Contents

Chapter I Overview

Welcome to use our L68 control system. The Manual introduces the functions of the control system in details and is accompanied by a large number of examples and charts to illustrate. Before using the carving machine or cutting machine, please read the Operating Manual carefully to ensure correct use and prevent accidents. Please keep the Manual properly for easy reference at any time.

The system adopts an industrial control host + L68 controller mode, so a PC needs to be configured. The system is easy to operate, easy to learn and understand, easy to install, and has a small volume, which is suitable for stone carving and aluminum plate cutting.

1.1 Safety precautions for use

Do not use this product in the environments with strong interference and magnetic field;

Do not plug or unplug the power supply of the operation box with electricity;

Pay attention to waterproofing, dust prevention, and fire prevention;

Prevent conductive substances (such as metals, etc.) from entering the shell;

Do not disassemble this product without authorization. There are no user repairable components inside;

Plug and unplug other cables with moderate force;

If not used for a long time, please power off and store it properly;

Turn off the power during the machine repair or adjustment;

Operators and maintenance personnel must undergo training.

1.2 Machining process

The machining process of the system is as follows:

1. Debug before returning to zero
2. Return to mechanical origin
3. Set the work origin
4. Load the machining program
5. Select the machining mode
6. Implement machining

Refer to the following function introduction for specific operations.

Chapter II Installation and Safety Precautions

2.1 System configuration

3C-IO11A master control, industrial personal computer, power pack, display, and expansion board

Industrial network cable

2.2 Safety reminder and system installation

Please read the Manual carefully before operating the control system.

Carefully read the Operating Manual and the User Safety Instructions. Users should take corresponding protection and safety measures before the operation. Before operating the system for the first time, operators should understand the correct usage of the corresponding functions. It is strictly prohibited to operate or change the unfamiliar system functions or parameters at will.

The safety labels and safety related contents in the Manual are very important and must be followed. Failure to follow the requirements can result in hazardous situations, even result in minor or moderate injuries, as well as equipment damage.

Note: The controller cannot be used for heavy machinery and equipment that can easily cause personal safety accidents. 2.2.1 Precautions for system

installation

1) Wiring operations must be carried out by professional electricians.
2) Confirm that the power supply is disconnected before the operation.
3) Install it on metal or other flame retardants and keep it away from combustible materials.
4) Ensure the safe grounding during use.
5) If there is an abnormality in the external power supply, the control system may malfunction. To ensure the safe operation of the entire system, please make sure to set up a safety circuit outside the control system.
6) Be familiar with the contents of the Manual before installation, wiring, operation and maintenance; know relevant mechanical and electronic knowledge and all safety precautions during use.
7) The electrical cabinet for installing the controller should have good ventilation, oil proof, and dust proof conditions. If the electrical cabinet is sealed, it is easy to cause the temperature of the controller to be too high, which affects normal operation. Therefore, it is necessary to install a fan. The suitable temperature inside the electrical cabinet is below 40℃, and it should not be used in areas with condensation or freezing.
8) The installation position of the controller should try to avoid being too close to the arrangement of AC accessories such as contactors and transformers, etc., to avoid unnecessary surge interference.

2.2.2 Installation environment of carving machine

1) Solid ground;
2) Avoid direct sunlight;
3) Leave a certain space for maintenance and repair;
4) Space temperature: 5-40℃;
5) Relative humidity: 30-95%;
6) Horizontal installation of equipment;
7) Good ventilation.

2.2.3 Precautions storage and handling

***Note: Do not store or place it in the following environment; otherwise, it may cause fire, electric shock, or machine damage.**

1) Places with direct sunlight, places where ambient temperature exceeds storage temperature, places where relative humidity exceeds storage humidity, and places with large temperature differences and condensation.
2) Places close to corrosive and combustible gases, places with high levels of dust, salt, and metal dust, places with dripping water, oil, and drugs, and places where vibration or impact can be transmitted to the main body.
3) Do not hold the cable for handling; otherwise, it may cause machine damage or fault.

2.2.4 General notes

1) Do not stack this product too much together; otherwise, it may cause damage or fault during use.
2) This product is a general industrial product and is not intended to harm any life or health.
3) If applied to devices that may cause major accidents or damage due to the fault of this product, please install the safety devices.
4) If applied in environments with high concentrations of sulfur or sulfurized gases, please note that the chip resistor may be disconnected or have poor contact due to sulfurization reactions.

5) If the input voltage exceeds the rated range of the power supply of this product, smoke and fire may occur due to damage to internal components. Please pay close attention to the input voltage.
6) Please note that this product cannot guarantee use beyond the product specification range.
7) If there is any inconsistency or incompleteness with the system function in the Manual, the system software function shall prevail.
8) The control system functions are subject to change or improvement (upgrade) without prior notice. In case of any other needs, please contact our Company.

