
2. Select the installation language. Go to **OK** > **Next**.
3. After all settings are finished, click **Finish**.

## 3.3 Modify I/O Port Polarity

After the software is started, modify the polarity of the I/O ports based on the type of the detection and position limit switches:

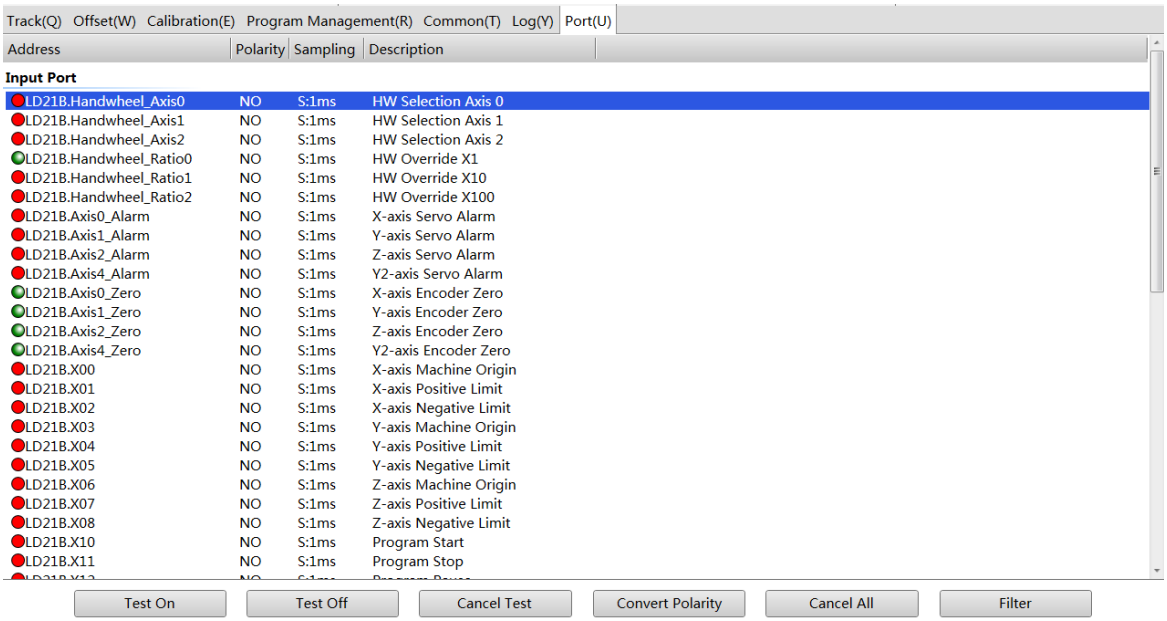
- Set the polarity of normally closed switches to **NC**.
- Set the polarity of normally open switches to **NO**.

The relationship between the signal status and the port symbols is shown below:

- Input port: 🔴 no signal is detected; 🟢 signal is detected
- Output port: 🔴 no signal is detected; 🟢 signal is detected

Follow the steps below to modify the I/O port polarity:

1. Click the **Port** tab in the function window:



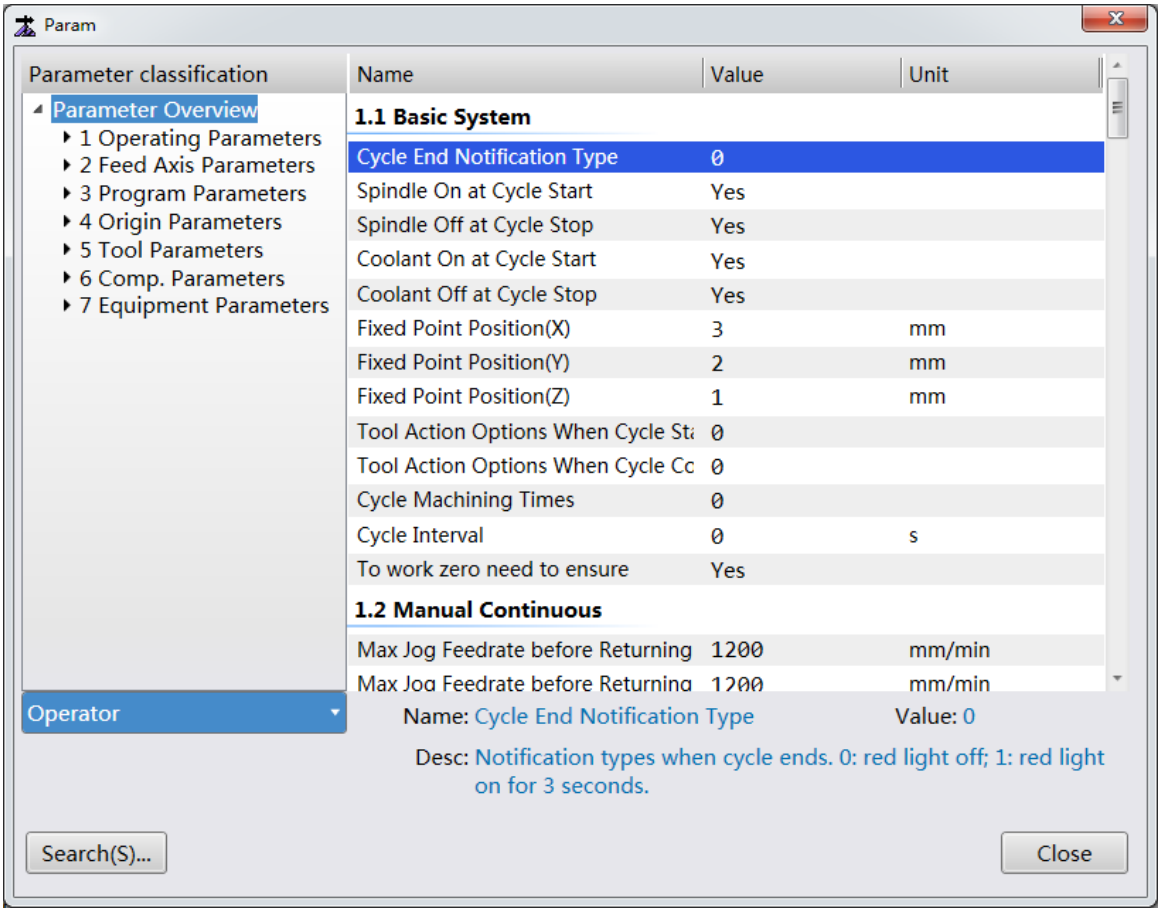

2. Click the target port and click **Convert Polarity** to reverse the current polarity.
3. The following functions are also supported:
    - To test port signal output: Click **Test On/Test Off**. When the port symbol becomes 🟢 or 🔴, the port is under test.
    - To cancel testing of the selected port: Click **Cancel Test**.
    - To cancel testing of all ports: Click **Cancel All**.
    - To set sampling interval, click **Filter** and drag the **Interval** slider. The system will rule out signals whose duration is smaller than the interval.

## 3.4 Set Debugging Parameters

Debugging parameters need to be set for bus control systems to prevent machine motion from causing unintentional damage.

Follow the steps below to set the debugging parameters:

1. In the menu bar, go to **System** > **Global Parameters**.



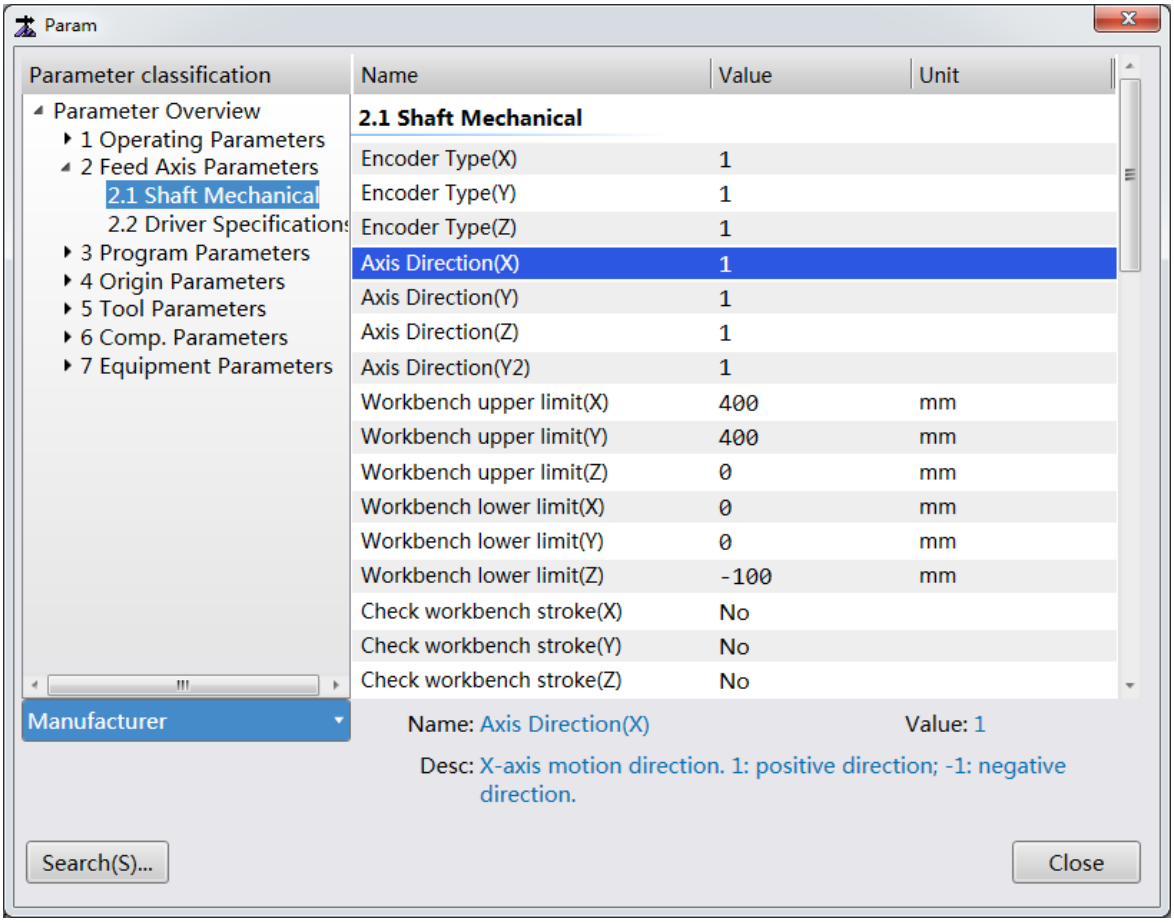

2. In the **Param** window, click the **Operator** pull-down menu in the left column and select **Manufacturer**.
3. Set the following parameters:
   - **Encoder Direction**: Indicates the direction of the axis encoders. 1: positive direction; -1: negative direction
   - **Encoder digits**: Indicates the digit number of the servo motor encoder. Value range: 10–29.
   - **Electronic gear ratio(numerator)/Electronic gear ratio(denominator)**: Indicates whether the servo drive enlarges or shrinks the received pulse frequency from the upper computer. If the **Electronic gear ratio(numerator)/Electronic gear ratio(denominator)** value is larger than 1, the servo drive enlarges the pulse frequency. If the **Electronic gear ratio(numerator)/Electronic gear ratio(denominator)** value is smaller than 1, the servo drive shrinks the pulse frequency.
   - **Pitch**: The movement or rotation angle of the feed axis per lead screw turn.

## 3.5 Check Axis Direction

Determine the positive direction for the machine axes based on the right-hand rule and check to see if the axis directions are correct to avoid possible damage.

Taking the X axis as an example, follow the steps below to check the axis direction:

1. Determine the X axis positive direction based on the right-hand rule.
2. In the menu bar, go to **System** > **Global Parameters**.




3. In the **Param** window, click the **Operator** pull-down menu in the left column and select **Manufacturer**.
4. Check the **Axis Direction(X)** value.
    - 1: positive direction
    - -1: negative direction
5. In **Manual** mode, click the **X+** button to move the X axis. Check to see if the X axis moves in the positive direction as determined by the right-hand rule.
    - Yes: the axis direction is correct.
    - No: Reverse the current **Axis Direction(X)** value.

## 3.6 Go to Mechanical Origin or Set Datum

Select one from the following two operations based on the encoder type:

- Make the axes go to the mechanical origin: Applicable to incremental encoders
- Set datum: Applicable to absolute encoders

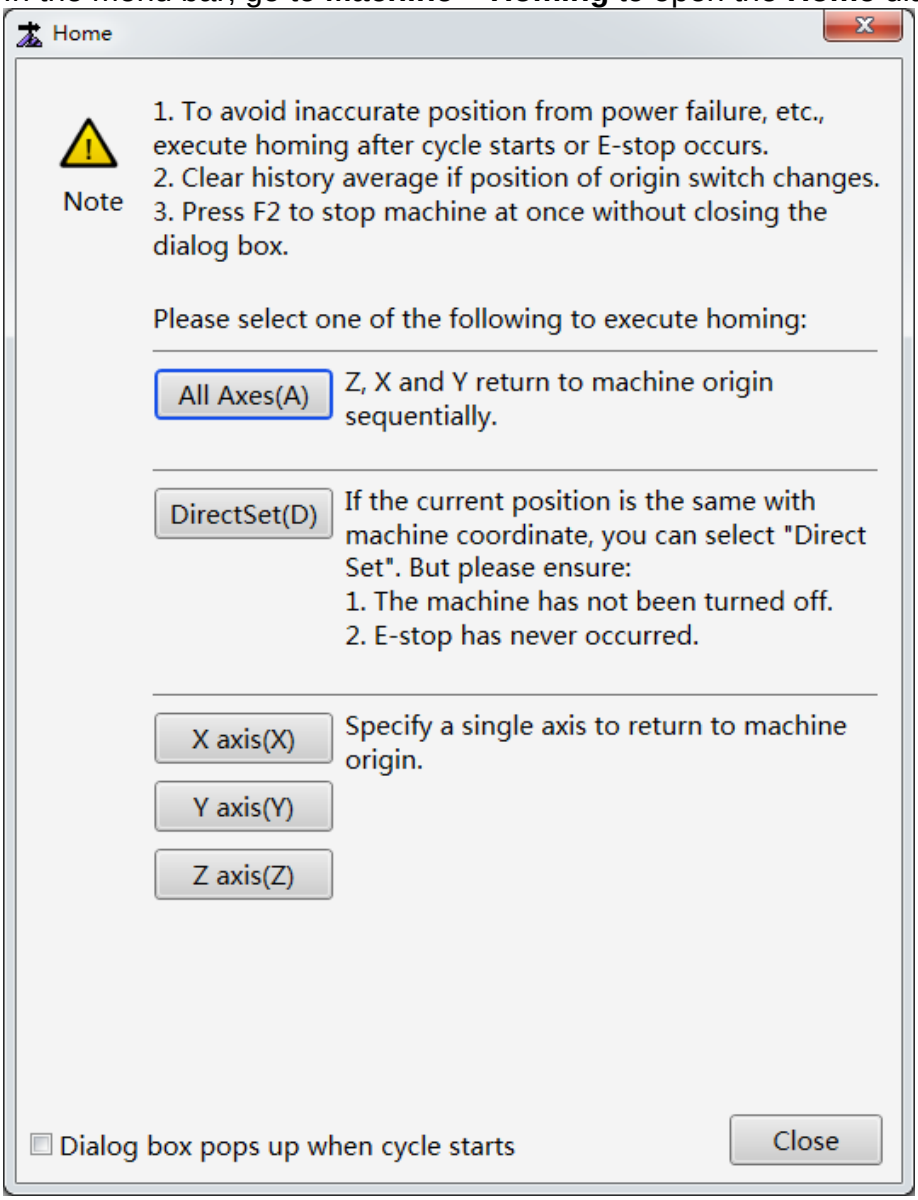
Before making the axes go to the mechanical origin or setting datum, ensure that there is no drive or motor alarm.

### 3.6.1 Making Axes Go to Mechanical Origin

The mechanical coordinate system is the inherent coordinate system of the machine itself. The origin of the mechanical coordinate system is called mechanical origin or reference point. The position of the mechanical origin/reference point is determined and fixed during the design and debugging stages before being shipped out of factory for delivery. If you are using an incremental encoder, you need to make the axes go to the mechanical origin every time the machining system is restarted before starting machining to calibrate the coordinates for more accurate machining.

Select one from the following methods to make the axes go to the mechanical origin:

- In **Homing** mode, click **X Homing/Y Homing/Z Homing** in the operation button bar to make the X/Y/Z axis go to the mechanical origin. You can also click **All Homing** to make all the axes to the mechanical origin in the sequence of Z axis, X axis and Y axis.
- In the menu bar, go to **Machine** > **Homing** to open the **Home** dialog box:




To make the **Home** dialog box automatically displayed when the software is started, tick **Dialog box pops up when cycle starts** at the dialog box lower left corner.

  - To make the X/Y/Z axis go to the mechanical origin, click **X axis(X)/Y axis(Y)/Z axis(Z)**.
  - To make all axes go to the mechanical origin, click **All Axes(A)**.
  - To set the current position as the mechanical origin: Ensure that the current position is consistent with the mechanical coordinates and there was no cases that may cause coordinate error, such as machine power-off or emergency stop. Click **DirectSet**.

