

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

Version: 2021.12.01 1st Version Author: Product Testing Department Weihong Corporation All Rights Reserved



Contents

1 System Introduction	1
1.1 Overview	1
1.2 Hardware Connection Diagram	1
1.3 Software Interface	2
1.3.1 NcConfig	2
1.3.2 NcTune	3
1.3.3 NcStudio	4
2 Wiring	9
2.1 Hardware	9
2.1.1 NC60A	9
2.1.2 Lambda Controller	9
2.1.3 EX31A	10
2.2 Operation Panel Ports	13
2.3 Signal Type	13
2.3.1 Switch Value Input Signal	14
2.3.2 Relay Output Signal	14
2.3.3 Relay Output Signal	14
2.3.4 Command Pulse Signal	15
3 Machine Debugging	16
3.1 Overview	16
3.2 Install the Software	16
3.3 Modify I/O Port Polarity	16
3.4 Set Debugging Parameters	17
3.5 Check Axis Direction	18
3.6 Go to Mechanical Origin or Set Datum	19
3.6.1 Making Axes Go to Mechanical Origin	19
3.6.2 Setting Datum	21
3.7 Set Workpiece Origin	21
3.7.1 Clearing	21
3.7.2 Centering	22
3.7.3 Setting Workpiece Offset and Public Offset	23
3.7.4 Tool Calibration	23
3.7.5 Setting Tool Compensation Parameters	26
4 Quick Start	27
4.1 Overview	27
4.2 Loading a Machining File	27
4.3 Running Simulation	27



	4.4 Starting Machining	.28
5	Program-related Functions	.29
	5.1 Program Management	.29
	5.1.1 Loading a Program	.29
	5.1.2 Unloading a Program	.29
	5.1.3 Creating, Editing, Deleting and Renaming a Program	.29
	5.2 Machining Wizard	.29
	5.3 Single Block Machining	.30
	5.4 Selective Machining	.30
	5.5 ENG Tool and Line Selection	.31
	5.6 Advanced Machining	.33
	5.6.1 Array	.33
	5.6.2 Rotation and Mirroring	.34
	5.7 Machining Cycle	.35
6	Check Machining Information and Statistics	.37
7	System Settings	.39
	7.1 Viewing and Setting Parameters	.39
	7.1.1 Global Parameters	.39
	7.1.2 Drive Parameters	.39
	7.1.3 Tool Compensation Parameters	.40
	7.2 Adding Common Parameters	.41
	7.3 Changing Password	.42
	7.4 Switching System Language	.43
	7.5 Data Backup	.43
	7.6 Remote Assistance	.43
	7.7 Using NcGateway	.43
	7.8 M Codes	.43
	7.9 Software Registration	.44
8	NcConfig	.45
	8.1 Opening NcConfig	.45
	8.2 Configuring Device	.45
	8.3 Configuring Machine Parts	.46
	8.3.1 Configure M Codes	.46
	8.3.2 Configure Function Buttons	.47
	8.3.3 Configure the Tool Magazine	.48
	8.3.4 Configuring Saws	.48
	8.3.5 Configure Dual Drive	.49
	8.3.6 Configure Dual Work Stations	.49
	8.3.7 Configure Press Wheel	.49
	8.3.8 Configure Clamps	.50
	8.3.9 Configure the Handwheel	.50
	8.4 Set Port Mapping	.50
	8.5 Set Port Protection	.51
	8.6 Creating Multi-configuration Installation Package	.52
9	NcTune	.53
1() Other Functions	.54
	10.1 Generating Installation Package	.54
	10.2 Using Handwheel	.55
	10.3 Dual-Y Origin Detection	.55
	10.4 Jiggle	.55



10.5 Dual-Y Axis Jiggle	56
10.6 Calibration	
10.7 Going to Workpiece Origin	
10.8 Going to Fixed Point	
10.9 MDI	
10.10 Clearing Drive Alarms	
10.11 Reconnecting to the Controller	
10.12 Switching between Multi-Z Modes	
Legal Notices	60
Requirements of Use	60
Intellectual Property Rights Notice	60
Termination	61



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
 - o Lambda 21A: Applicable to non-bus control systems
 - \circ $\,$ 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 nonbus control system and a bus control system using NC60A are shown below.

Non-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:



NcTune					
File(F) View(V) Machining(M) Machine(T) System(S) Help(H)				
通道 0 - Jog Idle					
Welcome ×			,		
NITURE					
INCLUNE					
Screw Error	Driver Parameter				
00	00				
88 MDI	Offset				
Data Acquisition	Control Panel				
QEC Learning	Statistics				
00 - 0	80				
QEC Debugging	Log				
00	00				
Line Centering	Tool Compensation				
Circle Centering	OO Update Firmware				
Parameter					
80					
I/O Port					
-					

- 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:

Axis < <mark>G54</mark>	WorkCoor	MachCoor
🕀х 📃	0.000	0.000
🗣 📃	0.000	0.000
€ z 📃	0.000	0.000

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 \bigcirc icon will be displayed before the axis in this area.