2.2.5 Safety precautions for use

 Note	Do not use this product in the environments with strong interference and magnetic field. Pay attention to waterproofing, dust prevention, and fire prevention. Plug and unplug USB disk and other cables with moderate force. If not used for a long time, please power off and store it properly. Prevent conductive substances (such as metals, etc.) from entering the shell. Encoders must use shielded wires, and the shielding layer must ensure reliable grounding of a single end! Do not install transformers or other devices that generate electromagnetic waves or interference around the servo drive; otherwise, it may cause the fault of servo drive. If such devices need to be installed, a shielding plate should be installed between the deveices and the servo drive.
	Please protect the branches, short circuits, and loops in accordance with local standards. If the protection measures for branches, short circuits, and loops are not appropriate, it may cause damage to the servo driver. Do not share the grounding wire with welding machines or power machines that require high current; otherwise, it may cause poor operation of the servo driver or machine. Do not connect or operate the machine if there is obvious damage or missing components. Wiring, inspection, etc. should be carried out by professional personnel. Do not disassemble this product without authorization. There are no user repairable components inside;
	The rotating motor will feed electrical energy to the servo driver, which will cause the servo driver to become energized even when the motor stops and the power is cut off. Before maintaining the servo drive, please ensure that the motor servo drive is safely disconnected.
 Danger	Do not perform wiring operations while the power is on; otherwise, there will be a risk of electric shock. Please cut off the power of all the equipment before inspection. Even if the power supply is cut off, there is residual voltage in the capacitor. After cutting off the power, please wait at least 10 min. The carving knife is very sharp and should not be touched by hand during operation to prevent injury. Do not contact it with handkerchiefs or scarves to prevent injury or damage to the equipment caused by entanglement;

	After power on
	Do not open the control box cover after power on; otherwise, there will be a risk of electric shock! Do not plug or unplug the cable of the operation box with electricity. Do not remove the cover of the servo drive or touch the printed circuit board while it is powered on; otherwise, there will be a risk of electric shock.
	During operation
	Non professionals cannot detect signals during operation; otherwise, it may cause personal injury or equipment damage! Do not touch the cooling fan and discharge resistor to test the temperature; otherwise, it may cause burns!
	During maintenance
	Operators and maintenance personnel must undergo training Turn off the power during the machine repair or adjustment. Personnel without professional training cannot repair or maintain the servo drives; otherwise, it may cause personal injury or equipment damage! Do not repair or maintain the equipment with electricity; otherwise, there will be a risk of electric shock! All pluggable plug-ins must be plugged and unplugged under the power outage! Parameters shall be set and checked after replacing the servo driver. Do not power on or run damaged machines; otherwise, it may further damage the machine.
 Warning	Some systems may experience mechanical self action when powered on, please be careful; otherwise; it may cause death or serious injury. Non electrical construction professionals cannot install, maintain, check, or replace components; otherwise, there will be a risk of electric shock. Do not disassemble this product without authorization. There are no user repairable components inside.

***Note: Improper handling may cause hazards, including personal injury or equipment accidents.**

2.2.6 Prohibitions

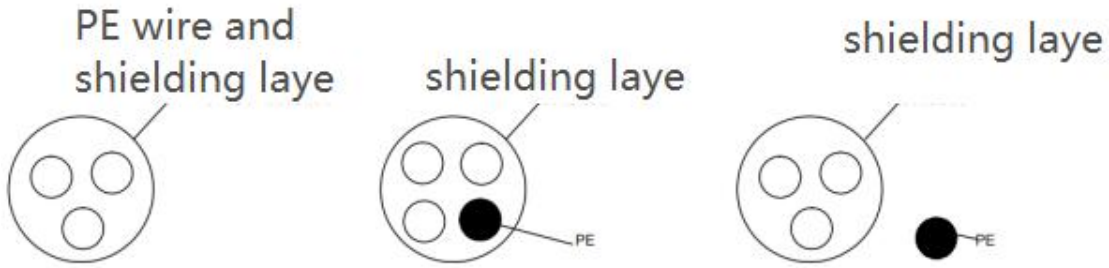
Do not disassemble or repair this product except for our staff.

2.2.7 Precautions for disposal

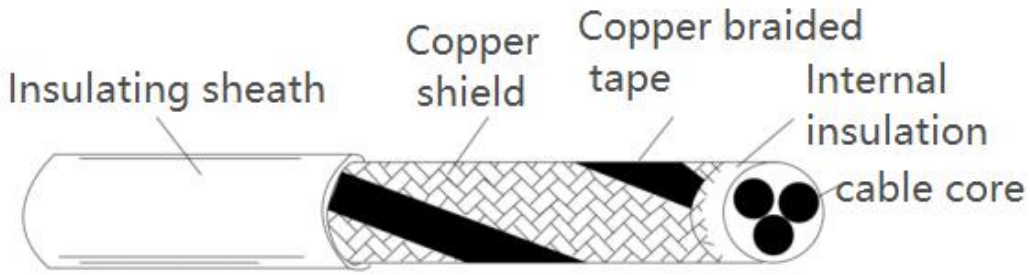
When the product needs to be disposed of as waste after normal use, please comply with the legal regulations of relevant departments regarding the recycling and reuse of electronic information products.

2.3 Cable requirements

In order to meet the requirements of EMC, encoder cables must use shielded cables with shielding layers. It is recommended to use shielded cables with shielding layers for power lines. The shielded cables include three phase-conductor shielded cables and four phase-conductor shielded cables, one of which is a PE line, as shown in the following figure:



In order to effectively suppress the emission and conduction of radio frequency interference, the shielding layer of the shielded wires is composed of coaxial copper woven belt. In order to increase shielding effectiveness and conductivity, the weaving density of the shielding layer should be greater than 90%. As shown in the figure below:



Installation precautions:
(1) It is recommended to use shielded balanced cables for all shielded cables, and to use four-core cables for input cables;
(2) The motor cable and its PE shielded wire (twisted shield) should be as short as possible to reduce electromagnetic radiation, stray current and capacitive current outside the cables;
(3) It is recommended to use shielded cables for all control cables;
(4) It is recommended to use shielded cables or steel pipe shielded power lines for the output power lines of the driver, and to use twisted pair shielded control wires for the leads of the affected equipment, and the shielding layer should be reliably grounded.