After an axis has gone to the mechanical origin, the icon will be displayed before the axis in the axis coordinate area.

### 3.6.2 Setting Datum

If an absolute encoder is used, you can directly set the encoder zero conveniently without having to make the axes go to the mechanical origin.

If the system has restarts, power-off, emergency stop or similar situations, the system will automatically read the datum information and there is no need for manual datum setting. However, if the drive or motor is replaced, manual setting of datum is needed.

Follow the steps below to set datum:

1. In **Manual** mode, control the target axis move to the target position.
2. Switch to **Homing** mode. In the menu bar, go to **Machine** > **Datum Setting** > **X-axis(X)/Y-axis(Y)/Z-axis(Z)/All Axis Datum Setting** to set datum for a single axis or all axes.

After setting datum for an axis, the icon will be displayed before the axis in the axis coordinate area.

## 3.7 Set Workpiece Origin

A certain point on the workpiece is selected as the workpiece origin (origin of the axis coordinate system of the machining paths). The workpiece origin is fixed relative to the points on the workpiece and mobile relative to the mechanical origin. Programmers select the workpiece origin commonly under considerations of convenience for programming and size conversion, and less machining error.

You can select one from the following methods to set the position of the workpiece origin:

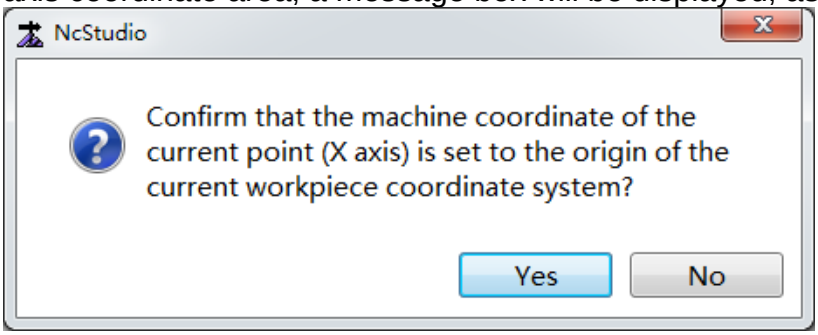
- Clearing
- Centering
- Setting workpiece offset and public offset parameters
- Tool calibration
- Setting tool compensation parameters

### 3.7.1 Clearing

When the machining accuracy requirement is not very high and the work piece has an irregular shape, you can select the workpiece origin with the clearing method. Clearing means to set the workpiece coordinates of the current point to 0.

Taking the X axis as an example, follow the steps below to execute clearing:

1. Control the X axis to move to the target point which you want to set as the workpiece origin.
2. In **Auto** mode, click the F3 pull-down menu in the operation button bar and select **XClear**; or press the **F3** key; or click the X-axis workpiece coordinate button in the axis coordinate area, a message box will be displayed, as shown below:



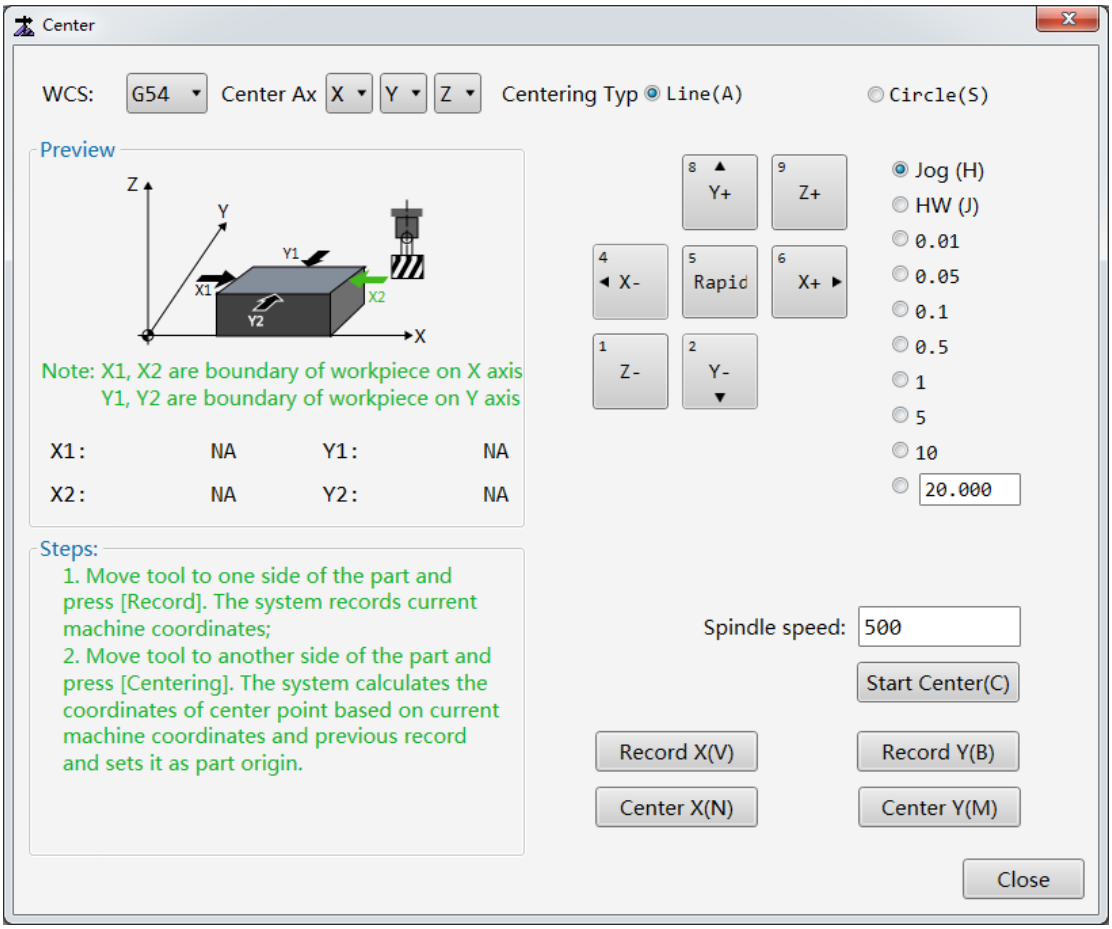
Click **Yes**. The X-axis workpiece coordinate of the current point becomes 0.

### 3.7.2 Centering

Two types of centering are supported: line centering and circle centering. Line centering is mostly used to locate the center of regular rectangle work pieces while circle centering circular work pieces.

Taking line centering for the X axis as an example, follow the steps below:

1. In **Manual** mode, go to **Machine** > **Centering** in the menu bar to open the **Center** window:




2. **Optional**: To change the workpiece coordinate system, click the **WCS** pull-down menu. Workpiece coordinate systems of G54–G59 are supported. The default workpiece coordinate system is G54.
3. Set **Centering Type** to **Line**.
4. **Optional**: If an edge finder is used for more accurate positioning, follow the steps below:
   a. In the **Spindle speed** field, enter the target spindle speed during centering. Its value should not be too large and is set to 500 (r/min) by default.
   b. Click **Start Center** to make it highlighted in green.
5. Click the **X+** / **X-** button to make the X axis move to one end of the work piece. Click **Record X** to record the X-axis mechanical coordinate of the current point into the system.
6. Click the **X+** / **X-** button to make the X axis move to the other end of the work piece. Click **Center X**. The system automatically calculates the X-axis midpoint

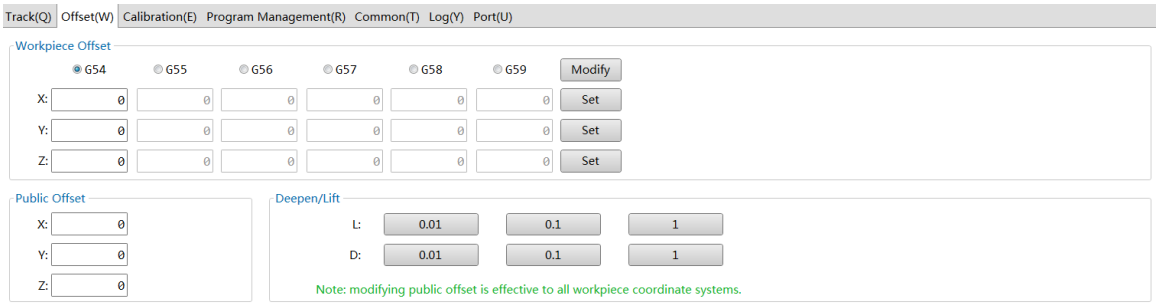
mechanical coordinate based on the X-axis mechanical coordinate of the current point and the X-axis mechanical coordinate recorded in the last step, and sets the X-axis workpiece coordinate of the midpoint to zero.

### 3.7.3 Setting Workpiece Offset and Public Offset

Workpiece offset indicates deviation of the workpiece origin from the mechanical origin. Public offset indicates deviation of a whole workpiece coordinate system from the positions established based on the workpiece offset.

Follow the steps below to set the workpiece offset and public offset parameters:

1. Click the **Offset** tab in the function window.



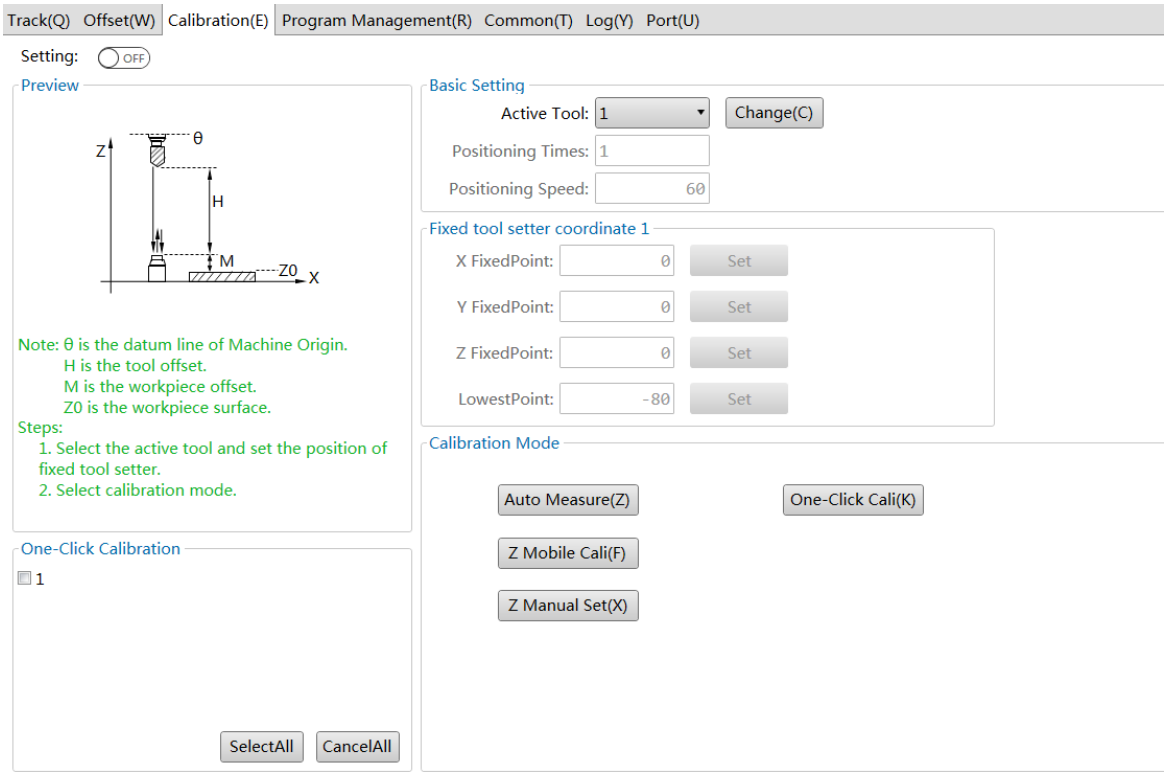

2. Select the target workpiece coordinate system, such as G54.
3. Enter X/Y/Z-axis workpiece offset in the **X/Y/Z** fields.
4. Click the **Set** buttons behind the **X/Y/Z** fields. In the popped up dialog box, click **Yes** to set the X/Y/Z-axis workpiece coordinate of the current point to 0.
5. **Optional**: Enter X/Y/Z-axis public offset values in the **X/Y/Z** fields in the **Public Offset** area.
6. **Optional**: Click the **0.01/0.1/1** buttons in the **Deepen/Lift** area to adjust the Z-axis public offset by certain amount separately.

### 3.7.4 Tool Calibration

Tool calibration is used to measure the selected tool and check the tool offset to ensure that the tool can work properly.

Follow the steps below to calibrate the selected tool:

1. In **Auto** or **Manual** mode, click the **Calibration** tab in the function window.




2. Set **Setting** in the upper left corner to **ON**.
3. In the **Basic Setting** area, click the **Active Tool** pull-down menu, select the target tool number, and set the following parameters:
    - **Positioning Times:** Indicates the times of fine positioning when approaching the tool sensor during tool calibration.
    - **Positioning Speed:** Indicates the speed when approaching the tool sensor during tool calibration.
4. In the **Fixed tool setter coordinate 1** area, set the parameters of the fixed tool sensor:
    - **X FixedPoint**: X-axis mechanical coordinate of the fixed tool sensor
    - **Y FixedPoint:** Y-axis mechanical coordinate of the fixed tool sensor
    - **Z FixedPoint:** Z-axis mechanical coordinate of the fixed tool sensor
    - **LowestPoint**: Mechanical coordinate of the Z-axis lowest point during tool calibration.
5. Select one from the following tool calibration methods:
    - Fixed calibration
    - First-time calibration/Calibration after tool change
    - Mobile calibration

### *3.7.4.1 Fixed Calibration*

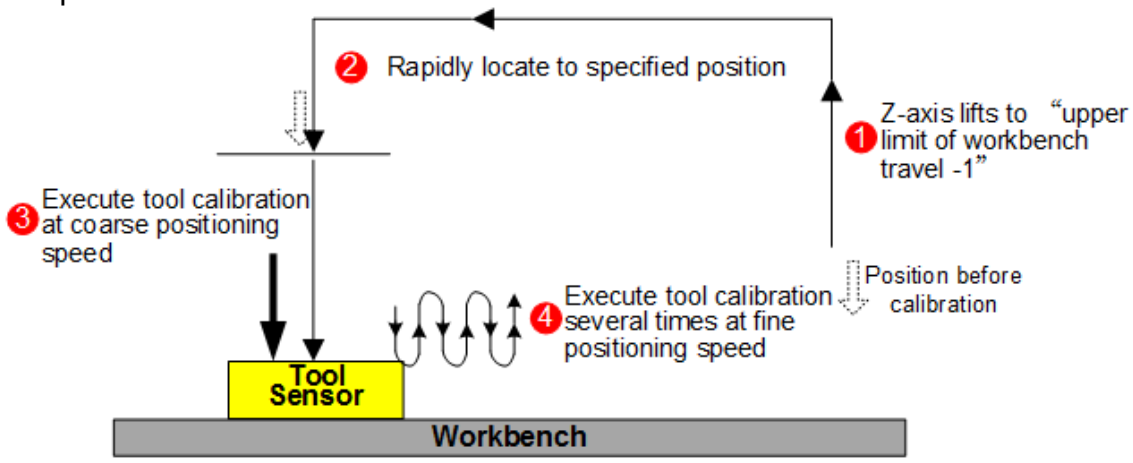
If a tool is replaced due to being broke or other reasons, the length of the new tool and where the new tool is clamped may be different from the replaced one. In this case, fixed calibration can be used to measure the length of the new tool from a fixed position of the machine and determine the tool offset for better machining accuracy. Fixed calibration is applicable to machine with a tool magazine.

Before executing fixed calibration, set the manufacturer parameter **Tool Calibration Type** to **1**.

Follow the steps below to execute fixed calibration:

In the **Calibration Mode** area of the **Calibration** tab, select one method:

- Automatic measuring: Click **Auto Measure**. The system carries out fixed calibration automatically.
  The process of automatic fixed calibration is shown below:




- Manual calibration: Control the tool to move to the tool sensor surface and click **Z Manual Set**. The system records the calibration result into the tool offset.

After fixed calibration is finished, control the tool to move to the work piece surface and execute Z-axis clearing.

### *3.7.4.2 First-time Calibration/Calibration After Tool Change*

First-time calibration/Calibration after tool change operations are used to record tool differences into the workpiece offset.

The operations are applicable to machine without automatic tool change functions and need to be executed every time a tool is changed.