1.3.3.4 Override Setting Area

The override setting area is shown below:

Feed:	100%	Spindle:	100%	G00:	100%
Set V:	3000	Set V:	12000	Count:	0
V:	0	V:	0		Machining Info

- 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:



Current line No: 0

Finished: 0% Spindle tool No.: 1

Machined Time: 00:00:00

Current Tool No.: 1

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:



Connected to EX31A

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:











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: O no signal is detected; O signal is detected

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

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

Address	Polarit	arity Sampling Description					
nput Port							
LD21B.Handwheel_Axis0	NO	S:1ms	HW Selection Axis 0				
LD21B.Handwheel_Axis1	NO	S:1ms	HW Selection Axis 1				
LD21B.Handwheel_Axis2	NO	S:1ms	HW Selection Axis 2				
LD21B.Handwheel_Ratio0	NO	S:1ms	HW Override X1				
LD21B.Handwheel_Ratio1	NO	S:1ms	HW Override X10				
LD21B.Handwheel_Ratio2	NO	S:1ms	HW Override X100				
LD21B.Axis0_Alarm	NO	S:1ms	X-axis Servo Alarm				
LD21B.Axis1_Alarm	NO	S:1ms	Y-axis Servo Alarm				
LD21B.Axis2_Alarm	NO	S:1ms	Z-axis Servo Alarm				
LD21B.Axis4_Alarm	NO	S:1ms	Y2-axis Servo Alarm				
LD21B.Axis0_Zero	NO	S:1ms	X-axis Encoder Zero				
LD21B.Axis1_Zero	NO	S:1ms	Y-axis Encoder Zero				
LD21B.Axis2_Zero	NO	S:1ms	Z-axis Encoder Zero				
LD21B.Axis4_Zero	NO	S:1ms	Y2-axis Encoder Zero				
LD21B.X00	NO	S:1ms	X-axis Machine Origin				
LD21B.X01	NO	S:1ms	X-axis Positive Limit				
LD21B.X02	NO	S:1ms	X-axis Negative Limit				
LD21B.X03	NO	S:1ms	Y-axis Machine Origin				
LD21B.X04	NO	S:1ms	Y-axis Positive Limit				
LD21B.X05	NO	S:1ms	Y-axis Negative Limit				
LD21B.X06	NO	S:1ms	Z-axis Machine Origin				
LD21B.X07	NO	S:1ms	Z-axis Positive Limit				
LD21B.X08	NO	S:1ms	Z-axis Negative Limit				
LD21B.X10	NO	S:1ms	Program Start				
LD21B.X11	NO	S:1ms	Program Stop				
	NO	Culara					

- 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 a or a, 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**.



A Param				x			
Parameter classification	Name	Value	Unit				
Parameter Overview	1.1 Basic System						
 2 Feed Axis Parameters 	Cycle End Notification Type	0					
▶ 3 Program Parameters	Spindle On at Cycle Start	Yes					
4 Origin Parameters	Spindle Off at Cycle Stop	Yes					
 5 Tool Parameters 6 Comp. Parameters 	Coolant On at Cycle Start	Yes					
 7 Equipment Parameters 	Coolant Off at Cycle Stop	Yes					
	Fixed Point Position(X)	3	mm				
	Fixed Point Position(Y)	2	mm				
	Fixed Point Position(Z)	1	mm				
	Tool Action Options When Cycle Sta	0					
	Tool Action Options When Cycle Cc	0					
	Cycle Machining Times	0					
	Cycle Interval	0	S				
	To work zero need to ensure	Yes					
	1.2 Manual Continuous						
	Max Jog Feedrate before Returning	1200	mm/min				
	Max Joa Feedrate before Returning	1200	mm/min	Ŧ			
Operator •	Name: Cycle End Notification	Туре	Value: 0				
	Desc: Notification types whe on for 3 seconds.	en cycle ends. 0: rec	l light off; 1: red ligl	nt			
Search(S)			Close				

- 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**.



🛣 Param				×
Parameter classification	Name	Value	Unit	^
Parameter Overview	2.1 Shaft Mechanical			
 2 Feed Axis Parameters 	Encoder Type(X)	1		
2.1 Shaft Mechanical	Encoder Type(Y)	1		
2.2 Driver Specifications	Encoder Type(Z)	1		
 3 Program Parameters 4 Origin Parameters 	Axis Direction(X)	1		
 5 Tool Parameters 	Axis Direction(Y)	1		
6 Comp. Parameters	Axis Direction(Z)	1		
7 Equipment Parameters	Axis Direction(Y2)	1		
	Workbench upper limit(X)	400	mm	
	Workbench upper limit(Y)	400	mm	
	Workbench upper limit(Z)	0	mm	
	Workbench lower limit(X)	0	mm	
	Workbench lower limit(Y)	0	mm	
	Workbench lower limit(Z)	-100	mm	
	Check workbench stroke(X)	No		
	Check workbench stroke(Y)	No		
۰ III >	Check workbench stroke(Z)	No		-
Manufacturer 🔹 🔻	Name: Axis Direction(X)		Value: 1	
	Desc: X-axis motion direct direction.	ion. 1: positiv	e direction; -1: negative	
Search(S)			Clo	se