2.4 Common EMC problem-solving suggestions

The driver belongs to equipment with strong interference, which may still cause interference due to the wiring, grounding, etc. during use. When there is mutual interference with other devices, the following methods can also be used for rectification.

Interference type	
Earth leakage circuit breaker switch tripped	Reduce carrier frequency; Reduce the length of the drive line; Add the magnetic ring on the input drive line (without PE wire); Once tripped immediately upon power on, disconnect the large ground capacitance at the input end; (Disconnect the grounding terminal of the external or internal filter, and the grounding terminal of the input port to the ground Y capacitor) Once tripped during operation or enabling, take leakage

	current suppression measures at the input end (leakage current filter, safety capacitor + magnetic ring, magnetic ring)
Communication interference	Connect the motor shell to the PE end of the driver; Connect the PE end of the driver to the power grid PE; Add a magnetic ring on the input power line; Add build-out resistors to the communication line source and load ends; Add a common communication grounding wire to the differential line of the external communication line; Shielded wire is used for communication lines, and the shielding layer is connected to the common communication grounding wire; The communication wiring needs to be twisted pair;

Table 2-1 Common EMC Interference Problems and Solutions

Chapter III System Wiring Instructions

Figure 3-1 Bus Wiring Diagram

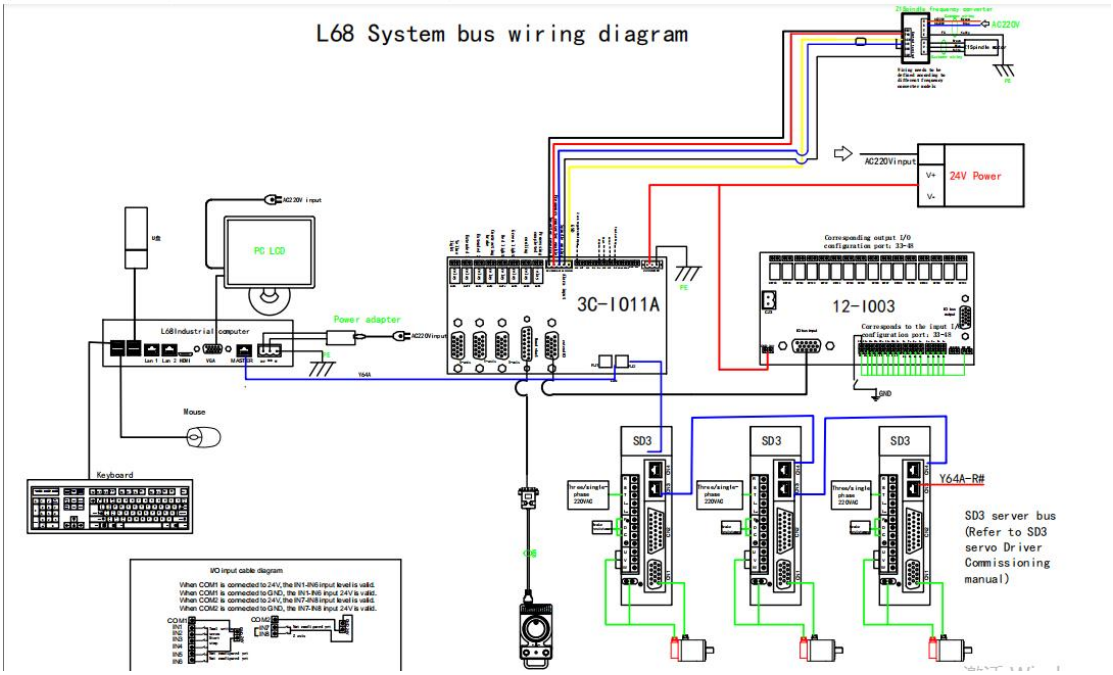
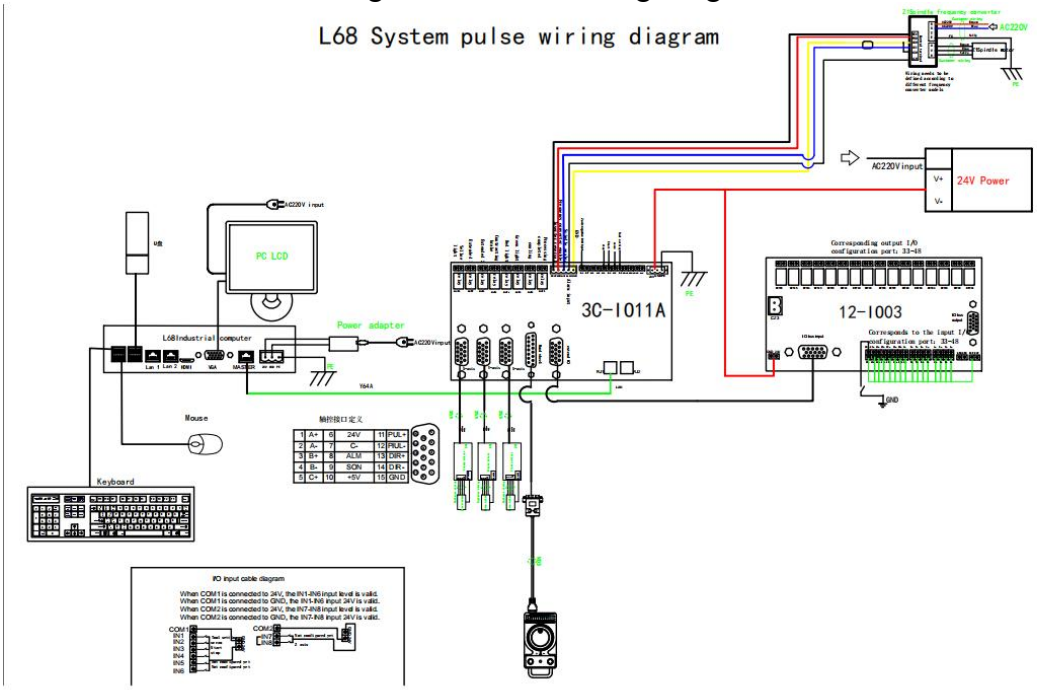