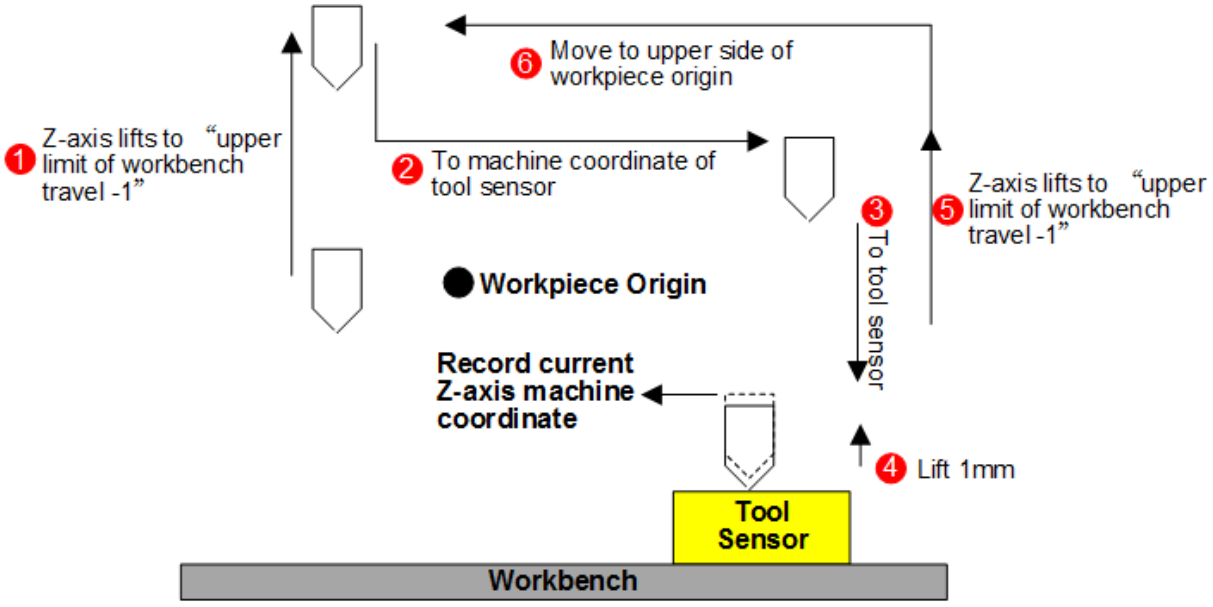
Before executing first-time calibration/calibration after tool change operations:

- Set the manufacturer parameter **Tool Calibration Type** to **2**.
- Control the Z axis to move to the work piece surface and execute clearing to determine the workpiece origin.

In the **Calibration Mode** area of the **Calibration** tab, select one method:

- First-time calibration: Click **First Cali**. The system automatically records the Z-axis mechanical coordinate of the current point.
- Calibration after tool change: Click **Second Cali**. The system automatically recovers the Z-axis workpiece coordinate of the current point.

First-time calibration process and calibration after tool change process are similar, as shown below:




### *3.7.4.3 Mobile Calibration*

During mobile calibration, the Z-axis workpiece coordinate is automatically cleared. Only workpiece offset of the current workpiece coordinate system will be changed.

Before executing mobile calibration, ensure that:

- Set the manufacturer parameter **Tool Calibration Type** to **1**.

In the **Calibration Mode** area of the **Calibration** tab, Click **Z Mobile Cali**. The system executes tool calibration at the current position and records the calibration result into the tool offset parameter.

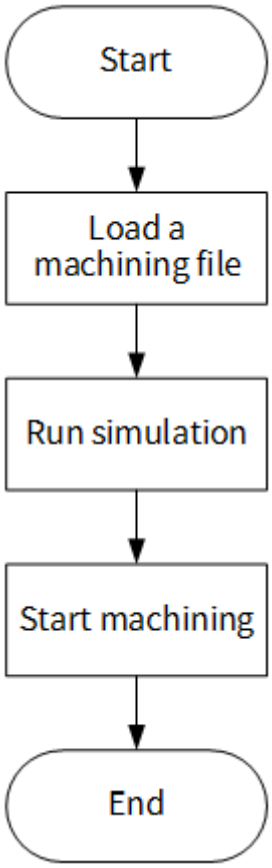
### 3.7.5 Setting Tool Compensation Parameters

For details, see section 3.7.5 Tool Compensation Parameters.

# 4 Quick Start

## 4.1 Overview

This section gives you an introduction of a simple machining process with the **NcStudio Phoenix Three-axis Engraving System**:




## 4.2 Loading a Machining File

Before starting machining, you need to load the machining program first.

For details, see section 5.1.1 Loading a Program.

## 4.3 Running Simulation

During simulation, the system does not drive machine movement or consume machine or workpiece resources. Only the machining path and process are displayed in the software for you to spot possible problems and edit the program accordingly.

Follow the steps below to run simulation:

1. In **Auto** or **Manual** mode, select one from the following methods to access simulation mode:
   - In the operation button bar, click **Simulate**.
   - In the menu bar, go to **Machine** > **Simulate**.
2. Select one from the following methods to start simulation:

- Press the **F9** key.
- In the operation button bar, click 

.
- In the menu bar, go to **Machine** > **Cycle Start**.

The simulation track is displayed in the **Track** tab of the function window.

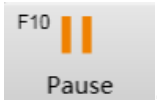
## 4.4 Starting Machining

Select one from the following methods to start machining:

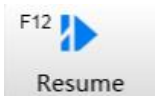
- Press the **F9** key.
- In the operation button bar, click 

.
- In the menu bar, go to **Machine** > **Cycle Start**.

After machining is started, the following functions are supported:

- To pause machining, select one from the following methods:
  - Press the **F10** key.
  - In the operation button bar, click 

.
  - In the menu bar, go to **Machine** > **Cycle Pause**.
- To stop machining, select one from the following methods:
  - Press the **F11** key.
  - In the operation button bar, click 

.
  - In the menu bar, go to **Machine** > **Cycle Stop**.
- To resume machining from the interrupted position in case of power-off, emergency stop, or other exception, select one from the following methods:
  - Press the **F12** key.
  - In the operation button bar, click 

.
  - In the menu bar, go to **Machine** > **Resume**.

# 5 Program-related Functions

## 5.1 Program Management

You can create, load, unload, edit, delete, and rename a program in the software.

### 5.1.1 Loading a Program

Select one from the following methods to load a machining program:

- For nc., g., eng., nce., ncex., dfx. and plt. files:
  - In the operation button bar, click **Load**.
  - In the menu bar, go to **File** > **Open and Load**.
  - In the function window, click the **Program Management** tab. Select the target path in the upper left corner. All g.,eng., dxf., nc., plt., txt., nce., anc., cnc. and tap. programs in the path will be displayed. Select the target file and click Load below.
- To load a file that was loaded before, go to **File** > **Load History File** in the menu bar.

### 5.1.2 Unloading a Program

Select one from the following methods to unload a machining program:

- In the operation button bar, click **Unload**.
- In the menu bar, go to **File** > **Unload**.

### 5.1.3 Creating, Editing, Deleting and Renaming a Program

Follow the steps below:

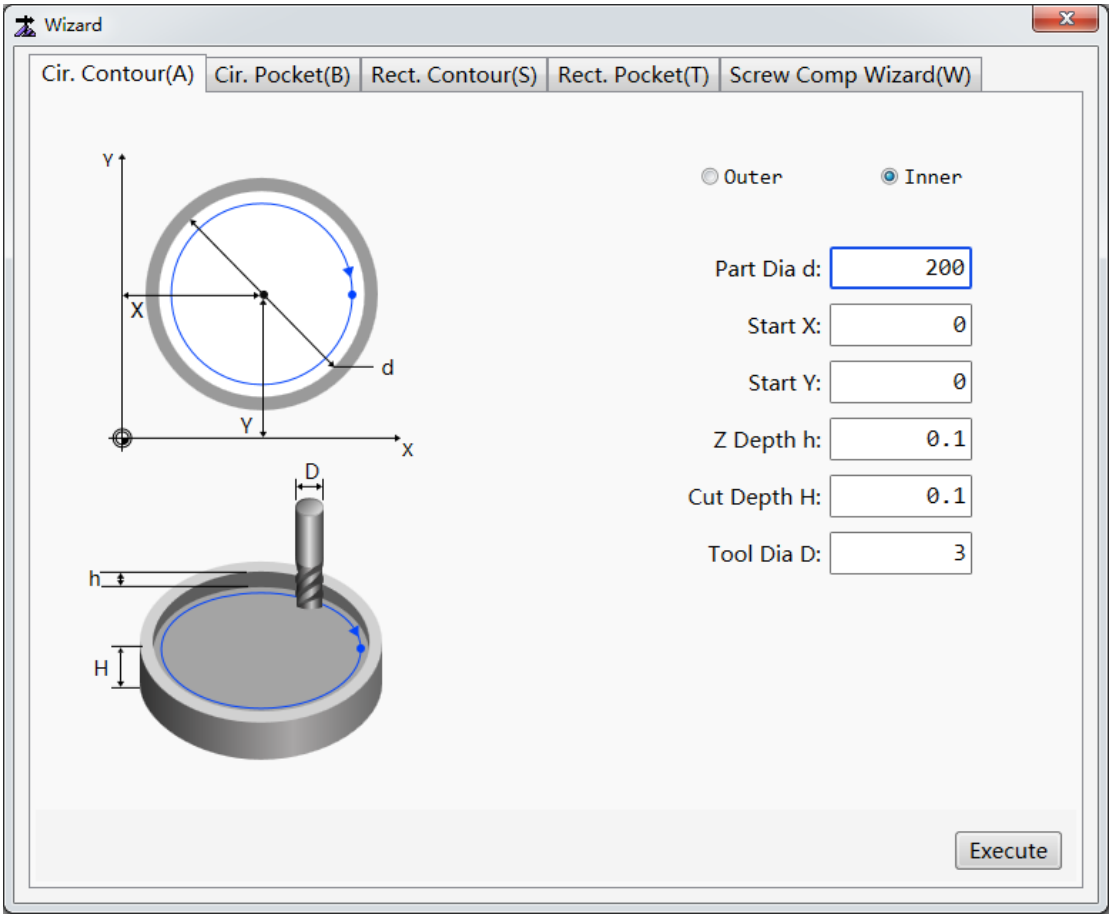
1. In the function window, click the **Program Management** tab.
2. Select the target path in the upper left corner. All g.,eng., dxf., nc., plt., txt., nce., anc., cnc. and tap. programs in the path will be displayed.
3. To create a file, click **New File**. To modify an existing file, click the target program and click **Edit/Delete/Rename**.

## 5.2 Machining Wizard

The system provides **Cir. Contour**, **Cir. Pocket**, **Rect. Contour**, and **Rect. Pocket** wizard. The system automatically generates corresponding machining programs to help you save the time of manual editing.

Follow the steps below to use the machining wizard:

1. In **Auto** mode, go to **Program** > **Machining Wizard** to open the **Wizard** window:




2. Select a tab and set the parameters based on the left figure.
3. Click **Execute**. The system starts machining.

## 5.3 Single Block Machining

In this mode, the program commands will be executed line by line for you to check the machine movement.

Follow the steps below to execute single block machining:

1. In **Auto** mode, go to **Machine** > **Single Block Execution** in the menu bar to enable single block machining mode.
2. Select one from the following methods to make the system run a command line and pauses:
   - Press the **F9** key.
   - In the operation button bar, click 

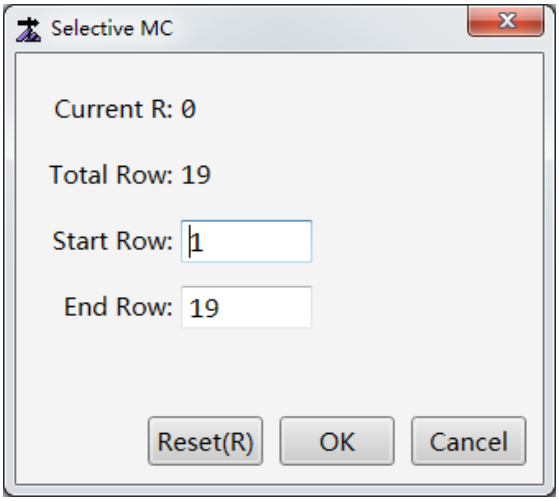
.
   - In the menu bar, go to **Machine** > **Cycle Start**.
3. Repeat step 2 to make the system run the next command line until all commands are executed.

## 5.4 Selective Machining

In this mode, you can specify the start and end command line number to run the target commands.

Follow the steps below to use selective machining:

1. Select one from the following methods to open the **Selective MC** dialog box:




   - In **Auto** mode, click **Selective MC** in the operation button bar.
   - In the menu bar, go to **Advanced** > **Selective Machining**.
2. Enter target values in the **Start Row** and **End Row** fields.
3. **Optional**: To reset the row numbers, click **Reset**. The **Start Row** and **End Row** values will be restored to their default values.
   The default start row is the first row and the default end row is the last row.
4. Select one from the following methods to start selective machining:
   - Press the **F9** key.
   - In the operation button bar, click 

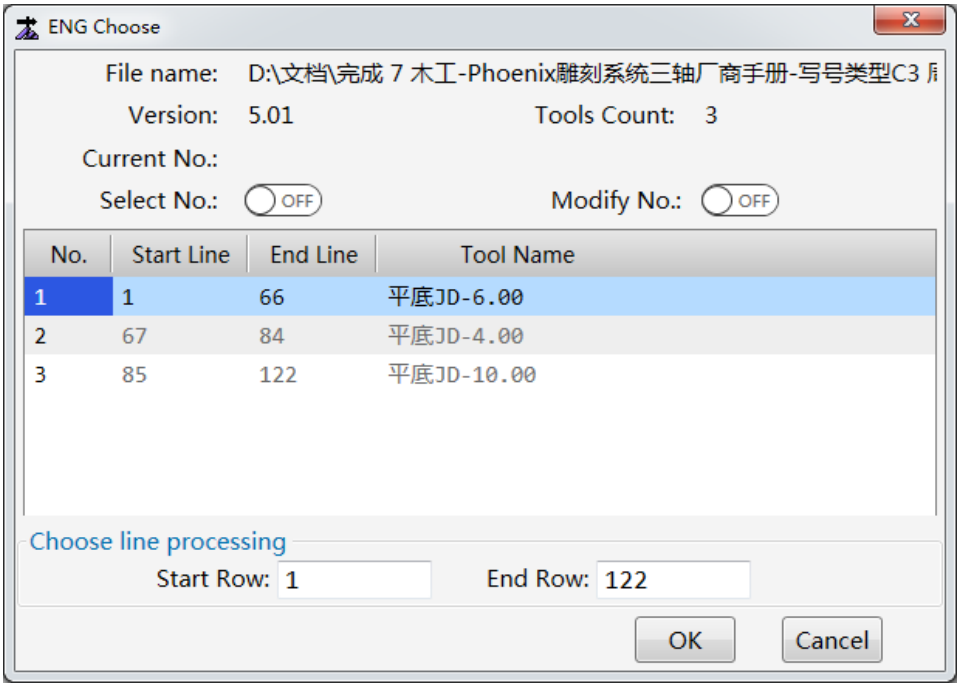
.
   - In the menu bar, go to **Machine** > **Cycle Start**.

## 5.5 ENG Tool and Line Selection

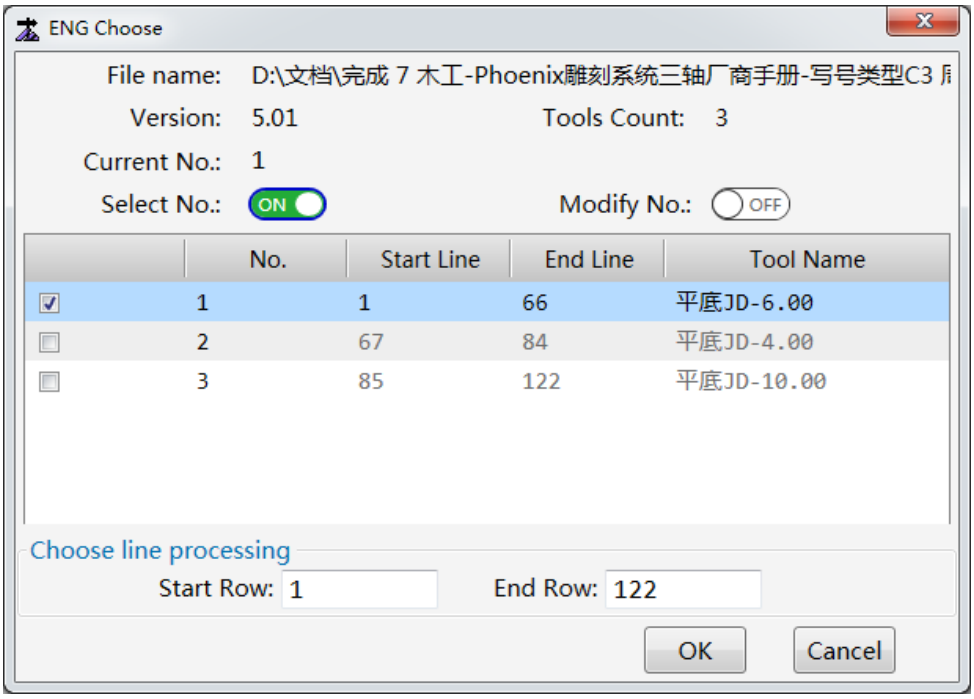
If the loaded program is an ENG file, you can specify a tool to run certain command lines.