- 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:

🛣 Home	×					
A Note	 To avoid inaccurate position from power failure, etc., execute homing after cycle starts or E-stop occurs. Clear history average if position of origin switch changes. Press F2 to stop machine at once without closing the dialog box. 					
	Please select one of the following to execute homing:					
	All Axes(A) Z, X and Y return to machine origin sequentially.					
	DirectSet(D) If the current position is the same with machine coordinate, you can select "Direct Set". But please ensure: 1. The machine has not been turned off. 2. E-stop has never occurred.					
	X axis(X)Specify a single axis to return to machine origin.Y axis(Y)Z axis(Z)					
🗆 Dialog	box pops up when cycle starts Close					

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 \bigcirc 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.
- Switch to Homing mode. In the menu bar, go to Machine > Datum Setting > Xaxis(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 \bigcirc 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.

Track(Q)	irack(Q) Offset(W) Calibration(E) Program Management(R) Common(T) Log(Y) Port(U)							
Worknie								
workpic	ce onset							
	G54 G54 G	© G55	© G56	© G57	© G58	© G59	Modify	
X:	0	0	0	0	0	0	Set	
Y:	0	0	0	0	0	0	Set	
Z:	0	0	0	0	0	0	Set	
Public C	offset		Deepen	/Lift				
X:	0			Ŀ	0.01	0.1	L	1
Y:	0			D:	0.01	0.1	L	1
Z:	0			Note: modifyir	ng public offset	is effective to	all workpied	e coordinate systems.

- 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.



Track(Q) Offset(W) Calibration(E) Program Management(R) Common(T) Log(Y) Port(U)						
Setting: OFF						
Preview	Basic Setting					
	Active Tool: 1 Change(C)					
z† 🖉	Positioning Times: 1					
	Positioning Speed: 60					
41	Fixed tool setter coordinate 1					
MZ0X	X FixedPoint: 0 Set					
	Y FixedPoint: 0 Set					
Note: θ is the datum line of Machine Origin. H is the tool offset.	Z FixedPoint: 0 Set					
M is the workpiece offset. Z0 is the workpiece surface.	LowestPoint: -80 Set					
Steps: 1. Select the active tool and set the position of	Calibration Mode					
fixed tool setter. 2. Select calibration mode.	Auto Measure(7) One-Click Cali(K)					
One-Click Calibration	Z Mobile Cali(F)					
	Z Manual Set(X)					
SelectAll CancelAll						

- 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
 - o 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 Start
- 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
 Start
- 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 Pause
- 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 Stop
- 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 Resume
- 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**.
 - \circ 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	🛣 Wizard						
	Cir. Contour(A)	Cir. Pocket(B)	Rect. Contour(S)	Rect. Pocket(T) Screw Cor	np Wizard(W)		
	Ϋ́Ι, s			© Outer	⊚ Inner		
l		\backslash		Part Dia d:	200		
	X		/	Start X:	0		
			– d	Start Y:	0		
	•	Y	→ _x	Z Depth h:	0.1		
		<u>t</u>		Cut Depth H:	0.1		
	h ŧ			Tool Dia D:	3		
	нŢ						
					Ð	œcute	

- 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:
 - $\circ~$ 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:



🛣 Selective MC
Current R: 0
Total Row: 19
Start Row: 1
End Row: 19
Reset(R) OK Cancel

- 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:
 - \circ Press the F9 key.



- In the operation button bar, click
- \circ 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:

🛣 ENG (Choose		
	File name:	D:\文档\完成	式7木工-Phoenix雕刻系统三轴厂商手册-写号类型C3 F
	Version:	5.01	Tools Count: 3
C	urrent No.:		
	Select No.:	OFF	Modify No.: OFF
No.	Start Line	End Line	Tool Name
1	1	66	平底JD-6.00
2	67	84	平底JD-4.00
3	85	122	平底JD-10.00
-Choose line process Start Roy		ow: 1	End Row: 122
			OK Cancel

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

🛣 EN	NG Choose				×		
	File name:	D:\文档\	D:\文档\完成7木工-Phoenix雕刻系统三轴厂商手册-写号类型C3系				
	Version: 5.01		Tools Count: 3				
	Current No.:	1	1				
	Select No.:	ON	Modify No.: OFF				
		No.	Start Line	End Line	Tool Name		
	1		1	66	平底JD-6.00		
	2		67	84	平底JD-4.00		
	3		85	122	平底JD-10.00		
Choose line processing Start Row: 1 End Row: 122							
					OK Cancel		

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:



	ENG Choose						
	File name:	D:\文档\完成 7	木工-Phoer	ix雕刻系统	三轴厂商手册-写号类型C3 F		
Version:		5.01 Tools Count: 3					
	Current No.:	1	1				
	Select No.:			Modify N	Io.: ON O		
	No.	Modify No	Start Line	End Line	Tool Name		
	1	1	1	66	平底JD-6.00		
	2	2	67	84	平底JD-4.00		
	3	3	85	122	平底JD-10.00		
Choose line proces		osing ow: 1	End	Row: 122			
					OK Cancel		