Figure 3-2 Pulse Wiring Diagram



3.1 Definition of terminal

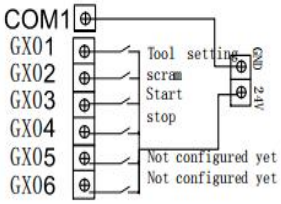
The master control input IO 1-8, output IO 1-8, and expansion board input and output each have 16 channels, and the corresponding input IO 33-48 and output IO 33-48 can freely set port functions, which can be used once configured.

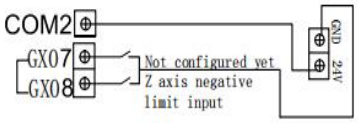
Note: There are two ways to connect input IO, as shown in the figure

Figure 3-3 Input IO Wiring Diagram

I/O input cable diagram

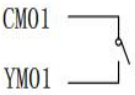
When COM1 is connected to 24V, the IN1-IN6 input level is valid.
When COM1 is connected to GND, the IN1-IN6 input 24V is valid.
When COM2 is connected to 24V, the IN7-IN8 input level is valid.
When COM2 is connected to GND, the IN7-IN8 input 24V is valid.





I/O Output wiring diagram

CM01 and YM01 are a set of switch signals,with on output signal in the normally open state and output in the normally closed state.
CM01-CM08 the principle is the same.



Classification	Port	Definitions	Description
Power input	24V	24V power input end	DC 24V input, supplying power for system operation.
	GND	24V GND	
	PE	Grounding wire end	
IO input	24V	24V power input end	24V power input, supplying power to the optoelectronic switch.
	GND	Common terminal	Power ground and switch common terminal.
	COM1	Common terminal	See Figure 3-3
	COM2	Common terminal	See Figure 3-3
	GX01	Tool setting	Switch input, able to be connected to normally open or normally closed.
	GX02	Emergency stop	Switch input, able to be connected to normally open or normally closed.
	GX03	Start machining	Switch input, able to be connected to normally open or normally closed.
	GX04	Pause machining	Switch input, able to be connected to normally open or normally closed.
	GX05	Not configured	Switch input, able to be connected to normally open or normally closed.
	GX06	Not configured	Switch input, able to be connected to normally open or normally closed.
	GX07	Not configured	Switch input, able to be connected to normally open or normally closed.
	GX08	Z-axis negative limit	Switch input, able to be connected to normally open or normally closed.
Axis control	X axis	X-axis control interface	Connect to external drives, see Figure 3-4 for details
	Y axis	Y-axis control interface	Connect to external drives, see Figure 3-4 for details
	Z axis	Z-axis control interface	Connect to external drives, see Figure 3-4 for details
Handwheel input	Handwheel	Electronic handwheel interface	Handwheel pin, see Figure 3-5
IO output	OUT1	Automatic machining completed	Transistor output
	OUT2	Cooling	Transistor output

	OUT3	Green light	Transistor output
	OUT4	Red light	Transistor output
	OUT5	Band-type brake	Transistor output
	OUT6	Expansion output 2	Transistor output
	OUT7	Expansion output 1	Transistor output
	OUT8	Yellow light	Transistor output
Spindle	GND	Common terminal	Power ground and switch common terminal.
	FWD	Spindle enable output end	Spindle on, usually connected to the positive rotation signal of the frequency converter.
	REV	Spindle enable reverse	Spindle on, usually connected to the negative rotation signal of the frequency converter
	ALM	Spindle alarm	Spindle alarm
	AVI	Spindle speed control	Spindle speed control output
Internet access	RJ1	Internet access	SLBUS protocol interface
	RJ2	Internet access	SLBUS protocol interface

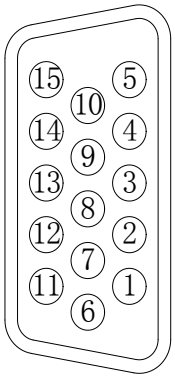


Figure 3-4 System Axis Control Interface

Port (IN)	Definitions	Description	Port (IN)	Definitions	Description
01	A+	A-phase feedback +	09	SON	Servo ON
02	A-	A-phase feedback -	10	ALM_RST	Alarm cleared
03	B+	B-phase feedback +	11	PUL+	Pulse output +
04	B-	B-phase feedback -	12	PUL-	Pulse output -
05	C+	C-phase feedback +	13	DIR+	Direction output +
06	24V	24V output	14	DIR-	Direction output -
07	C-	C-phase feedback -	15	GND	Common terminal
08	ALM	Alarm input			