Follow the steps below to use this function:

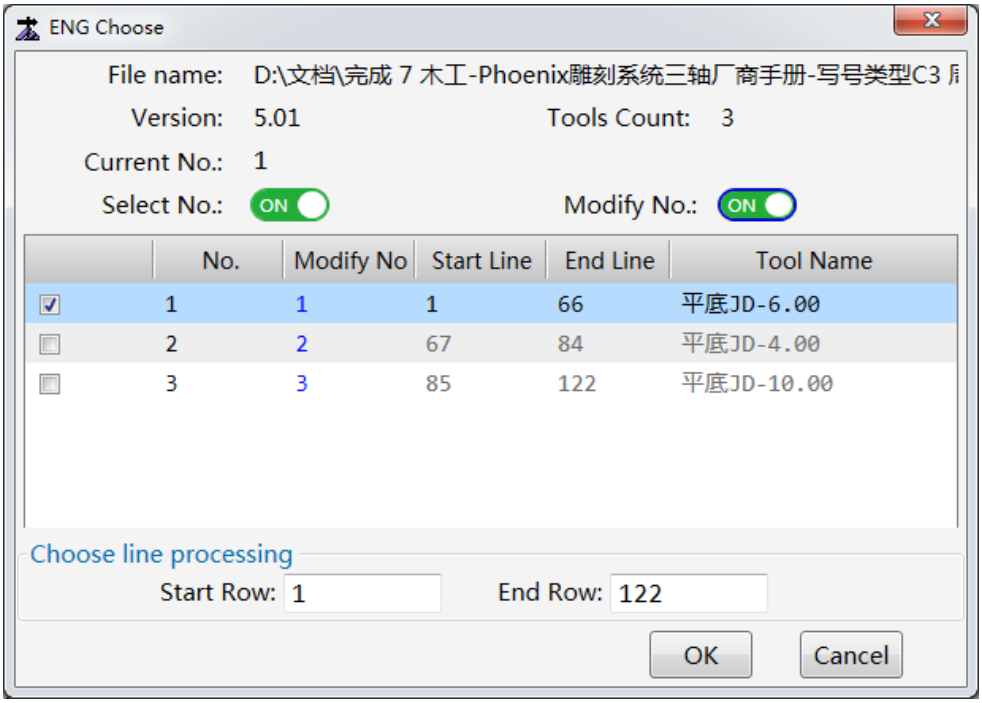
1. After loading the ENG program, go to **Advanced** > **ENG choose tool and row** in the menu bar to open the **ENG Choose** dialog box:




2. To specify the tool, set **Select No.** to **ON**. A selection box column will be displayed before the tool number column.




Tick the target tool number.
To modify a tool number, set **Modify No.** to **ON**. The **Modify No.** column will be displayed, as shown below:




Double-click the target field in the **Modify No.** column and enter the target tool number. Click **Confirm**.

3. In the **Choose line processing** area, enter values in the **Start Row** and **End Row** fields. Click **OK**.

## 5.6 Advanced Machining

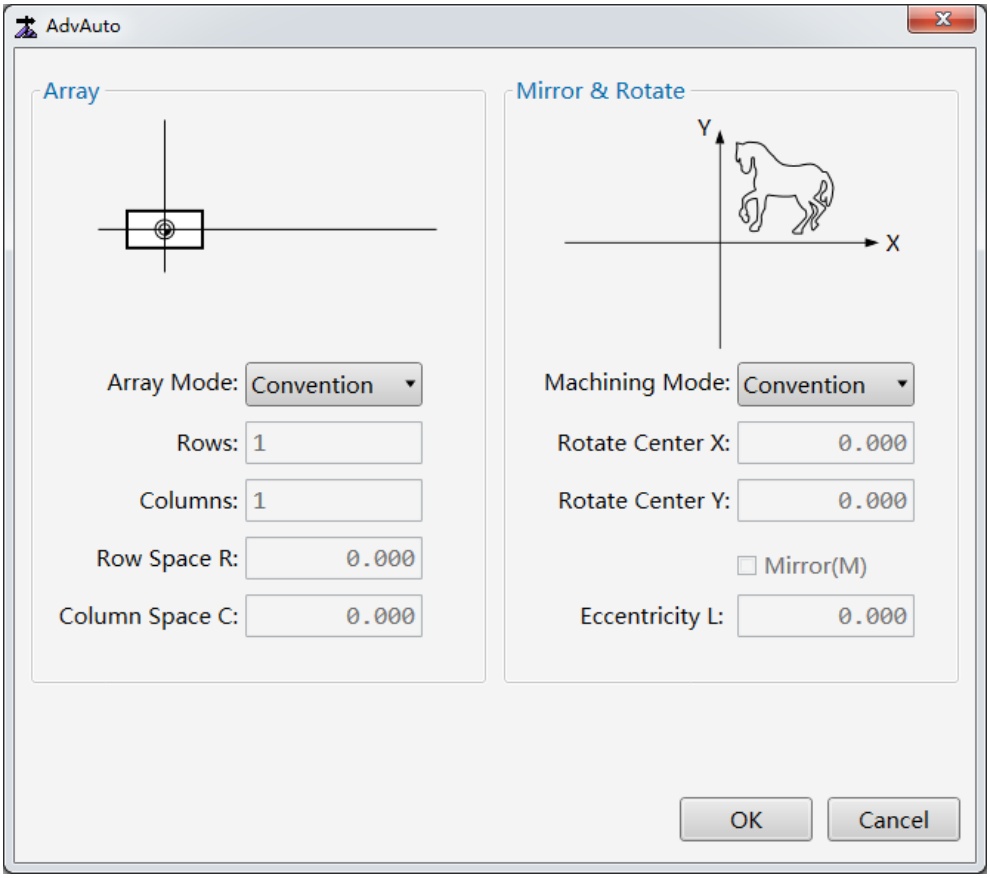
Advanced machining includes adding of machining path array, rotation, and mirroring.

### 5.6.1 Array

Array machining means to copy the machining path along a rectangular array.

Follow the steps below to use array machining:

1. In **Auto** or **Manual** mode, go to **Advanced** > **AdvAuto** in the menu bar to open the **AdvAuto** window:



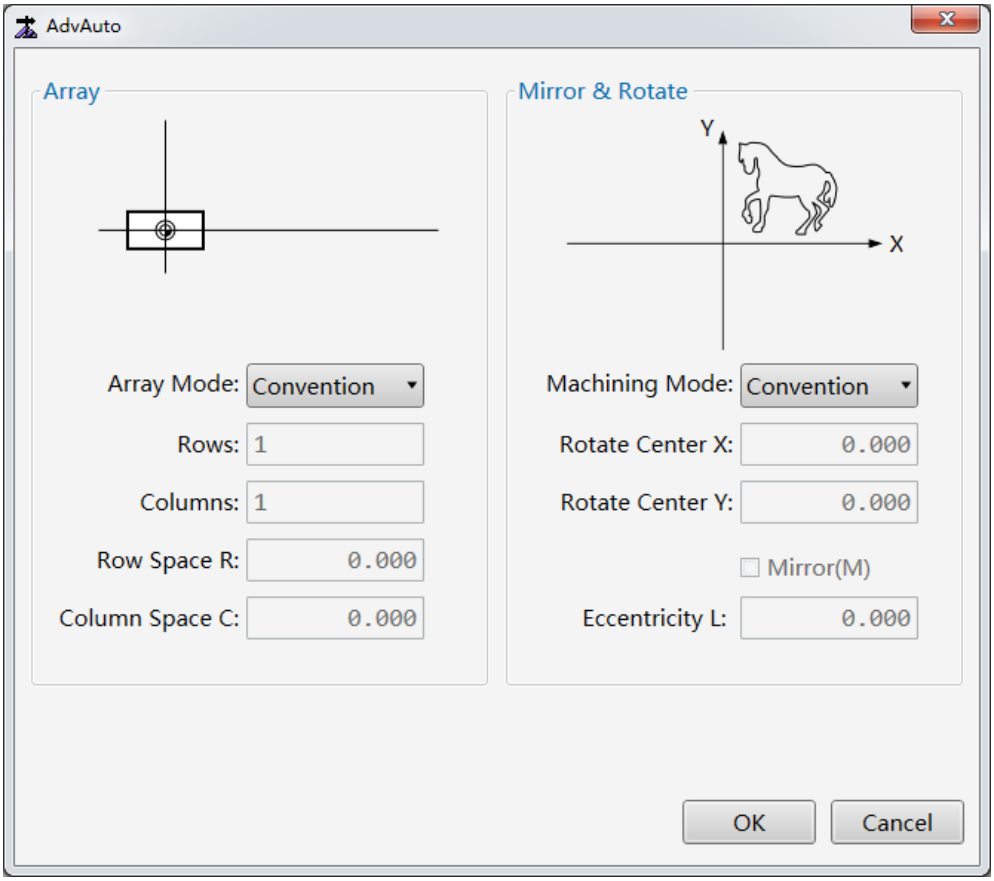

2. In the **Array** area, click the **Array Mode** pull-down menu and select **Rectangular**. The array mode is set to **Convention** by default, which means does not enable array.
3. Enter the number of rows and columns, row interval, and column interval in the **Rows**, **Columns**, **Row Space R**, and **Column Space C** fields.

### 5.6.2 Rotation and Mirroring

Rotation means to rotate the machining path by a certain angle while mirroring means to generate a symmetrical path along the X axis or Y axis.

Follow the steps below to use the rotation and mirroring function:

1. In **Auto** or **Manual** mode, go to **Advanced** > **AdvAuto** in the menu bar to open the **AdvAuto** window:




2. In the **Mirror & Rotate** area, click the **Machining Mode** pull-down menu and select an option:
   - To rotate the path:
     a. Select **CW 90°/CW 180°/CCW 90°**.
     b. Enter the rotation center X and Y coordinates in the **Rotate Center X** and **Rotate Center Y** fields.
   - To mirror the path:
     a. Select **Mirroring by X/Mirroring by Y**.
     b. Tick **Mirror**.
     c. **Optional:** To adjust the position of the paths, enter the target deviation in the **Eccentricity L** field.

   The machining mode is set to **Convention** by default, which means not to enable rotation or mirroring.

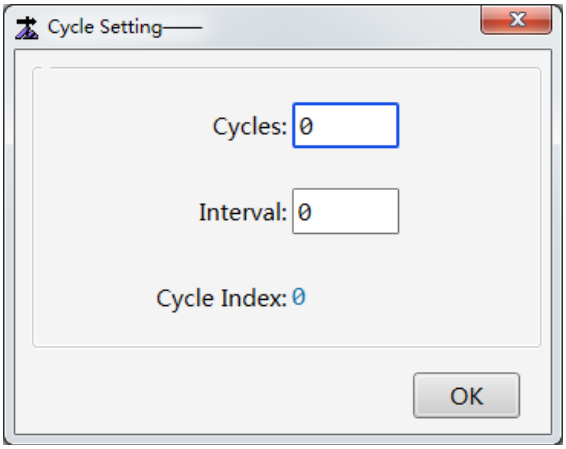
## 5.7 Machining Cycle

You can set the machining to be repeated for a certain number of times and at certain interval, and check the current times of machining.

Every complete running of the program counts as one time.

Follow the steps below to set machining cycle:

1. In the menu bar, go to **Advanced** > **Cycle Setting** to open the **Cycle Setting** dialog box:




2. Enter the target cycle times and interval in the **Cycles** and **Interval** fields. The current times of machining will be displayed after **Cycle Index**.

# 6 Check Machining Information and Statistics

You can check the machining information automatically collected by the system to formulate machining plans and avoid possible problems.

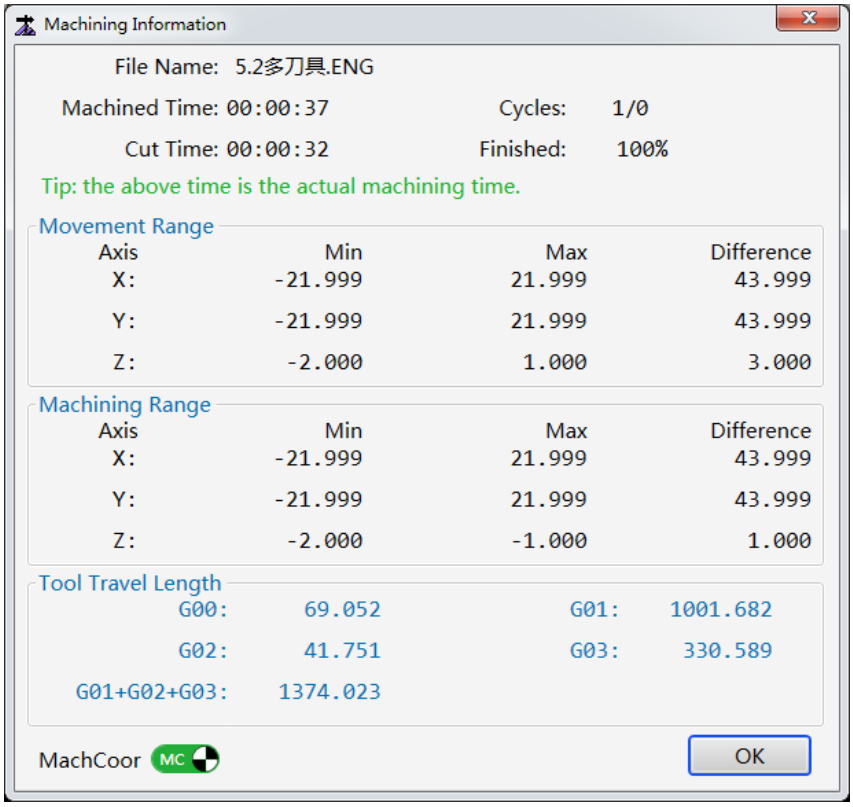
Follow the steps below to check the machining information/statistics:

In the menu bar, go to **Program** > **Machining Information/Machining Statistics** to check corresponding information:

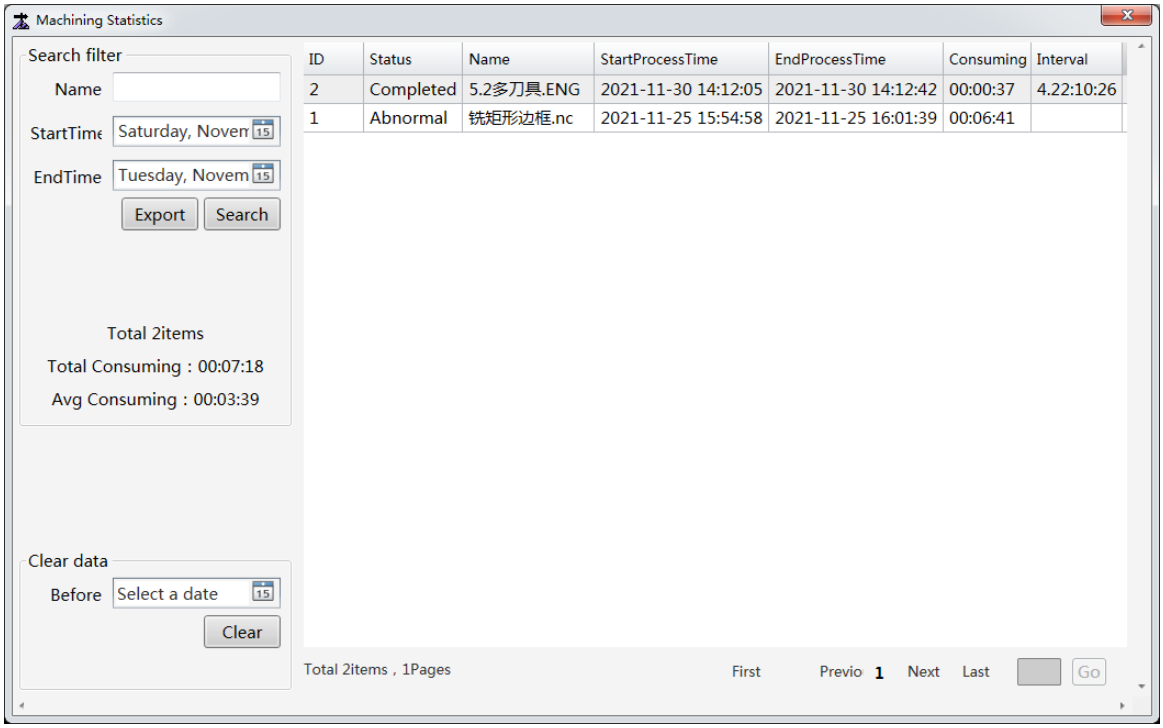
In the **Machining Information** window, the movement range, machining range and tool travel length are displayed.

In the **Machining Statistics** window, you can search for machining records by name or date and clear records before a certain date.