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:



🛣 AdvAuto		X
Array		Mirror & Rotate
		× 5 30 × ×
Array Mode:	Convention •	Machining Mode: Convention
Rows:	1	Rotate Center X: 0.000
Columns:	1	Rotate Center Y: 0.000
Row Space R:	0.000	□ Mirror(M)
Column Space C:	0.000	Eccentricity L: 0.000
		OK Cancel

- 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:



AdvAuto			×	
Array		Mirror & Rotate		
		× 550 × ×		
Array Mode:	Convention •	Machining Mode:	Convention 🔹	
Rows:	1	Rotate Center X:	0.000	
Columns:	1	Rotate Center Y:	0.000	
Row Space R:	0.000		Mirror(M)	
Column Space C:	0.000	Eccentricity L:	0.000	
		O	K Cancel	

- 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:



Z Cycle Setting
Cycles: 0 Interval: 0
Cycle Index: 0
ОК

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 Information			×					
File Name: 5.2多刀具.ENG								
Machined Time: 00	:00:37	Cycles: 1/0						
Cut Time: 00	:00:32	Finished: 100	0%					
Tip: the above time is	the actual machir	ning time.						
Movement Range								
Axis	Min	Max	Difference					
Χ:	-21.999	21.999	43.999					
Y:	-21.999	21.999	43.999					
Z:	-2.000	1.000	3.000					
Machining Range —								
Axis	Min	Max	Difference					
X:	-21.999	21.999	43.999					
Y:	-21.999	21.999	43.999					
Z:	-2.000	-1.000	1.000					
Tool Travel Length								
G00:	69.052	G01:	1001.682					
G02:	41.751	G03:	330.589					
G01+G02+G03:	1374.023							
MachCoor Mc			ОК					

Machining Statistics



A Machining Statistics	Machining Statistics						
Search filter	ID	Status	Name	StartProcessTime	EndProcessTime	Consuming	Interval
Name	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
StartTime Saturday, Nover	1	Abnormal	铣矩形边框.nc	2021-11-25 15:54:58	2021-11-25 16:01:39	00:06:41	
EndTime Tuesday, Novem							
Total 2items Total Consuming : 00:07:18 Avg Consuming : 00:03:39							
Clear data Before Select a date Clear							
4	Total 2ite	ems , 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:

Parameter classification	Name	Value	Unit	-		
Parameter Overview	1.1 Basic System					
 1 Operating Parameters 2 Feed Axis Parameters 	Cycle End Notification Type	0				
3 Program Parameters	Spindle On at Cycle Start	Yes				
4 Origin Parameters	Spindle Off at Cycle Stop	Yes				
5 Tool Parameters	Coolant On at Cycle Start	Yes				
 7 Equipment Parameters 	Coolant Off at Cycle Stop	Yes				
	Fixed Point Position(X)	3	mm			
	Fixed Point Position(Y)	2	mm			
	Fixed Point Position(Z)	1	mm			
	Tool Action Options When Cycle Sta	0				
	Tool Action Options When Cycle Cc	0				
	Cycle Machining Times	0				
	Cycle Interval	0	s			
	To work zero need to ensure	Yes				
	1.2 Manual Continuous					
	Max Jog Feedrate before Returning	1200	mm/min			
	Max Joa Feedrate before Returning	1200	mm/min	-		
Operator 🔹	Name: Cycle End Notification	Туре	Value: 0			
	Desc: Notification types whe on for 3 seconds.	n cycle ends. 0:	red light off; 1: rec	l light		
Search(S)				lose		

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:

🛣 Driver Param	1					×
Param Num.	Param Name	Х	Y	Z	Y2	
Pr0.00	Rotational direction setup	0	0	0	0	A
Pr0.01	Control mode setup	0	0	0	0	E
Pr0.02	Real-time auto-gain tuning s	0	0	0	0	
Pr0.03	Selection of machine stiffne	16	16	16	16	
Pr0.04	Inertia ratio	100	100	100	100	
Pr0.08	Command pulse counts per one	0	0	0	0	
Pr0.09	1st numerator of electronic	0	0	0	0	
Pr0.10	Denominator of electronic g ε	0	0	0	0	
Pr0.11	Output pulse counts per one	0	0	0	0	
Pr0.13	1st torque limit	0	0	0	0	
Pr0.14	Position deviation excess $\boldsymbol{s}\boldsymbol{\varepsilon}$	0	0	0	0	
Pr0.15	Absolute encoder setup	0	0	0	0	
Pr0.16	External regenerative resist	0	0	0	0	
Pr0.17	Heat transfer coefficient of	0	0	0	0	
Pr0.18	Regenerative resistor capaci	0	0	0	0	
Pr0.19	Regenerative resistor resist	0	0	0	0	
Pr1.00	1st gain of position loop	0	0	0	0	-
Number: Value: Range:	N Effe	ame: Unit: ctive				
WSLM	• Show Comr	Refre	esh(R) Im	port(D)	Export(F)	Reset(H)