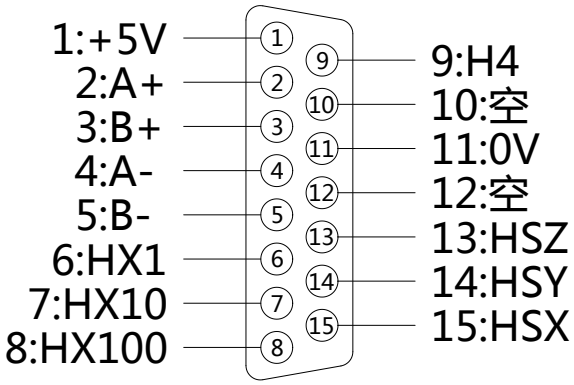


Figure 3-5 Handwheel Interface Definition

Port (IN)	Definitions	Description	Port (IN)	Definitions	Description
1	+5V	Supply power for the handwheel	9	H4	Select 4 axes
2	A+	Encoder A signal	10	Null	Null
3	B+	Encoder B signal	11	0V	Digital ground
4	A-	Encoder A signal ground	12	Null	Null
5	B-	Encoder B signal ground	13	HSZ	Select Z axis
6	HX1	Select X1 magnification	14	HSY	Select Y axis
7	HX10	Select X10 magnification	15	HSX	Select X axis
8	HX100	Select X100 magnification			

Table 3-1 System Terminal Definition Description

3.2 Wiring example

L68 system adopts DC power supply, with a rated power of 24V/2.2A, as shown in Figure 3-2. Please provide a switch power supply with sufficient power according to the actual use of external relays, solenoid valves, and other external accessories. It is recommended to use a waterproof power supply.

Chapter IV System Operation Interface

4.1 System interface

The system interface consists of a title bar, menu bar, status bar, toolbar, machining path window, and function window. As shown in Figure 4-1 below:

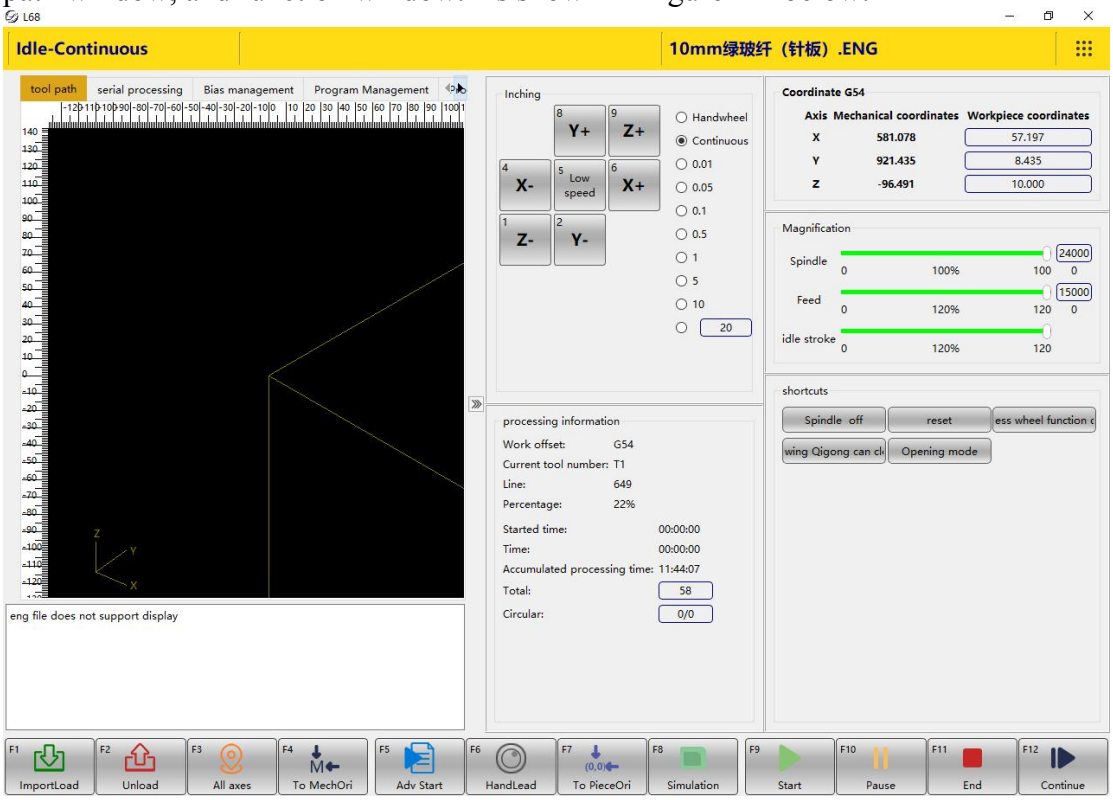


Figure 4-1 System Interface

Menu bar: It contains multiple drop-down submenus, representing four main operations: "Automatic", "Manual", "Parameter Management", and "Auxiliary". Users can select the corresponding menu item to perform an action or implement a function through the [Menu Bar].

Status bar: It displays the running status, alarm or prompt messages, loaded file names, or process names.

Monitoring bar: It includes four monitoring windows: "Coordinates", "Machining", "Feed", and "Spindle", mainly used to display some status information during the machining. By clicking the display button, the corresponding coordinates or speed parameters can be changed.

Toolbar: The shortcut button performs the corresponding operation through the button.