### Machining Information




### Machining Statistics


Machining Statistics
Search filter
Name
StartTime
Saturday, Noven
EndTime
Tuesday, Novem
Export
Search
Total 2items
Total Consuming : 00:07:18
Avg Consuming : 00:03:39
Clear data
Before
Select a date
Clear
ID
Status
Name
StartProcessTime
EndProcessTime
Consuming
Interval
2
Completed
5.2多刀具.ENG
2021-11-30 14:12:05
2021-11-30 14:12:42
00:00:37
4.22:10:26
1
Abnormal
铣矩形边框.nc
2021-11-25 15:54:58
2021-11-25 16:01:39
00:06:41
Total 2items , 1Pages
First
Previo
1
Next
Last
Go

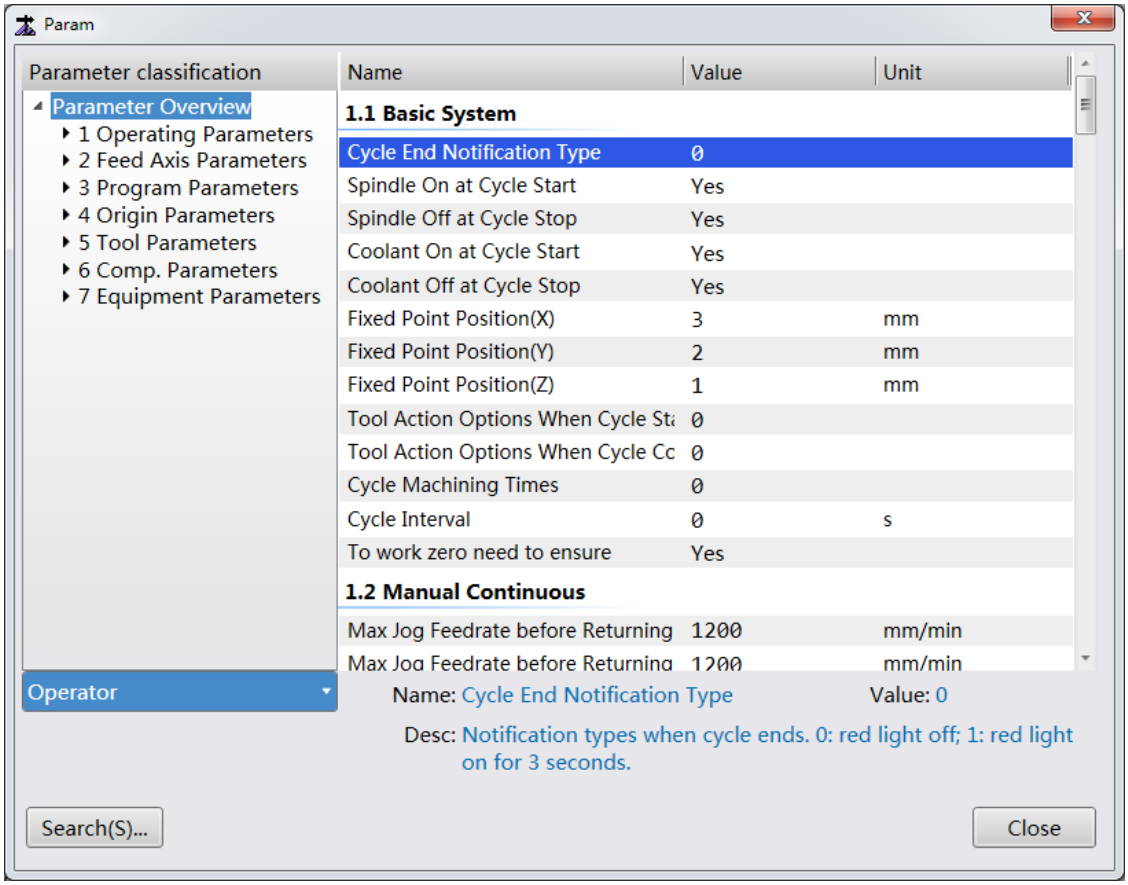
# 7 System Settings

## 7.1 Viewing and Setting Parameters

This section introduces how to set the system global parameters, drive parameters, and tool compensation parameters.

### 7.1.1 Global Parameters

In the menu bar, go to **System** > **Global Parameters** to open the **Param** window:




Click the **Operator** pull-down menu on the left side. Select **Operator** or **Manufacturer** to check and set corresponding parameters.

There are seven types of global parameters: operating parameters, feed axis parameters, program parameters, origin parameters, tool parameters, compensation parameters, and equipment parameters.

You can search for a parameter by clicking a parameter type in the left list or clicking **Search** in the lower left corner, entering the parameter name in the field, and clicking **Next**.

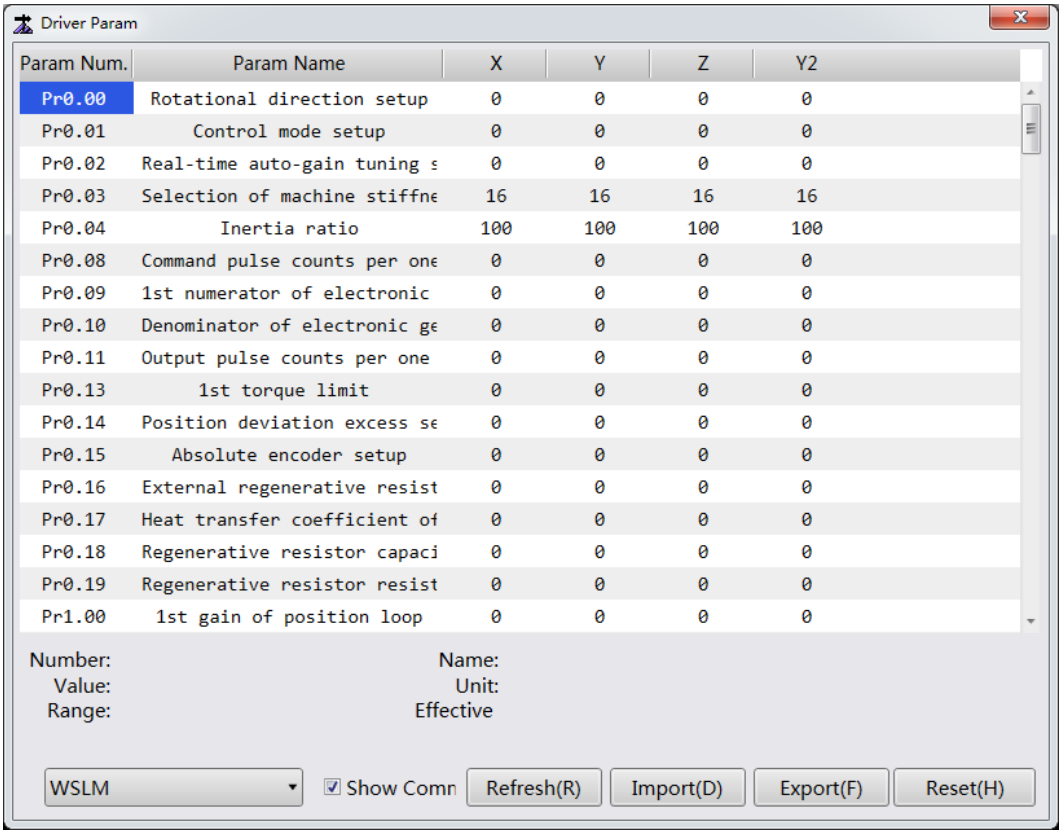
### 7.1.2 Drive Parameters

After the system is connected to the drive, it automatically reads the drive parameter settings. You can also modify the drive parameter settings based on the machine structure and your demands.

Before setting drive parameters, ensure that the system is connected to the hardware correctly.

Follow the steps below to set drive parameters:

1. In the menu bar, go to **System** > **Driver Parameters** to open the **Driver Param** window:




2. Double-click the target field, enter the target value, and click **OK**.

In the **Driver Param** window, the following functions are also supported:

- To show common driver parameters, tick **Show common**.
- To refresh the parameter list, click **Refresh**.
- To import drive parameter settings, click **Import**, select the target file, and click **Open**.
- To export drive parameter settings, click **Export**, select the target storage path, and click **Save**.
- To restore parameter default values, click **Reset**.

### 7.1.3 Tool Compensation Parameters

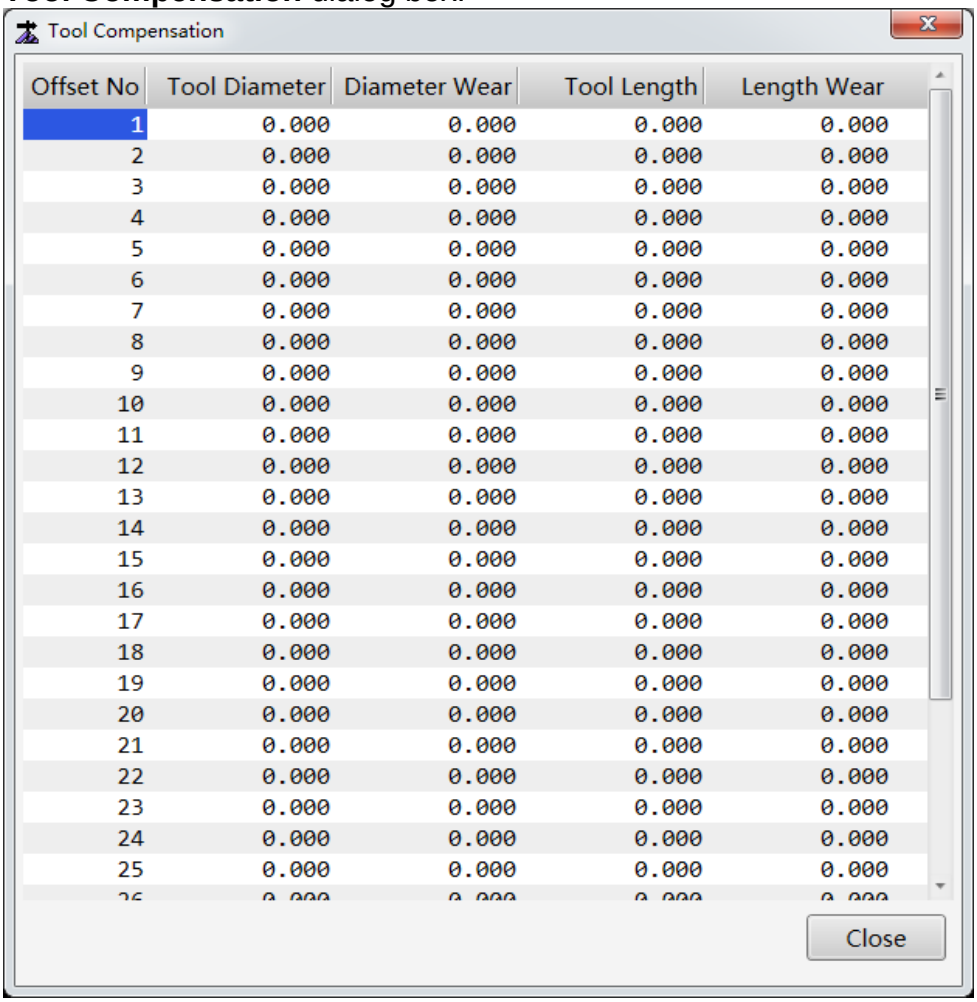
If a tool is worn or changed, the diameter or length of the new tool may be different. In this case, you need to modify the tool compensation parameter values to achieve accurate machining.

Before setting tool compensation parameters:

- Set the global parameter **Enable Tool Length Compensation** to **Yes**.
- Set the global parameter **Enable Tool Radius Compensation** to **Yes**.
- Measure and record the following parameters:
  - **Tool Diameter**: The diameter of the tool.
  - **Diameter Wear**: The amount of the tool diameter that is worn.
  - **Tool Length**: The length of the tool.
  - **Length Wear**: The amount of the tool length that is worn.

Follow the steps below to set tool compensation values:

1. In the menu bar, go to **System** > **Tool Compensation Parameters** to open the **Tool Compensation** dialog box:



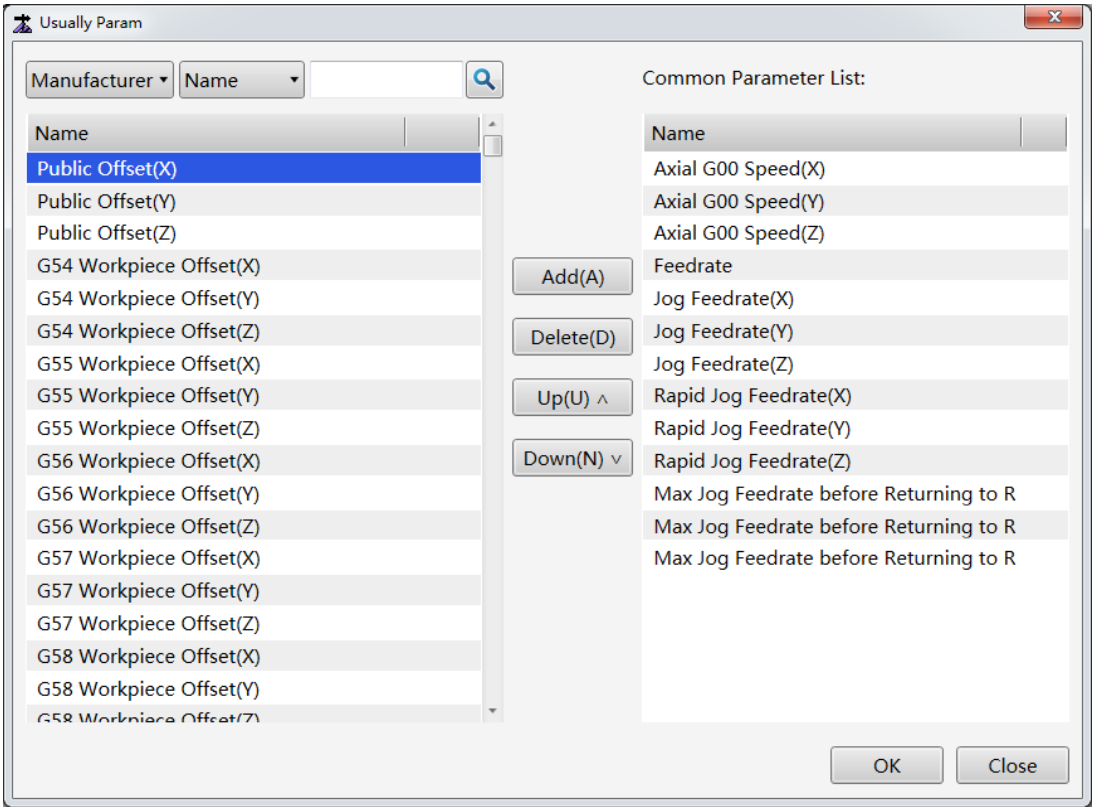

2. Double-click the target field, enter the measured value, and click **OK**.

## 7.2 Adding Common Parameters

You can set target parameters to become common parameters and displayed under the **Common** tab of the function window.

Follow the steps below to add a parameter into the common parameter list:

1. Click the **Common** tab in the function window.
2. Click **Set** in the lower right corner to open the **Usually Param** window:



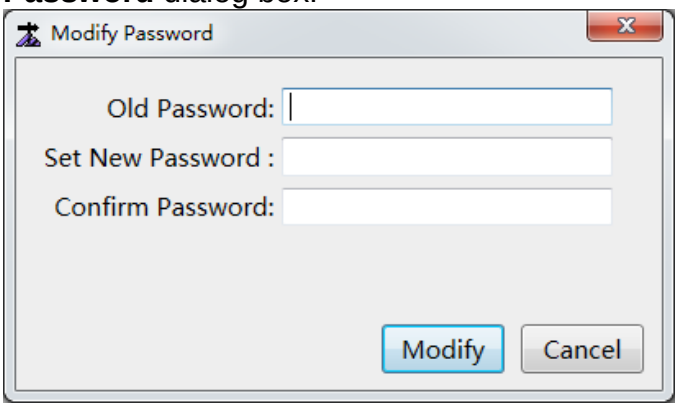

3. Click the first pull-down menu and select **Operator** or **Manufacturer** to display operator or manufacturer common parameters. Click the second pull-down menu and select **Name**, **Description**, or **Address** to determine the query condition. Enter the searching keyword in the blank field and click the searching icon.
4. After locating the target parameter, click it and click **Add**. The target parameter will also be displayed under the right **Common Parameter List**.
5. Repeat step 3 and step 4 until all target parameters are added.
6. **Optional**: To remove a parameter from the common parameter list or adjust its position in the list, click the parameter and click **Delete/Up/Down**.

## 7.3 Changing Password

The password is used to view and modify manufacturer parameters.

Follow the steps below to change the password:

1. In the menu bar, go to **System** > **Change Password** to open the **Modify Password** dialog box:



2. Enter the old password and new password in the corresponding fields and click **Modify**.

## 7.4 Switching System Language

The software supports Chinese and English.

Follow the steps below to switch to English interface: In the menu bar, go to **System** > **Language** > **English**.

Restart the software.

## 7.5 Data Backup

The system supports backup of the parameter settings and log.

Follow the steps below to back up the data:

1. In the menu bar, go to **System** > **Data Backup** to open the **Save as** dialog box.
2. Select a storage path for the backup files and click **Save**.

## 7.6 Remote Assistance

You can use the remote assistance service to let your machining system remotely controlled by technical support personnel to help troubleshoot.

Before using the remote assistance service, ensure that your computer is connected to the Internet.

Follow the steps below to use the remote assistance service:

1. In the menu bar, go to **System** > **Remote Assistance**.
2. Give the computer identification code and the verification code to the technical support staff for remote control.

## 7.7 Using NcGateway

NcGateway is a component of the WEIHONG NcCloud system. It provides data interfaces and communication protocols for device Internet connection. You can use NcGateway to manage devices.

In the menu bar, go to **System** > **NcCloud** to open the NcGateway window.