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.
 - o 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:

🛣 Tool Compe	ensation				x
Offset No	Tool Diameter	Diameter Wear	Tool Length	Length Wear	*
1	0.000	0.000	0.000	0.000	
2	0.000	0.000	0.000	0.000	
3	0.000	0.000	0.000	0.000	
4	0.000	0.000	0.000	0.000	
5	0.000	0.000	0.000	0.000	
6	0.000	0.000	0.000	0.000	
7	0.000	0.000	0.000	0.000	
8	0.000	0.000	0.000	0.000	
9	0.000	0.000	0.000	0.000	
10	0.000	0.000	0.000	0.000	E
11	0.000	0.000	0.000	0.000	
12	0.000	0.000	0.000	0.000	
13	0.000	0.000	0.000	0.000	
14	0.000	0.000	0.000	0.000	
15	0.000	0.000	0.000	0.000	
16	0.000	0.000	0.000	0.000	
17	0.000	0.000	0.000	0.000	
18	0.000	0.000	0.000	0.000	
19	0.000	0.000	0.000	0.000	
20	0.000	0.000	0.000	0.000	
21	0.000	0.000	0.000	0.000	
22	0.000	0.000	0.000	0.000	
23	0.000	0.000	0.000	0.000	
24	0.000	0.000	0.000	0.000	
25	0.000	0.000	0.000	0.000	
26	0 000	0 000	0 000	0 000	Ŧ
				Clos	e

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:



🛣 Usually Param		×
Manufacturer Name]	Common Parameter List:
Name	1	Name
Public Offset(X)	J	Axial G00 Speed(X)
Public Offset(Y)		Axial G00 Speed(Y)
Public Offset(Z)		Axial G00 Speed(Z)
G54 Workpiece Offset(X)	Add(A)	Feedrate
G54 Workpiece Offset(Y)	, ladini	Jog Feedrate(X)
G54 Workpiece Offset(Z)	Delete(D)	Jog Feedrate(Y)
G55 Workpiece Offset(X)		Jog Feedrate(Z)
G55 Workpiece Offset(Y)	Up(U) ^	Rapid Jog Feedrate(X)
G55 Workpiece Offset(Z)		Rapid Jog Feedrate(Y)
G56 Workpiece Offset(X)	Down(N) v	Rapid Jog Feedrate(Z)
G56 Workpiece Offset(Y)		Max Jog Feedrate before Returning to R
G56 Workpiece Offset(Z)		Max Jog Feedrate before Returning to R
G57 Workpiece Offset(X)		Max Jog Feedrate before Returning to R
G57 Workpiece Offset(Y)		
G57 Workpiece Offset(Z)		
G58 Workpiece Offset(X)		
G58 Workpiece Offset(Y)		
G58 Workniece Offset/7) *		
		OK Close

- 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:

🛣 Modify Password	×
Old Password: Set New Password :	
Confirm Password:	
	Modify Cancel



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**:

EMD	×
Device number	
WHNS-PM85-509A4C-0E157F-000	
Registration code	
Tip: Please notify the supplier to generate the registration code or use the "Apply for Registration Code" fun	ction to apply for it.
Apply for	r registered

- 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:

Topology Configuration	×
🔺 📽 Lambda Control Chain	Add
■LD21B	
🖫 Panel control chain	Replace
	Delete
	Move Up
	Move Down
Confirm	Cancel

- 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.

S NcConfig		×
File(F) Edit(E) View(V) Debug(D) System(S) Tools(T	D Window(W) Help(H)	
응답값(응답 지수 전) 비용 한 한 대	efault layout + 🔄 + 😒 🚍 🖂 😂 😂 🗶 (Clean Up) 🔎 🛟 Param upgrade 🚍 ConfigManagement	
Configuration(Default setting)	Conservative V	-
A 2 NeStudio		
A Blandware Connection and Configuration	Distribute	
4 ⁴⁹ Topology Configuration	Distribute	
4 ^{or} Lambda Control Chain	✓ Distribute	
- LD218		
Papel control chain	Component	
Port Mapping	Component	
Port protection	DoubleArea V Clamp V PressWheel Saw V Multitasking	
A III Machine Parts		
M MCodeConfig		
E PanelConfig	ToolMagazine	
[""] Distribute	······································	
III DoubleArea	NoneMagazine OLinerMagazine OliskMagazine ServoToolMagazine OCylinder	
III PressWheel		
Clamp		
ToolMagazine		
🕲 Saw		
Handwheel Configuration		
1		
1	Output	* 0 ×
	Build • 😿 🔤	
Project Configuration(Default setting)	ataggagjXXH"C: VProgram Files (Weinong VicStudio VicContig (data VicContig)(LUDesc.ctg",	1
Ready		- 50

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

page on the ligi	
MCodeConfig X Components*	-
Reset	
Functional class	
MCode	Function
	Press wheel on
	Press wheel off
	Saw on
	Saw off
	Saw blade retracted
	Saw blade swings in X direction
	Saw blade swings in Y direction
ValvesClass	
MCode	Function