Machining path window: It displays 3D path images for simulation or machining, which can be zoomed in, out, moved, and viewed for machining details.

Multi-function window: It includes: "Automatic", "Manual", "Tool Setting", "Program Management", "Program Editing", "IO Status", "System Log", "Pulse Feedback", "Test Data", and each window represents a classification function.

4.2 Toolbar

The toolbar is located below the menu bar, consisting of some operation buttons that correspond to certain menu commands or option functions. Users can directly click these buttons through mouse to achieve the corresponding specified functions.

The information prompt box is on the right side of the toolbar, which displays alarm or prompt information, facilitating human-computer interaction.

Figure 4-2 Toolbar

Toolbar button function:

: Import files and load them onto the system

: Unload the loaded files

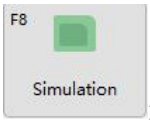
: Return to the workpiece origin

: Set the current point (all axes) as the workpiece origin

: Return to mechanical origin

: Advanced start, select line number for machining

: Handwheel guided

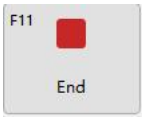


: Simulation

: Start

: Pause

: Breakpoint continuing



: Stop

4.3 Status bar

Status bar: It includes four information display windows, as shown in the following figure:

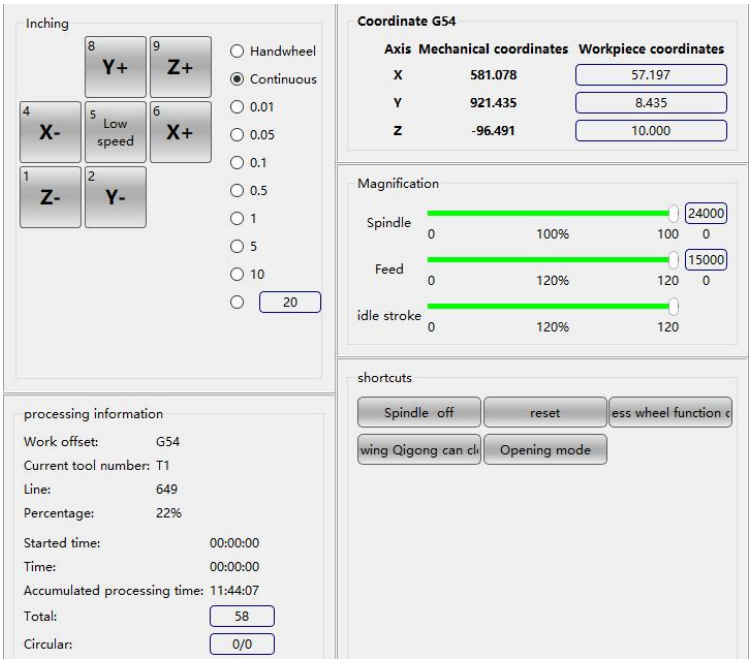


Figure 4-3 Status Bar

"Coordinates" are coordinate information

It displays the current mechanical coordinates and workpiece coordinates. To set the current point position as the workpiece origin at any time, simply click on the workpiece coordinate with the mouse to pop up a dialog box, and then click "OK" to set the corresponding axis coordinate position as the workpiece origin.

"Feed" is the feed rate area

Users can set the feed rate, adjust the feed rate, display the rate, and the actual value of the feed rate. In addition, it also displays manual high and low speeds, and G00 fixed speeds.

During the automatic machining, users can adjust the machining speed by pulling the slider with the mouse to adjust the feed rate. Alternatively, users can also select to adjust the feed rate through the submenu of [Feed Rate].

Users can adjust the current motion speed multiplier within the range of 0-120% by pulling the slider with the mouse. The feed rate is displayed as a percentage, and the actual maximum speed = set speed value * speed multiplier.

When the system is idle, click on the speed box of the set value to pop up the "Modify Speed" dialog box. Enter the value and click "OK" to complete the modification of the manual high speed and manual low speed values. The setting of the machining speed cannot exceed the maximum single axis speed set in the parameters; otherwise, the system will report an error.

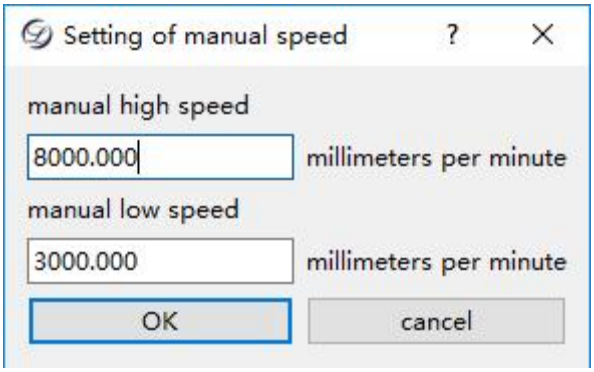


Figure 4-4 Adjust the Feed Rate

"Spindle" is the spindle speed area

Users can set the spindle speed, adjust the spindle rate, display the rate, and the actual value of the spindle speed. It can also start/stop the rotation of the spindle. Users can adjust the spindle speed by pulling the slider with the mouse and modify the spindle speed parameters.

When the system is idle, click on the speed box of the set value to pop up the "Modify Speed" dialog box. Enter the value and click "OK" to complete the modification of the machining speed. The setting of the machining speed cannot exceed the maximum spindle speed set in the parameters; otherwise, the system will report an error.