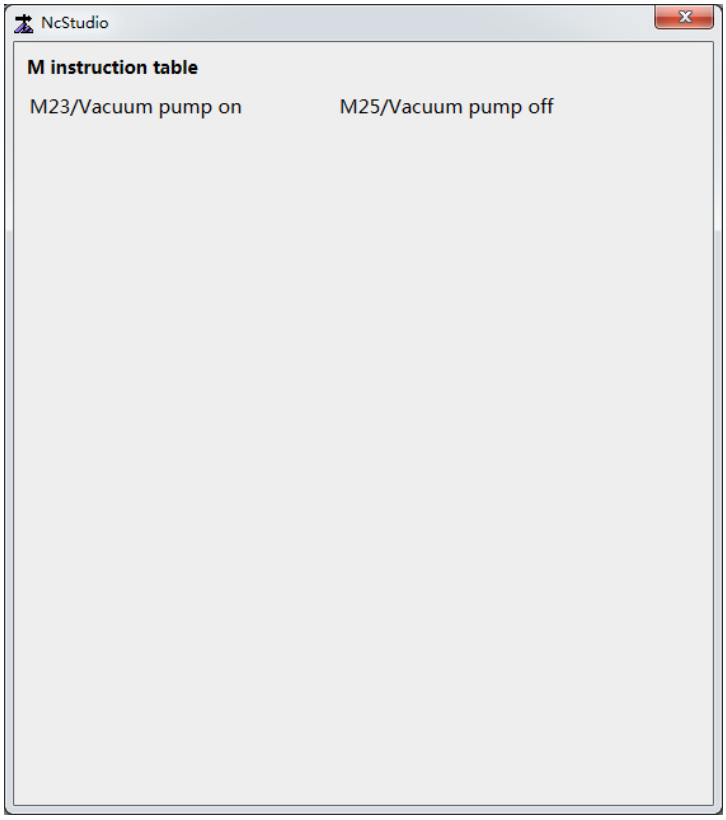
For details, see NcGateway manuals.

## 7.8 M Codes

You can use M codes to run pre-defined sub-programs.

Follow the steps below to check the currently available M codes:

In the menu bar, go to **System** > **M Instruction Table**:



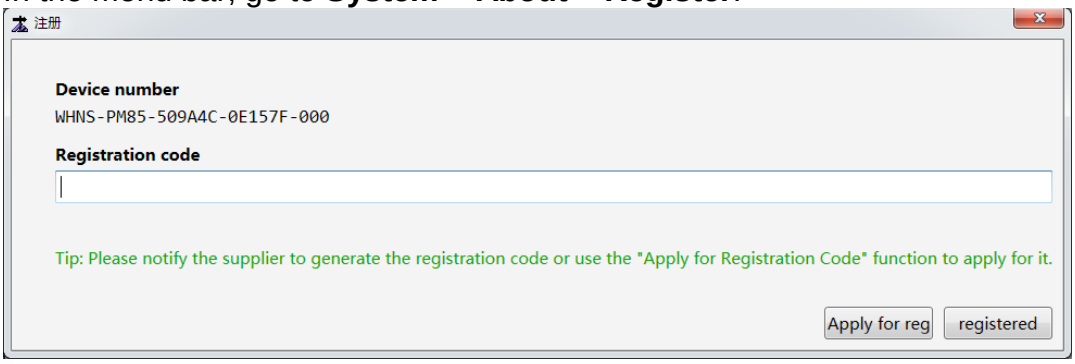

After adding software components in NcConfig, M codes of the components will also be added. Modification is supported.

## 7.9 Software Registration

You need to register the software if the software use period has or is about to expire, or if the writing type does not match.

Follow the steps below to register the software:

1. In the menu bar, go to **System** > **About** > **Register**:




2. Click **Apply for reg** and acquire the registration code by following instructions in the application guide, or contacting the supplier.
3. Enter the registration code in the **Registration Code** field.
4. Click **registered**.

# 8 NcConfig

With NcConfig, you can configure Lambda controllers, terminal boards and machine parts, set port mapping and port protection to improve project efficiency.

## 8.1 Opening NcConfig

Use the following method to open NcConfig:

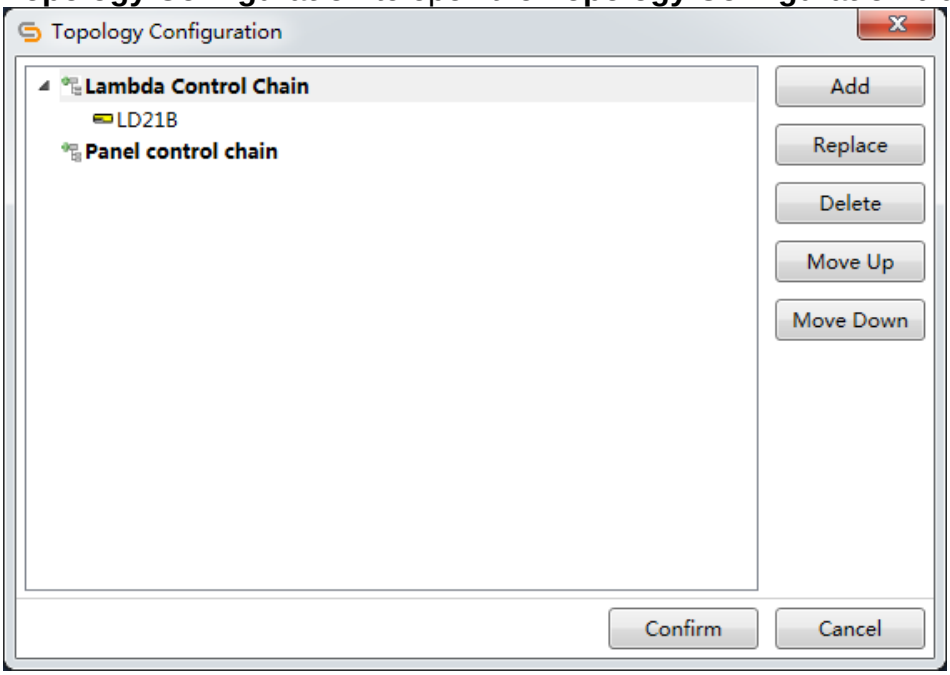
1. Open **C:\Program Files\Weihong\NcStudio\NcConfig\Bin**. Double-click **NcConfig.exe**.
2. **Optional**: In the menu bar, go to **View** > **Configuration/Project** to show the **Configuration** or **Project** tab in the left column.

## 8.2 Configuring Device

You can configure the Lambda controller and terminal boards used in the system.

Follow the steps below to configure the system devices and mapping ports:

1. In the left column, click the **Configuration** tab. Go to **NcStudio** > **Hardware Connection and Configuration** > **Topology Configuration**.
2. Right-click on **Topology Configuration** and click **Edit Topology**, or double-click **Topology Configuration** to open the **Topology Configuration** dialog box:



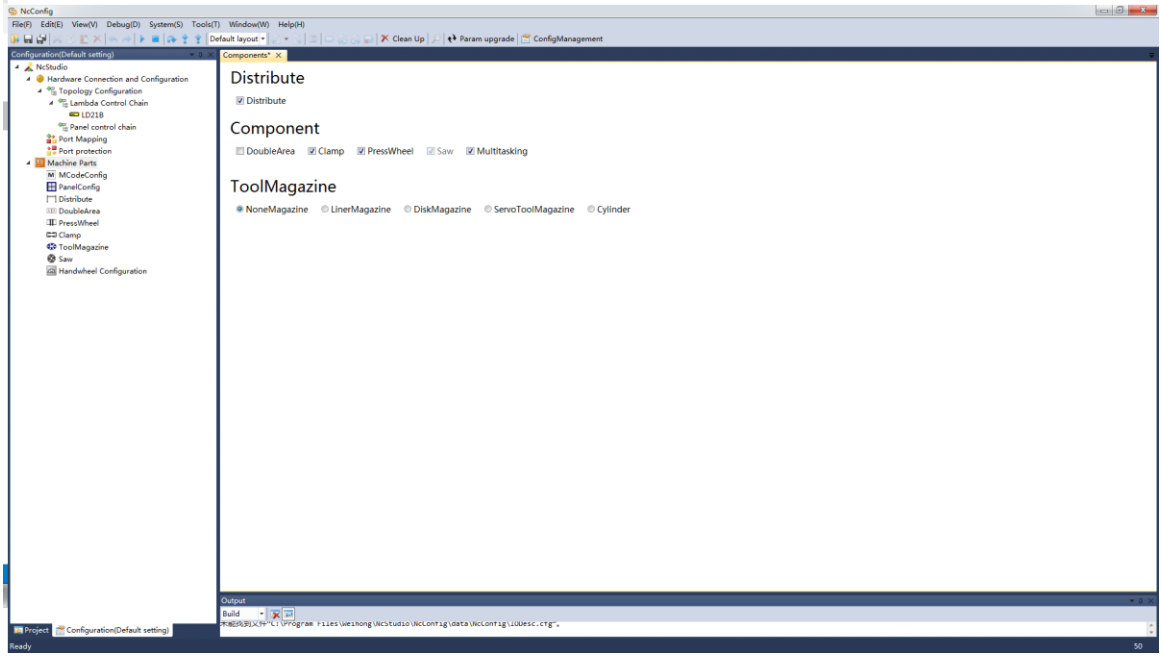

3. Configure the devices based on your demands:
   - To add a controller/terminal board, click **Add** and double-click the target controller/terminal board in the displayed list.
   - To replace a controller/terminal board, click the controller/terminal board to be replaced on the left side and click **Replace**. Double-click the target controller/terminal board in the displayed list.
   - To delete a controller/terminal board, click the controller/terminal board to be deleted on the left side and click **Delete**.
   - To move a controller/terminal board, click the controller/terminal board to be moved on the left side and click **Move Up/Move Down**.
4. Click **Confirm**.

## 8.3 Configuring Machine Parts

In NcConfig, you can select and configure the machine parts in use, such as M codes, operation panel, tool magazine, saw, double work station, clamp, and handwheel.

Follow the steps below to configure machine parts:

1. On the left **Configuration** list, go to **NcStudio** > **Machine Parts**. Double-click **Machine Parts** to open the **Components** page on the right.




2. Tick required parts. The functions of the ticked parts will be displayed in the software.
3. Configure the parts as shown below:
    - Configure M codes
    - Configure function buttons
    - Configure the tool magazine
    - Configure saws
    - Configure dual drive
    - Configure dual work stations
    - Configure press wheels
    - Configure clamps
    - Configure the handwheel

### 8.3.1 Configure M Codes

You can configure the M codes for control of machine movement and port status.

After a machine part is ticked and the settings saved, corresponding available M codes will be displayed on the **MCodeConfig** page.

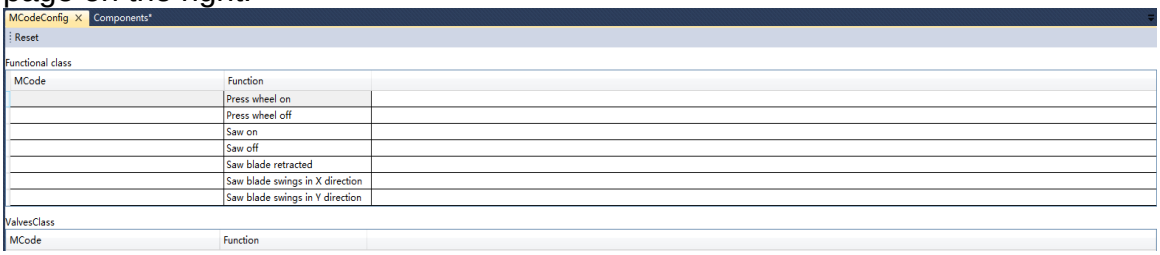
Note: Configure M codes based on the selected machine parts.

M codes include:

- Function M codes: Running functional M codes generates machine movement.
- Valve M codes: Running valve M codes controls port status.

Follow the steps below to configure M codes:

1. In the left **Configuration** list, double-click **MCodeConfig** to open the **MCodeConfig** page on the right:




2. Select the target row and double-click the target **MCode** field to modify the M code.

### 8.3.2 Configure Function Buttons

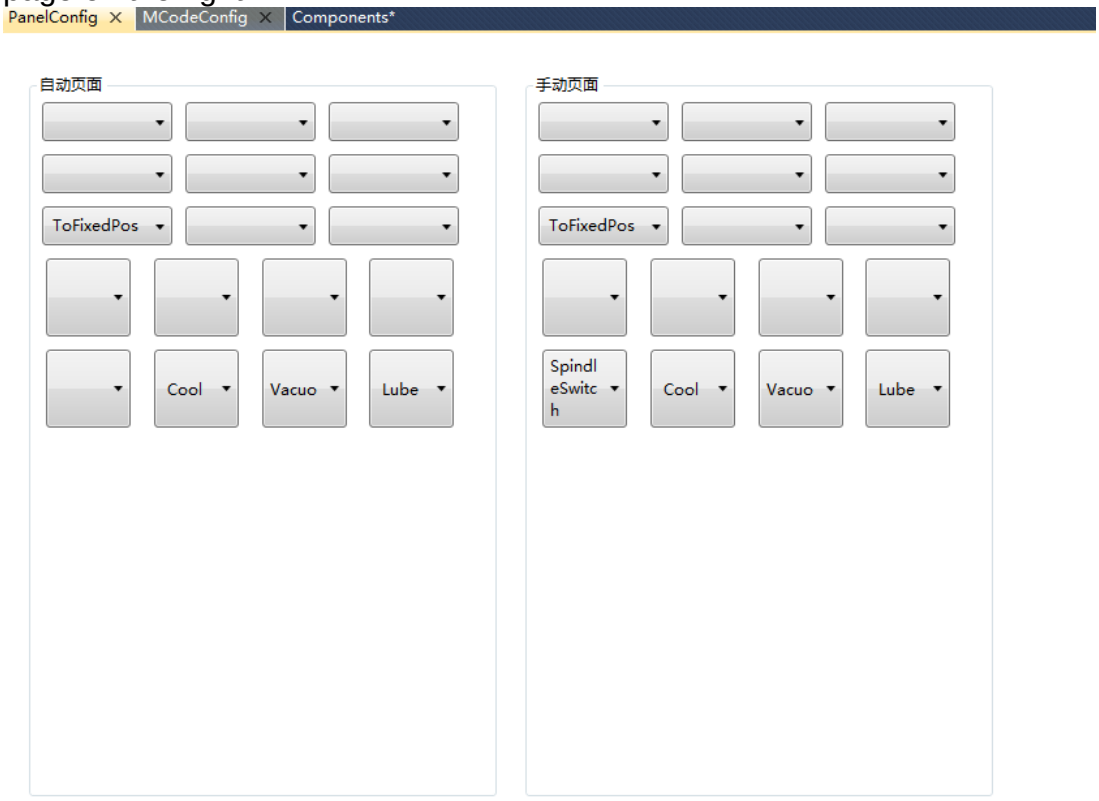
In NcConfig, you can add or delete function buttons in NcStudio.

Available function buttons include those for spindle switch, going to workpiece origin, cooling, vacuum pump, lubrication, etc.

Before configuring function buttons, tick required machine parts on the **Components** page.

Follow the steps below to configure function buttons:

1. In the left **Configuration** list, double-click **PanelConfig** to open the **PanelConfig** page on the right:




2. To add a function button, click the pull-down menu at the target position and select the target function. To remove an existing function button, click the pull-down menu of the function button and select the first blank option. The removed button will no longer be displayed in NcStudio.

### 8.3.3 Configure the Tool Magazine

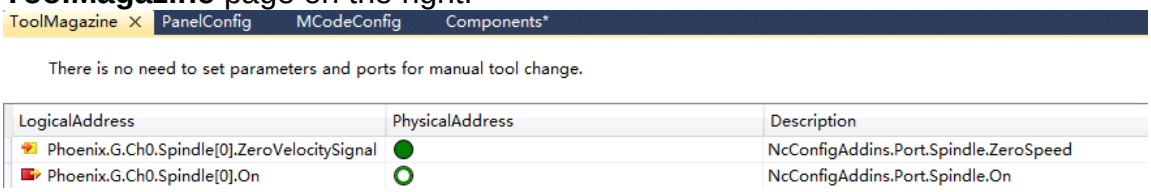
The system supports tool magazines of different mechanical structures, combination of multi-cylinder single-frequency and multi-cylinder multi-frequency functions, and tool magazine selection.

Tool magazine options on the **Components** page include:

- **NoneMagazine**
- **LinearMagazine**
- **DiskMagazine**
- **ServoToolMagazine**
- **Cylinder**

Follow the steps below to configure the tool magazine:

1. In the left **Configuration** list, double-click **ToolMagazine** to open the **ToolMagazine** page on the right:

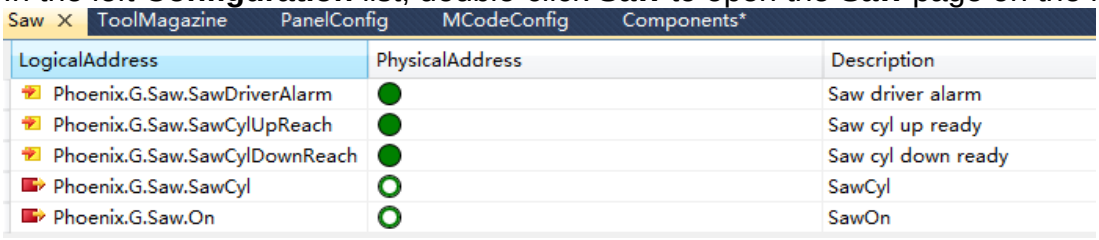



2. Set the tool magazine parameters based on the tool magazine type specified on the **Components** page.
   - **NoneMagazine**: No tool magazine parameters need setting.
   - **LinearMagazine**: Set the parameter **Tool magazine capacity** (the maximum number of tools).
   - **DiskMagazine**: Set the parameter **Tool magazine capacity** (the maximum number of tools) and **Count tool overtime** (maximum time waiting for tool count signals).
   - **ServoToolMagazine**: Set the parameter **Tool magazine capacity** (the maximum number of tools) and rotary axis name.
   - **Cylinder**: Select the cylinder type and set the parameter **Cylinder count**.
     - **Multi-Cylinder Single-Fre**: Every Z axis has multiple cylinders. One inverter controls multiple spindles.
     - **Multi-Cylinder Multi-Fre**: Every Z axis has multiple cylinders. Multiple inverters control multiple spindles.
     - **Four-Cylinder Double-Fre**: Every Z axis has four cylinders. Two inverters control multiple spindles.
3. Double-click the target **PhysicalAddress** field to specify a physical address (controller and port) for the corresponding logical address.