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:

In the left Configuration list, double-click PanelConfig to open the PanelConfig page on the right:
 PanelConfig × MCodeConfig × Components*

自动页面	手动页面
•	• • •
• • •	• • •
ToFixedPos 🔹 🔹	ToFixedPos •
▼ Cool ▼ Vacuo ▼ Lube ▼	Spindl eSwitc • h

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:

 In the left Configuration list, double-click ToolMagazine to open the ToolMagazine page on the right: <u>ToolMagazine × PanelConfig Components*</u>

There is no need to set parameters and ports for manual tool change.				
LogicalAddress	PhysicalAddress	Description		
Phoenix.G.Ch0.Spindle[0].ZeroVelocitySignal		NcConfigAddins.Port.Spindle.ZeroSpeed		
Phoenix.G.Ch0.Spindle[0].On	0	NcConfigAddins.Port.Spindle.On		

- 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:

saw X Toolwagazine PanelCon	ing incodecoming components	
LogicalAddress	PhysicalAddress	Description
🕫 Phoenix.G.Saw.SawDriverAlarm		Saw driver alarm
🕫 Phoenix.G.Saw.SawCylUpReach		Saw cyl up ready
🕫 Phoenix.G.Saw.SawCylDownReach		Saw cyl down ready
Phoenix.G.Saw.SawCyl	0	SawCyl
Phoenix.G.Saw.On	0	SawOn



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:

<mark>Distribute X</mark> Saw ToolMa	gazine PanelConfig	MCodeConfig	Components
LogicalAddress	PhysicalAddress		Description
✤ Phoenix.G.Home[4].CoarseO	n 🔴		Y2 Machine Origin

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:

DoubleArea X Distribute Sav	v ToolMagazine	PanelConfig	MCodeConfig	Components
LogicalAddress	PhysicalAddress		D	escription
🕫 Phoenix.G.DoubleArea.Area19	tart 🔵		Ar	ea 1 start
🔊 Phoenix.G.DoubleArea.Area29	tart 🔵		Ar	ea 2 start
🔊 Phoenix.G.DoubleArea.Area10	lamp 🔵		Ar	ea 1 clamp
🕫 Phoenix.G.DoubleArea.Area20	lamp 🔵		Ar	ea 2 clamp

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:

Presswheel × DoubleArea	Distribute	Saw	ToolMagazine	PanelConfig	MCodeConfig	Components
Number of Presswheel	4					
LogicalAddress		Physical	Address		Description	
🕫 Phoenix.G.PressWheel.Pres	sserUpReady1				Presser 1 up rea	ady
🔁 Phoenix.G.PressWheel.Pres	sserUpReady2				Presser 2 up rea	ady
😕 Phoenix.G.PressWheel.Pres	sserUpReady3				Presser 3 up rea	ady
😕 Phoenix.G.PressWheel.Pres	sserUpReady4				Presser 4 up rea	ady
Phoenix.G.PressWheel.Pres	sserl	0			Presser1	
Phoenix.G.PressWheel.Pres	sser2	0			Presser2	
Phoenix.G.PressWheel.Pres	sser3	0			Presser3	
Phoenix.G.PressWheel.Pres	sser4	0			Presser4	

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: <u>Clamp × Presswheel DoubleArea Distribute Saw ToolMagazine PanelConfig MCodeConfig Components</u>

Numb	er of Clar	8		
gicalAddr	ress			PhysicalAddress
Phoenix	x.G.Grab.A	rea One Grab	۲	
Phoenix	x.G.Grab.A	reaTwoGrab		
Phoenix	x.G.Grab[1]].GrabCyl	0	
Phoenix	x.G.Grab[2]].GrabCyl	0	
Phoenix	x.G.Grab[3]].GrabCyl	0	
Phoenix	x.G.Grab[4]].GrabCyl	0	
Phoenix	x.G.Grab[5]].GrabCyl	0	
Phoenix	x.G.Grab[6]].GrabCyl	0	
Phoenix	x.G.Grab[7]].GrabCyl	0	
Phoenix	x.G.Grab[8]].GrabCyl	0	

- 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:



Port Mapping ×		
Logical Port	Physical Port	Description
Phoenix.G.AutoOption.EStopOn	Phoenix.G.LD21B.X26	ESTOP
Phoenix.G.AutoOption.PauseOn	Phoenix.G.LD21B.X12	Program Pause
Phoenix.G.AutoOption.PressionAlarm	Phoenix.G.LD21B.X18	Pression Alarm
Phoenix.G.AutoOption.ResumeOn	Phoenix.G.LD21B.X13	Resume
Phoenix.G.AutoOption.StartOn	Phoenix.G.LD21B.X10	Program Start
Phoenix.G.AutoOption.StopOn	Phoenix.G.LD21B.X11	Program Stop
Phoenix.G.Axes[0].AxisAlarmOn	Phoenix.G.LD21B.Axis0_Alarm	X-axis Servo Alarm
Phoenix.G.Axes[1].AxisAlarmOn	Phoenix.G.LD21B.Axis1 Alarm	Y-axis Servo Alarm
Phoenix.G.Axes[2].AxisAlarmOn	Phoenix.G.LD21B.Axis2_Alarm	Z-axis Servo Alarm
Phoenix.G.Axes[2].IBrakeOn	Phoenix.G.Servos[2].IsBrkLoose	Z-axis Brake Input
Phoenix.G.Axes[4].AxisAlarmOn	Phoenix.G.LD21B.Axis4_Alarm	Y2-axis Servo Alarm
Phoenix.G.Ch0.MicroSets[0].LimitProtectOn	Phoenix.G.LD21B.X20	Tool Sensor Overtravel Protection
Phoenix.G.Ch0.MicroSets[0].On	Phoenix.G.LD21B.X19	Tool Calibration Signal
Phoenix.G.Ch0.Spindle[0].AlarmOn	Phoenix.G.LD21B.X23	Spindle alarm
Phoenix.G.HandwheelDevice.AxisNo	Phoenix.G.LD21B.Handwheel_AxisNo	
Phoenix.G.HandwheelDevice.IsAxis0	Phoenix.G.LD21B.Handwheel_Axis0	HW Selection Axis 0
Phoenix.G.HandwheelDevice.IsAxis1	Phoenix.G.LD21B.Handwheel_Axis1	HW Selection Axis 1
Phoenix.G.HandwheelDevice.IsAxis2	Phoenix.G.LD21B.Handwheel Axis2	HW Selection Axis 2
Phoenix.G.HandwheelDevice.IsNull	Phoenix.G.LD21B.Handwheel_None	HW Axis Not Selection
Phoenix.G.HandwheelDevice.IsRatio0	Phoenix.G.LD21B.Handwheel_Ratio0	HW Override X1
Phoenix.G.HandwheelDevice.IsRatio1	Phoenix.G.LD21B.Handwheel_Ratio1	HW Override X10
Phoenix.G.HandwheelDevice.IsRatio2	Phoenix.G.LD21B.Handwheel_Ratio2	HW Override X100
Phoenix.G.HandwheelDevice.PulseCount	Phoenix.G.LD21B.Handwheel_PulseCount	
Phoenix.G.HandwheelDevice.Ratio	Phoenix.G.LD21B.Handwheel_Ratio	
Phoenix.G.Home[0].AccurateOn	Phoenix.G.LD21B.Axis0_Zero	X-axis Encoder Zero
Phoenix.G.Home[0].CoarseOn	Phoenix.G.LD21B.X00	X-axis Machine Origin
Phoenix.G.Home[1].AccurateOn	Phoenix.G.LD21B.Axis1_Zero	Y-axis Encoder Zero
Phoenix.G.Home[1].CoarseOn	Phoenix.G.LD21B.X03	Y-axis Machine Origin
Phoenix.G.Home[2].AccurateOn	Phoenix.G.LD21B.Axis2_Zero	Z-axis Encoder Zero
Phoenix.G.Home[2].CoarseOn	Phoenix.G.LD21B.X06	Z-axis Machine Origin
Phoenix.G.Home[4].AccurateOn	Phoenix.G.LD21B.Axis4_Zero	Y2-axis Encoder Zero
Phoenix.G.Limit[0].NLimitOn	Phoenix.G.LD21B.X02	X-axis Negative Limit
Phoenix.G.Limit[0].PLimitOn	Phoenix.G.LD21B.X01	X-axis Positive Limit
Phoenix.G.Limit[1].NLimitOn	Phoenix.G.LD21B.X05	Y-axis Negative Limit
Phoenix.G.Limit[1].PLimitOn	Phoenix.G.LD21B.X04	Y-axis Positive Limit
Phoenix.G.Limit[2].NLimitOn	Phoenix.G.LD21B.X08	Z-axis Negative Limit
Phoenix.G.Limit[2].PLimitOn	Phoenix.G.LD21B.X07	Z-axis Positive Limit
Phoenix.G.Lube.LevelDetectOn	Phoenix.G.LD21B.X17	Lube Level Detection
Phoenix.G.AutoOption.Vacuo	O Phoenix.G.LD21B.Y05	Vacuum pump
Phoenix.G.Axes[0].AxisAlarmReset	O Phoenix.G.LD21B.Axis0_AlarmReset	X-axis Servo Alarm Cleared
Phoenix.G.Axes[0].AxisOn	O Phoenix.G.LD21B.Axis0_On	X-axis Servo Enabled
Phoenix.G.Axes[0].Enable	O Phoenix.G.LD21B.Axis0_Enable	X-axis Enabled
Phoenix.G.Axes[1].AxisAlarmReset	Phoenix.G.I.D.21B.Axis1_AlarmReset	Y-axis Servo Alarm Cleared
Dutput		

- 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 **w** under the menu bar.
 - To save changes on all pages, click when 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:



Port protection ×				
Protected Port	Description	Enable	Protection Type	Follow Port
Phoenix.G.AutoOption.Vacuo	Vacuum pump	ON	Close	
Phoenix.G.Axes[0].AxisAlarmReset	X-axis Servo Alarm Cleared	OOFF)		
Phoenix.G.Axes[0].AxisOn	X-axis Servo Enabled	OOFF		
Phoenix.G.Axes[0].Enable	X-axis Enabled	OOFF)		
Phoenix.G.Axes[1].AxisAlarmReset	Y-axis Servo Alarm Cleared	O OFF		
Phoenix.G.Axes[1].AxisOn	Y-axis Servo Enabled	OOFF		
Phoenix.G.Axes[1].Enable	Y-axis Enabled	O OFF		
Phoenix.G.Axes[2].AxisAlarmReset	Z-axis Servo Alarm Cleared	OOFF		
Phoenix.G.Axes[2].AxisOn	Z-axis Servo Enabled	OOFF		
Phoenix.G.Axes[2].Enable	Z-axis Enabled	OOFF		
Phoenix.G.Axes[2].OBrakeOn	Z-axis Brake Output	ON	Close	
Phoenix.G.Axes[4].AxisAlarmReset	Y2-axis Servo Alarm Cleared	OOFF		
Phoenix.G.Axes[4].AxisOn	Y2-axis Servo Enabled	OOFF		
Phoenix.G.Axes[4].Enable	Y2-axis Enabled	OOFF		
Phoenix.G.Ch0.MicroSets[0].BlowOn	Blow port	ON	Close	
Phoenix.G.Ch0.Spindle.AVC	Spindle analog output	OOFF		
Phoenix.G.Ch0.Spindle.CoolOn	Spindle Coolant	ON	Close	
Phoenix.G.Cool.On	Coolant	ON	Close	
Phoenix.G.Lamp.GreenOn	Green Lamp	ON	Close	
Phoenix.G.Lamp.RedOn	Red Lamp	ON	Close	
Phoenix.G.Lamp.YellowOn	Yellow Lamp	ON	Close	
Phoenix.G.Lube.On	Lube	ON	Close	
Phoenix.G.Saw.SawCyl		OOFF		
Phoenix.G.Saw.On		O OFF		
Phoenix.G.PressWheel.Presser1		OOFF		
Phoenix.G.PressWheel.Presser2		O OFF		
Phoenix.G.PressWheel.Presser3		OOFF		
Phoenix.G.PressWheel.Presser4		OOFF		
Phoenix.G.Grab[1].GrabCyl		OOFF		
Phoenix.G.Grab[2].GrabCyl		OOFF		
Phoenix.G.Grab[3].GrabCyl		OOFF		
Phoenix.G.Grab[4].GrabCyl		OOFF		
Phoenix.G.Grab[5].GrabCyl		OOFF)		
Phoenix.G.Grab[6].GrabCyl		OOFF		
Phoenix.G.Grab[7].GrabCyl		OOFF		
Phoenix.G.Grab[8].GrabCyl		OOFF		
Phoenix.G.Ch0.Spindle[0].On		ON	Close	
Phoenix.G.ToolCylinders[1].Cyl		OOFF)		
Phoenix.G.ToolCylinders[2].Cyl		OOFF		
Phoenix.G.ToolCylinders[3].Cyl		OOFF)		
Phoenix.G.ToolCylinders[4].Cyl		OOFF		
2010/01				▼ # A

- 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 **and** under the menu bar.
 - To save changes on all pages, click **I 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.

File(F) View(V) Machining(M) Machine(T 通道 0 - Jog Idle	stem(S) Help00	
NcTune		l
Screw Error	B Driver Parameter	
D MDI	Offset	
Data Acquisition	Control Panel	
QEC Learning	Bo Statistics	
QEC Debugging	🖰 Log	
Line Centering	Col Compensation	
Circle Centering	😓 Update Firmware	
Parameter		
I/O Port		



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:

🛣 Packup Tool	×
Installer name: pric_XYZ-15.570.10_Beta-2021_11_30_15_56_52	
Installer path: C:\Users\Whtest\Desktop	Browse
⊘ Advanced	Pack up

- 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:

🛣 Packup Tool		×
Installer name:	eric_XYZ-15.570.10_Beta-2021_11_30_15_56_52	
Installer path:	C:\Users\Whtest\Desktop	Browse
Advanced Settings:		
Language:	中文(简体)	
Auto Boot:	Do Not Auto Boot 🔹	
Parameter Migrate:	Keep local manufacturer parameters	
[Keep local user parameters	
Other:	Support Language Selection	
[Create A Desktop Shortcut	
[Start The Software After Installation	
Note:	The above advanced settings are the default va	alues of each
 Advanced 		Pack up

- 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.
- 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:



🛣 Double Y Jiggle	×
Enable Level OFF	
Feedback	
Y -0.992	Y2 -0.992
	0.01
V- V+	◎0.1
	◎ 1
	◎ 20
Tip: After the dual-drive Ji you need to reset the date return to the machine orig	ggle, um or gin! Close

- 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:

之 MDI	x
	Execute
Note: MDI is editable in this window.	Close

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 untransferred;
- 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