"Machining" is the machining information area

It displays the current workpiece coordinate system, current line number, start machining time, machined time, completion percentage, and the number of the currently used tool, T1, T2... respectively.

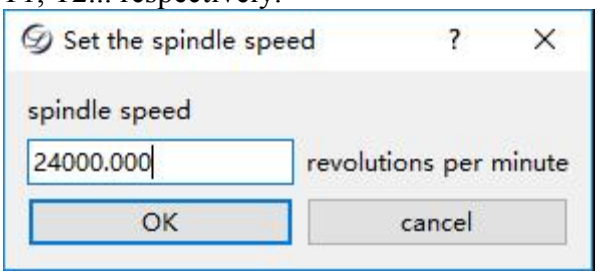


Figure 4-5 Adjust the Spindle Speed

4.4 Machining path window

When the machine tool executes machining programs or simulations, the machining path window can track the tool machining path in real-time. Passed

The 3D real-time display function for tracking machining path allows users to visually detect the path of the tool to confirm correct machining.

In 3D tracking mode, click on the left view function icon of the machining path window to switch between different views, including front view, bottom view, top view, back view, left view, right view, southwest isometric view, southeast isometric view, northeast isometric view, and northwest isometric view, making it convenient for users to view graphics from different angles with appropriate zoom scales. As shown in Figure 4-6.

Users can zoom in/out through the mouse wheel, hold down the left mouse button, and drag the currently displayed machining path. When performing a second or manual machining, if users need to clear the previous machining path, users can right-click and click [Clear] to avoid confusion.

In addition to the machining path window, there are also windows for program management, program editing, IO status, etc. Users can switch through the buttons "Program Management", "Program Editing", "System Log", "Pulse Feedback", and "Test Data" on the right side of the screen.

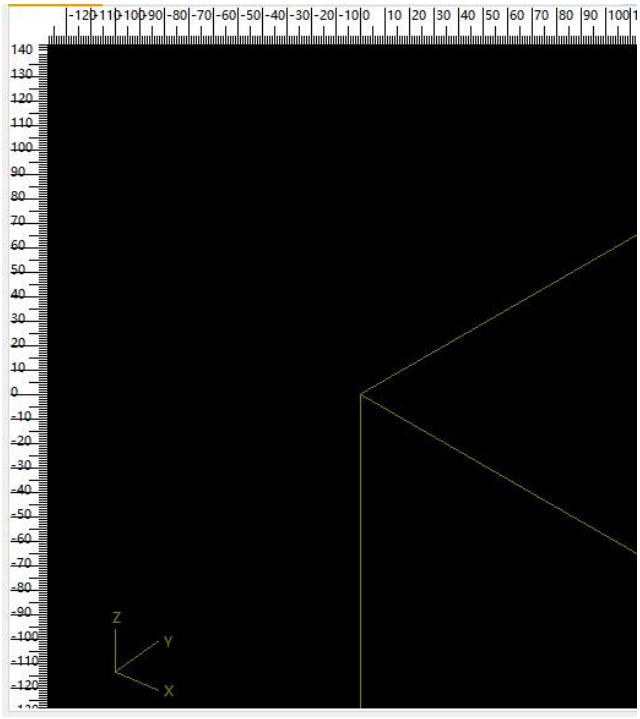


Figure 4-6 Machining Path Window

Chapter V Machining File Import

There are three methods for inputting machining files: 1. Import and load, 2. Manually write in the system, and 3. Copy and paste to a shared folder. The first method is generally used for importing commonly used machining files into the system; the second method is suitable for relatively simple machining files; the third method is suitable for batch importing into the system.

5.1 Import in program management

Click the [Program Management] interface, then click the "Import and Load" function key at the lower part of the program management interface or the " 

" at the lower part of the main interface to enter the file selection window, select the machining file to import and click "OK". In the program management interface, users can see that the machining file has been imported and loaded into the system.

5.2 Manually write in the system

Click on the [Program Management] interface to create a new nc file, then select New nc File, and click the "Edit" button at the bottom of the window to edit the file. After the user completes editing, right-click and select "Save".

Figure 5-2 Edit Machining Files

5.3 Copy and paste in bulk to a shared folder

Open the shared folder, select to copy the files that need to be imported into the system, paste them into the path of the shared folder "processfiles" folder, and then click "Refresh" in "Program Management" to see the copied and pasted machining files.

Chapter VI Return to Mechanical Origin

6.1 Debug before returning to zero

Before returning to the mechanical origin, it is necessary to carry out debugging before returning to zero, which mainly includes adjusting pulse equivalent, checking the direction of the machine tool spindle, and setting the table travel.

6.1.1 Adjust the pulse equivalent

Impulse equivalent: The smaller the pulse equivalent value is, the higher the machining accuracy of the machine tool and the surface quality of the workpiece will be; The higher the value is, the higher the machine tool feed rate will be.

Select the menu item [Parameter Setting] in the [Parameter Management] menu, and click on the manufacturer parameter to set the pulse equivalent for the feed shaft parameter.

6.1.2 Checking the direction of the machine tool spindle

Set the pulse equivalent to ensure that the position of the spindle movement is consistent with the actual situation.

Select the menu item [Parameter Setting] in the [Parameter Management] menu, and click on the operating parameters of the user parameters to view the spindle direction setting value. Then click on the [Manual] window, select continuous or step length mode, move the spindle, and check whether the actual direction of spindle movement is consistent with the parameter settings. If the direction is consistent, it indicates that the direction of the machine tool spindle is set correctly; if the direction is inconsistent, it is necessary to set the spindle direction parameter value to be opposite to the original direction.