### 8.3.4 Configuring Saws

Follow the steps below to configure saw:

1. In the left **Configuration** list, double-click **Saw** to open the **Saw** page on the right:

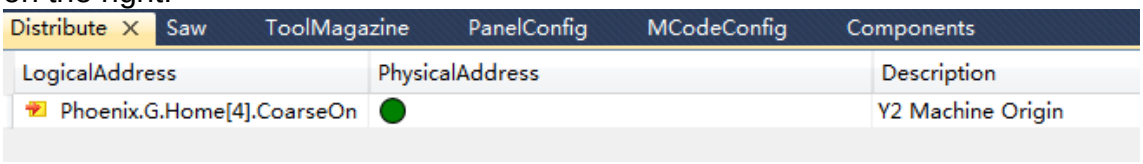


2. Double-click the target **PhysicalAddress** field to specify a physical address (controller and port) for the corresponding logical address.

### 8.3.5 Configure Dual Drive

Follow the steps below to configure dual drive axis:

1. In the left **Configuration** list, double-click **Distribute** to open the **Distribute** page on the right:



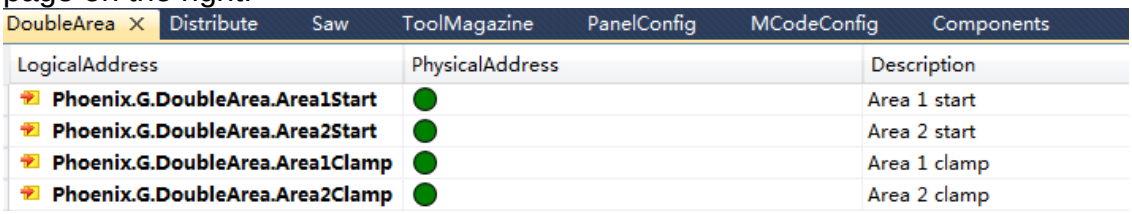

2. Double-click the target **PhysicalAddress** field to specify a physical address (controller and port) for the corresponding logical address.

### 8.3.6 Configure Dual Work Stations

If dual work stations are enabled, the system directly starts machining on work station 2 after machining on work station 1 is finished to save loading and unloading time and improve efficiency.

Follow the steps below to configure dual work stations:

1. In the left **Configuration** list, double-click **DoubleArea** to open the **DoubleArea** page on the right:



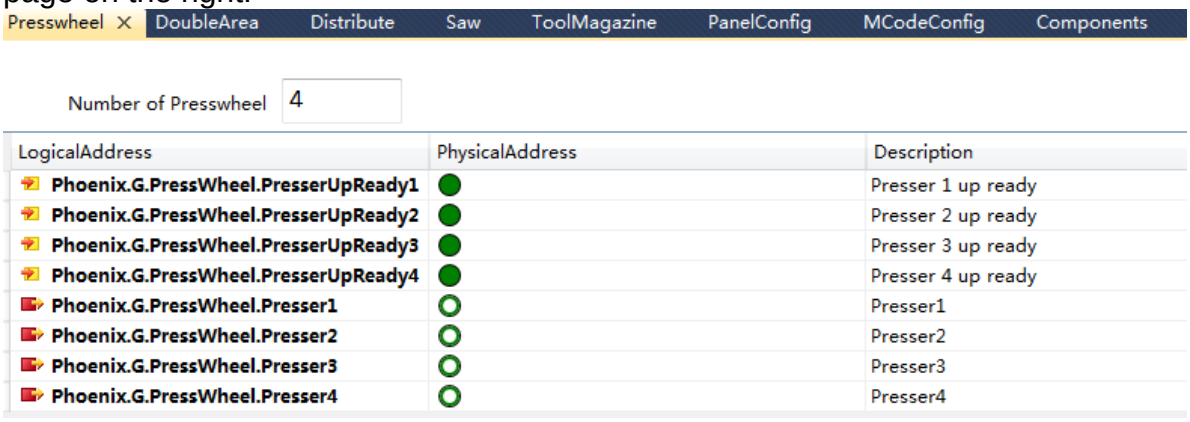

2. Double-click the target **PhysicalAddress** field to specify a physical address (controller and port) for the corresponding logical address.

### 8.3.7 Configure Press Wheel

Press wheels can be used to prevent plates from moving when being processed.

Follow the steps below to configure press wheels:

1. In the left **Configuration** list, double-click **PressWheel** to open the **PressWheel** page on the right:



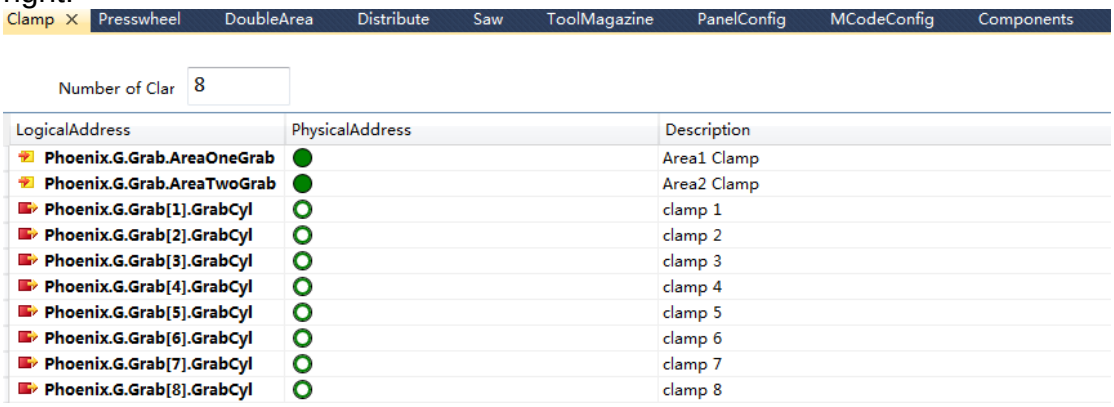

2. Set the parameter **Number of Presswheel**.

3. Double-click the target **PhysicalAddress** field to specify a physical address (controller and port) for the corresponding logical address.

### 8.3.8 Configure Clamps

Follow the steps below to configure clamps:

1. In the left **Configuration** list, double-click **Clamp** to open the **Clamp** page on the right:



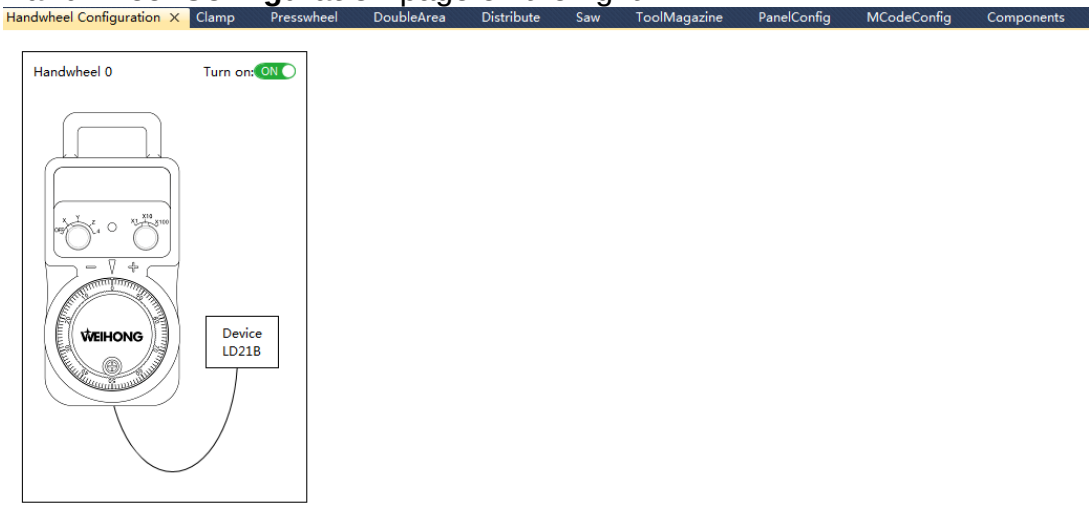

2. Set the parameter **Number of Clamp**.
3. Double-click the target **PhysicalAddress** field to specify a physical address (controller and port) for the corresponding logical address.

### 8.3.9 Configure the Handwheel

You can use a handwheel to control machine movement.

Follow the steps below to configure the handwheel:

1. In the left **Configuration** list, double-click **Handwheel Configuration** to open the **Handwheel Configuration** page on the right:



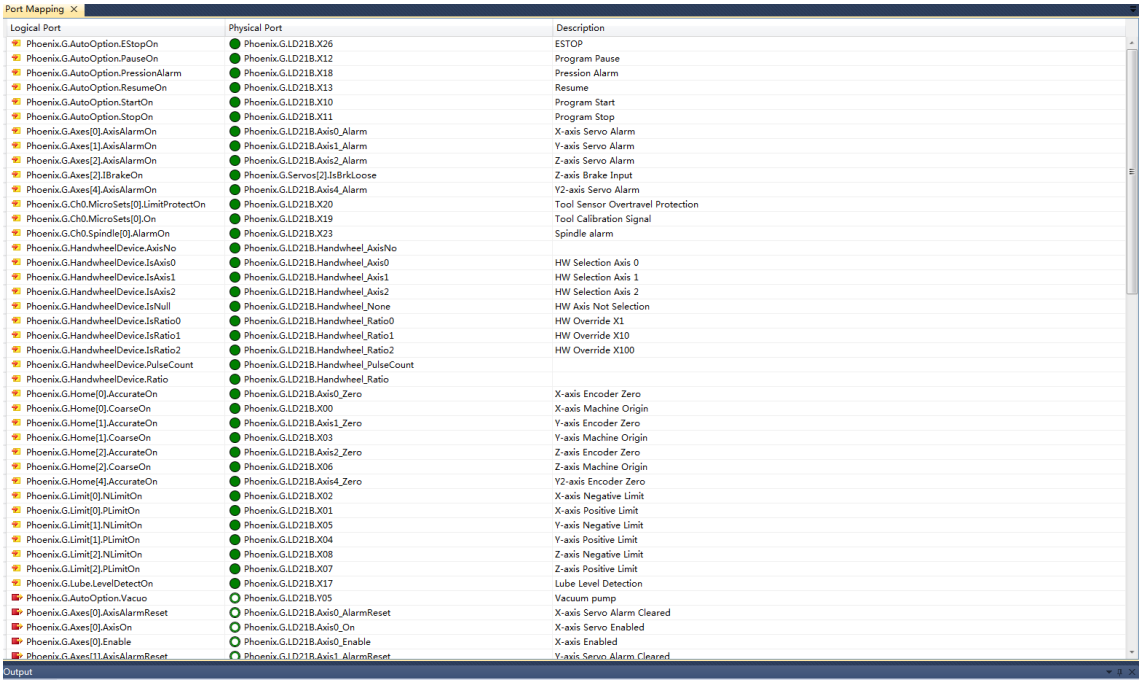

2. Set **Turn on** to **ON**.

## 8.4 Set Port Mapping

You can change the physical address of a logical address based on the port mapping.

Follow the steps below to set port mapping:

1. In the left **Configuration** list, double-click **Port Mapping** to open the **Port Mapping** page on the right:




2. Double-click the target **Physical Port** field to specify a physical address (controller and port) for the corresponding logical address.
3. Select one from the following methods to save the changes:
   - To save changes on the current page, click  under the menu bar.
   - To save changes on all pages, click  under the menu bar.

The * symbol on an open window tab indicates that the changes on the page have not been saved. The * symbol disappears after changes on the page are saved.

## 8.5 Set Port Protection

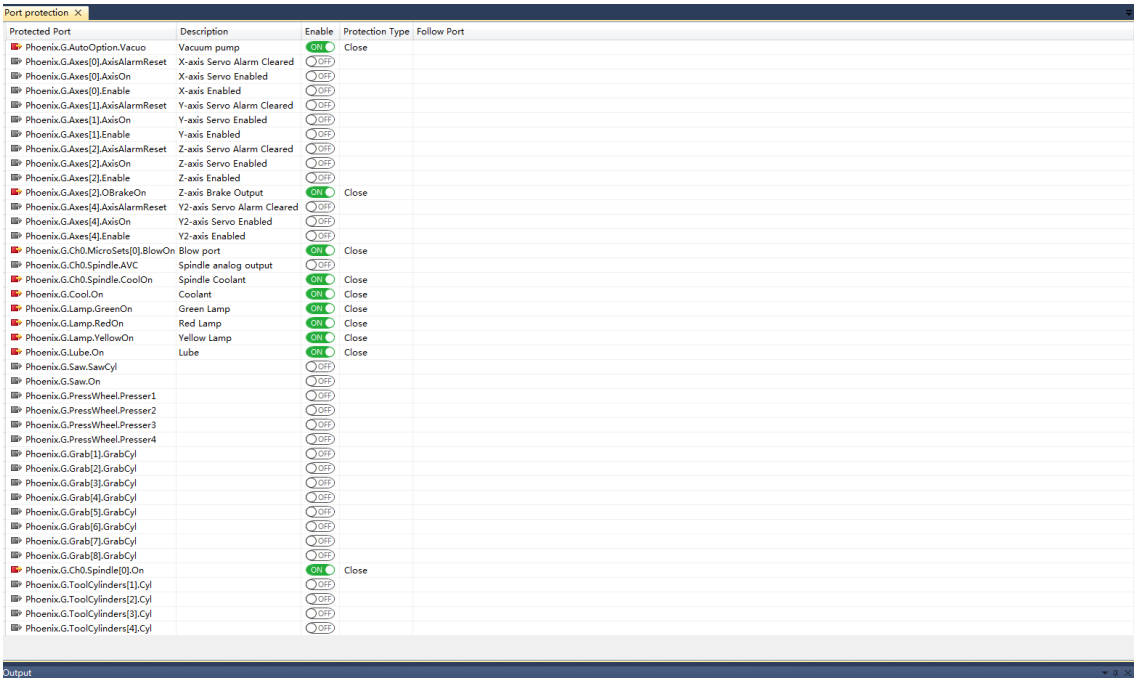
When the software is closed, the protected ports will be automatically closed or opened based on their port protection type.

There are several types of port protection:

- **Open**: The port will be automatically opened when the software is closed.
- **Close**: The port will be automatically closed when the software is closed.
- **Follow**: The protected port status will be turned the same with that of the port specified in the **Follow Port** column.
- **RFollow**: The protected port status will be turned the opposite to that of the port specified in the **Follow Port** column.
- **Hold**: The port state will not be changed when the software is closed.

Follow the steps below to set port protection:

1. In the left **Configuration** list, double-click **Port protection** to open the **Port protection** page on the right:




2. Set the **Enable** switch of the target port to **ON**.
3. Double-click the **Protection Type** field of the target port and select a protection type in the pull-down menu.
4. Select one from the following methods to save the changes:
   - To save changes on the current page, click  under the menu bar.
   - To save changes on all pages, click  under the menu bar.

The * symbol on an open window tab indicates that the changes on the page have not been saved. The * symbol disappears after changes on the page are saved.

## 8.6 Creating Multi-configuration Installation Package

You can create a complete installation package in the current system to keep a backup of the system files and a steady version of system.

Note: A multi-configuration installation package includes all folders whose names are started with "DefaultConfig".

Follow the steps below to create a multi-configuration installation package:

1. In the menu bar, go to **File** > **Multi-Config Pack Up**.
2. Enter a name and storage path for the installation package.
3. Set the parameters as desired.

After generating an installation package, you can find it under the selected path.

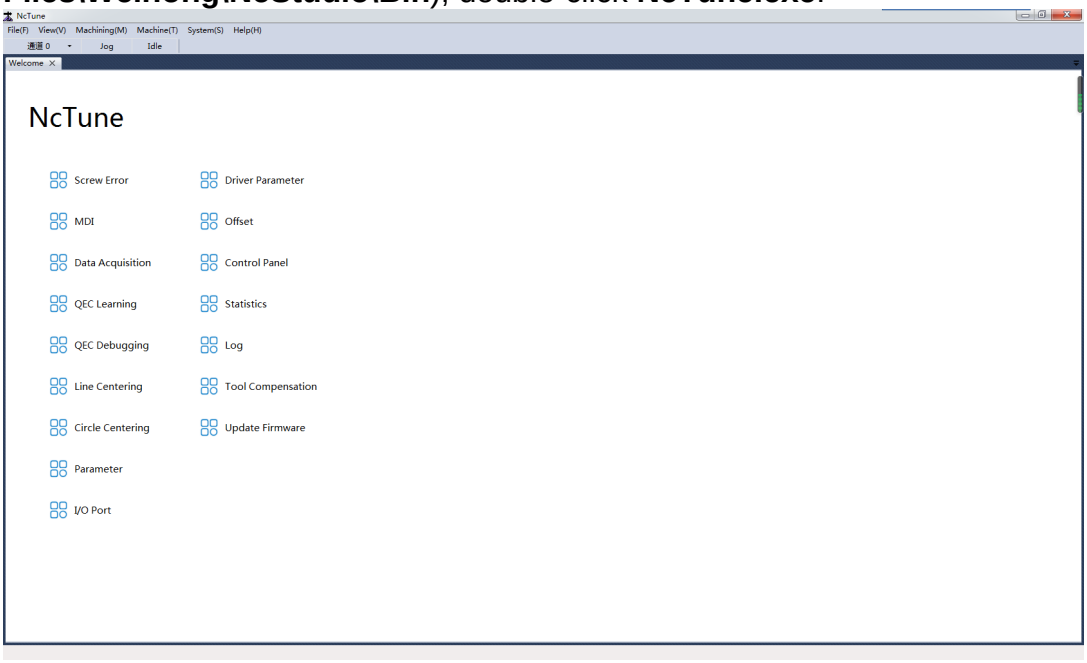
# 9 NcTune

NcTune includes functions that can be used during debugging instead of normal machining.

Functions supported by NcTune are displayed on the welcome page, as shown below, such as setting lead screw error compensation and centering parameters, etc.

Select one from the following methods to open NcTune:

- Open NcStudio. In the menu bar, go to **System** > **NcTune**.
- Close NcStudio. In the NcStudio installation path (**C:\Program Files\Weihong\NcStudio\Bin**), double-click **NcTune.exe**.



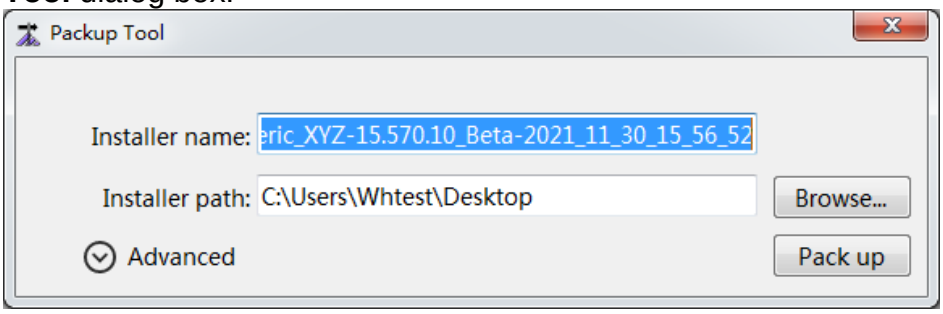
# 10 Other Functions

## 10.1 Generating Installation Package

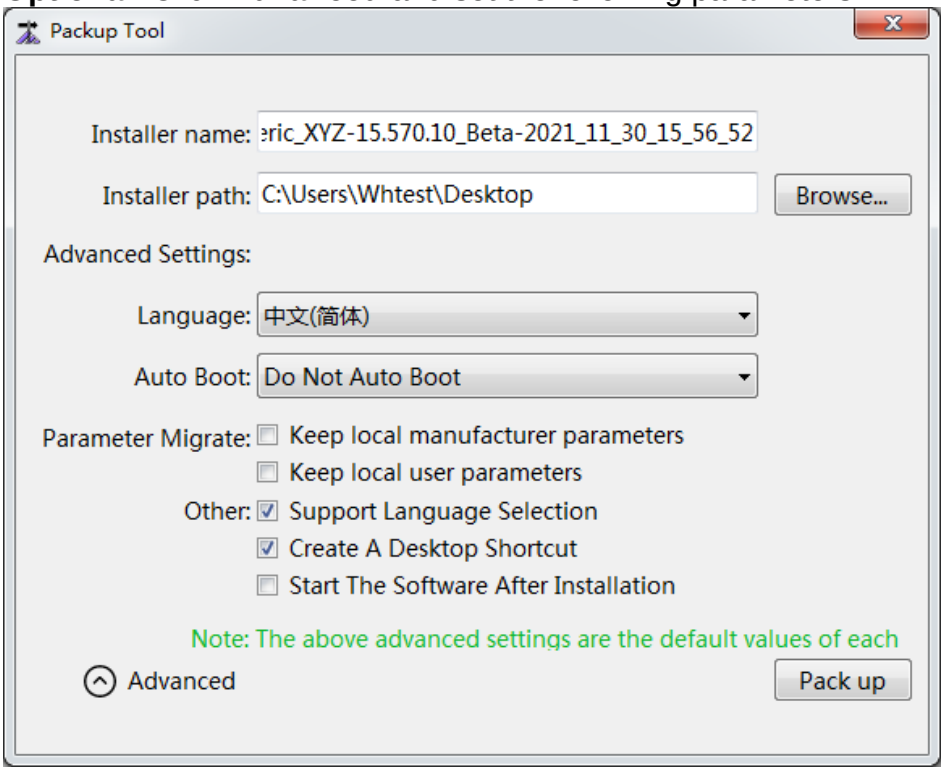
You can create a complete installation package in the current system to have a backup of the system files and a steady version of system.

Follow the steps below to create an installation package:

1. In the menu bar, go to **File** > **Generate Installation Package** to open the **Packup Tool** dialog box:




2. Enter a name for the installation package.
3. Click **Browse** to select a storage path.
4. **Optional**: Click **Advanced** and set the following parameters:




- **Language**: The software supports interface in Chinese and English.
- **Parameter Migrate**
  - **Keep local manufacturer parameters**: Keeps manufacturer parameter settings.
  - **Keep local user parameters**: Keeps user local settings.
- **Auto Boot**: Whether to start the software automatically upon computer system boot.
- **Support Language Selection**: Whether supports switching between Chinese and English interfaces.

- **Create A Desktop Shortcut**: Whether to create a software shortcut on the desktop.
- **Start The Software After Installation**: Whether to automatically start the software after installation is complete.

1. Click **Pack up**.

After generating an installation package, you can find it under the selected path.

## 10.2 Using Handwheel

In **Auto** mode, you can use a handwheel to control machining speed and avoid tool damage caused by loading wrong programs.

A WEIHONG handwheel is shown below:


Follow the steps below to use a WEIHONG handwheel for machining control:

1. In **Auto** mode, select one from the following methods to enter handwheel mode:
   - In the operation button bar, click **HW Guide**.
   - In the menu bar, go to **Machine** > **HW Guide**.
2. Rotate the step knob to select the movement unit (step).
   X1, X10, and X100 represent steps of 0.001 mm, 0.01 mm, and 0.1 mm respectively.
3. Rotate the axis knob to select the target axis.
4. Rotate the handwheel control knob clockwise to run the machining program.
   The program running speed is determined by how fast you rotate the handwheel control dial.

## 10.3 Dual-Y Origin Detection

This function can be used when your machine has two Y axes that use incremental encoders.

Before executing origin detection for the Y axes, you need to manually level the Y axes. Execute origin detection to record the distance between the origins of the Y axes. Use the recorded value when making the Y axes go to the mechanical origin to ensure that the Y axes are still leveled.

Follow the step below to execute origin detection for the Y axes:

In the menu bar, go to **Machine** > **Double Y Origin Detection**.

## 10.4 Jiggle

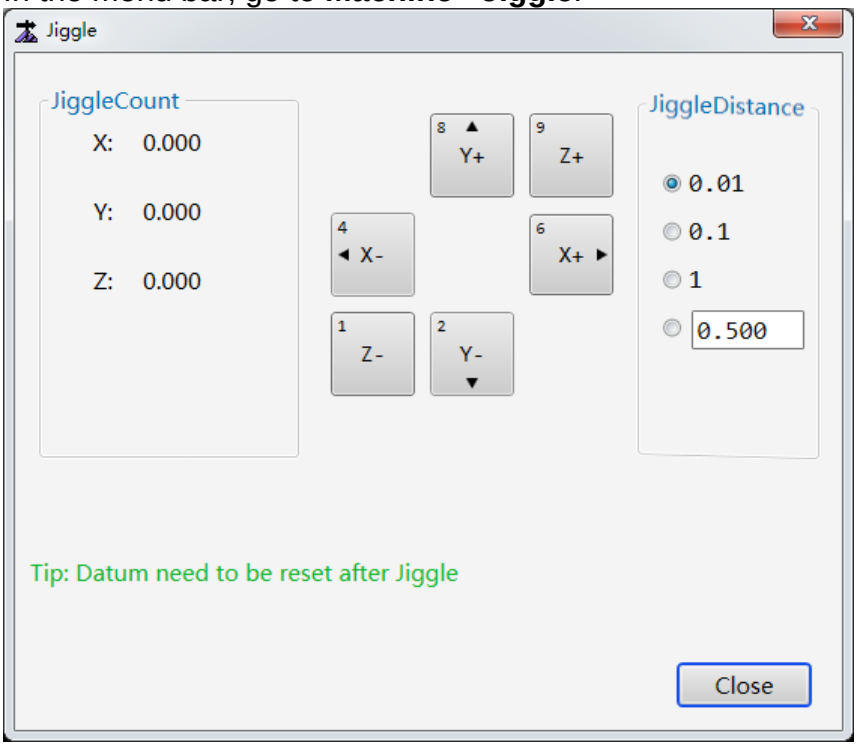
This function is used to adjust the cutter position to find the best cutting distance. Jiggle settings are applicable to the current machining task and will lose effect after machining is stopped.

Before jiggling the cutter:

- Switch to **Auto** mode.
- Ensure that the system is in running or paused state.

Follow the steps below to jiggle the cutter:

1. In the menu bar, go to **Machine** >**Jiggle**:



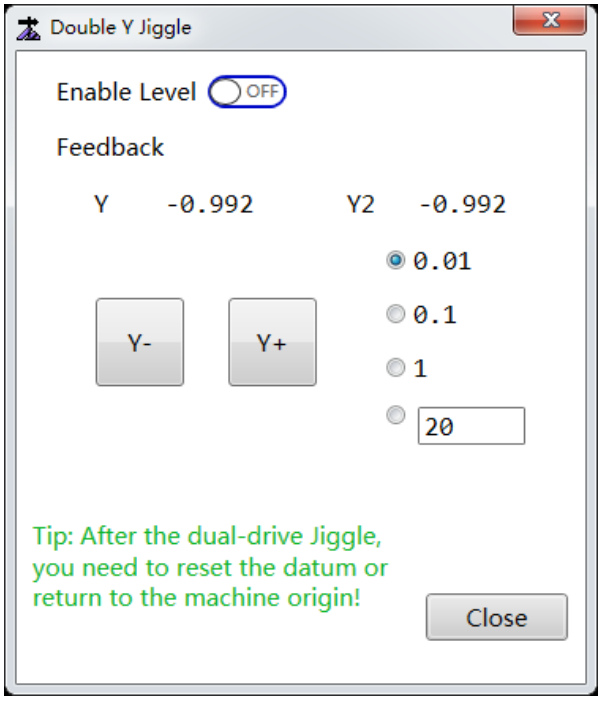

2. In the right **JiggleDistance** area, select or enter a movement unit/step.
3. Click axis buttons in the middle to control the target axis to move in the target direction.

The accumulated movement of the axes are displayed in the left **JiggleCount** area.

## 10.5 Dual-Y Axis Jiggle

You can use this function to adjust the position of a single Y axis or both Y axes.

In the menu bar, go to **Machine** > **Double Y Jiggle**:




- To adjust the positions of the two Y axes at the same time:
  a. Select or enter a movement unit/step on the right side (0.01, 0.1, 1, or other values).
  b. Click **Y-** or **Y+** to move the Y axes in the positive or negative direction by the selected or entered amount.
- To adjust the position of one Y axis:
  a. Set **Enable Level** to **ON**.
  b. Select or enter a movement unit/step on the right side (0.01, 0.1, 1, or other values).
  c. Click **Y-** or **Y+** to move the Y axis in the positive or negative direction by the selected or entered amount. Click **Y2-** or **Y2+** to move the Y2 axis in the positive or negative direction by the selected or entered amount.

The Y-axis and Y2-axis coordinates after jiggling are displayed in the **Feedback** area.

Note: After jiggling the Y axes, you need to set datum or make them go to the mechanical origin.

## 10.6 Calibration

If the motor feedback coordinates are different from the mechanical coordinates, you can manually calibrate the axes to synchronize the actual position of the absolute encoder and update the mechanical coordinates by reading the encoder feedback.

Follow the step below to manually calibrate the axes:

In the menu bar, go to **Machine** > **Adjustment** and select the target axis.

## 10.7 Going to Workpiece Origin

Making the axes go to the workpiece origin means to make the X axis and Y axis move to the workpiece origin from their current positions and make the Z axis move to the safe height (determined by the **Safe Height** parameter value) from its current position.

Follow the step below to make the axes go to workpiece origin:

In the menu bar, go to **Machine** > **To Work Zero**.

## 10.8 Going to Fixed Point

You can control the axes to move to a point with specified mechanical coordinates.

You can change the point mechanical coordinates by modifying the values of **Fixed Point Position(X)**, **Fixed Point Position(Y)**, and **Fixed Point Position(Z)** parameters.

Follow the step below to make the axes move to the point:

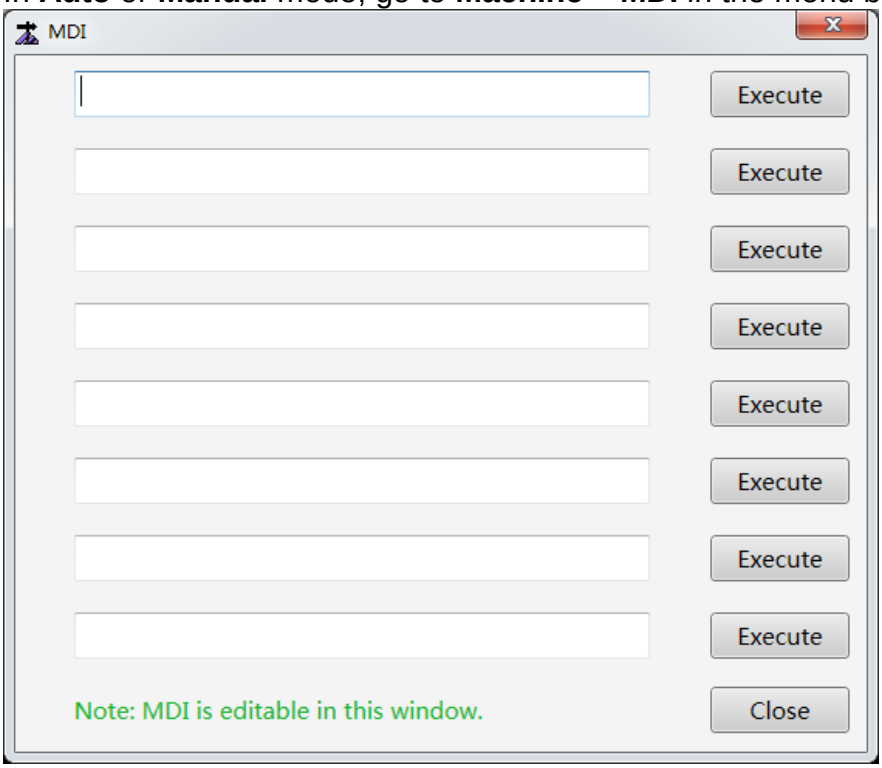
In the menu bar, go to **Machine** > **To Fixed Point**.

## 10.9 MDI

The system supports at most 8 user-entered simple commands to achieve quick movement or machining.

Follow the steps below to use the MDI function:

1. In **Auto** or **Manual** mode, go to **Machine** > **MDI** in the menu bar:




2. Enter a command in a field and click **Execute** to make the system run the entered command.

## 10.10 Clearing Drive Alarms

Some types of drive alarms can be cleared in the software. In some cases, after such alarms are cleared, no other drive alarms will be reported and the system will exit emergency stop state.

Follow the step below to clear drive alarms:

In the menu bar, go to **Machine** > **Clear Servo Alarm**.

## 10.11 Reconnecting to the Controller

In cases such as the controller is powered off and on, you can reconnect to the controller without restarting the software.

Follow the step below to reconnect to the controller:

In the menu bar, go to **Machine** > **Reconnect Controller**.

## 10.12 Switching between Multi-Z Modes

This function is available in software with multiple-Z axis configuration to switch between sequential movement mode and linkage movement mode.

Follow the step below to switch between the modes:

In the menu bar, go to **System** > **Multi-Z Mode Switching**.

# 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.

## 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.

## 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.

## 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.

SPECIALIZED / CONCENTRATED / FOCUSED

Shanghai Weihong Electronic Technology Co., Ltd.

Address: No. 1590, Huhang Rd., Fengxian, Shanghai, China, 201401

Hot-line: 400 882 9188

Website: www.weihong.com.cn/en