6.1.3 Set the table travel

Set the table travel according to the actual size of the machine tool

Select the menu item [Parameter Setting] in the [Parameter Management] menu, and click on the feed shaft parameters of the manufacturer parameters to set the lower and upper limits of the table travel based on the actual size of the machine tool.

6.2 Return to mechanical origin

The mechanical origin is a fixed position of the machine tool, which is jointly determined by the mechanical switch and electrical system, and is the zero point of the mechanical coordinate system. To execute the "Return to Mechanical Origin" function, the machine tool itself needs to be equipped with an origin switch. If the machine tool does not have relevant hardware support, this function needs to be disabled. Refer to the "Origin Parameter" setting in Chapter XI "Parameter Management" for details. Since the mechanical origin is the benchmark of the whole machine tool, the important role of this function is to correct the current point coordinates. To prevent power outage or incorrect current position, please perform the operation of returning to mechanical origin after program startup or emergency stop.

After the system is powered on and started, a "Return to Mechanical Origin" dialog box will automatically pop up. After clicking the button, the corresponding axis will automatically return to the mechanical origin and correct the system coordinates. Before returning the X-axis or Y-axis to the origin, please return the Z-axis to the mechanical origin first.

After selecting the menu item "Return to Mechanical Origin" in the "Manual" menu, the system will pop up the "Return to Origin" dialog box, as shown in the following figure:

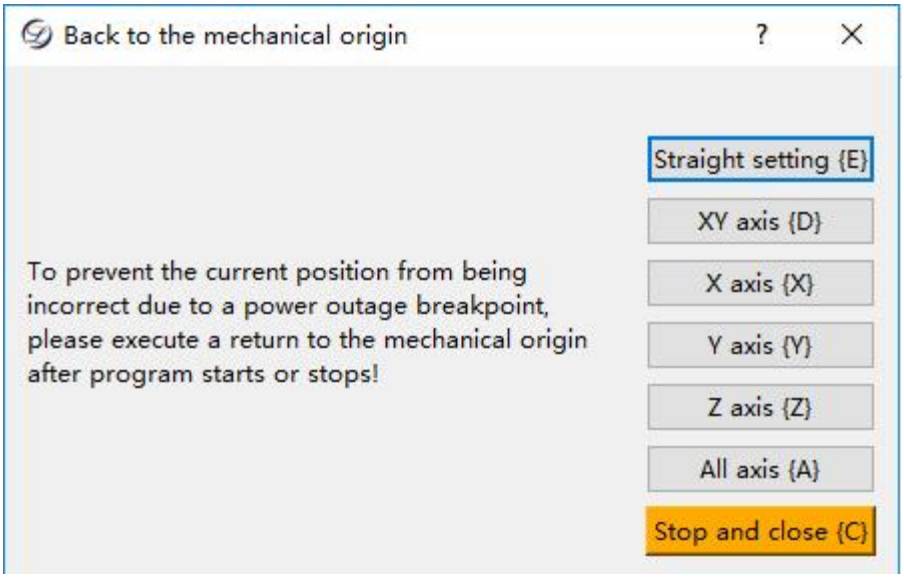


Figure 6-1 Mechanical Origin Function Screen

Single axis return to mechanical origin includes the following methods:

Direct setting: If it is determined that the current position is not consistent with the mechanical coordinates, and the machine tool has not been turned off and has not performed an emergency stop, direct setting can be used.

X-axis return to mechanical origin: Select this command to return the X-axis to the mechanical origin.

Y-axis return to mechanical origin: Select this command to return the Y-axis to the mechanical origin.

Z-axis return to mechanical origin: Select this command to return the Z-axis to the mechanical origin.

All axes return to mechanical origin: Select this command to return all axes to the mechanical origin.

Note:

If the return to the mechanical origin is not performed, please raise the Z-axis as high as possible during manual operation to ensure that the tool head will not collide with the workpiece to be machined.

When exiting, the system will automatically save the current coordinate information. If there is a sudden power outage during the automatic machining, the system will automatically save the relevant information before the power outage to the breakpoint protection file (that is, save the breakpoint information, file name, etc. to the system memory during the power outage. The same machining file only corresponds to one breakpoint protection file). After power restoration, the system will pop up a prompt box to prompt the user that the last machining file suffered from power outage. Users need to manually perform the operation of returning to the mechanical origin first, and then continue machining the file with the power outage, or choose a new machining file:

1. To continue machining the file with power the outage, users can click the "Breakpoint Continuing" button on the toolbar at the top left of the main interface, and the machine tool will return to the position before power outage. After clicking "Start", the machine tool will continue to seamlessly machine the file that was not finished before power outage from the breakpoint.
2. To select a new file for machining, after the machining is completed, users can still continue machining the file with the power outage, and the machine tool will continue seamlessly machining from the breakpoint of the corresponding file.

Chapter VII Set the Workpiece Origin

Before machining the file, users need to manually adjust the position of the tool and workpiece to start machining from the predetermined position of the workpiece.

There are two ways to set the workpiece origin of the Z-axis: 1. Manual setting, 2. Fixed tool setting

7.1 Manual origin removal

X-axis and Y-axis workpiece origin setting: Manually move the X-axis and Y-axis to the predetermined machining position, and click on the workpiece coordinate to pop up a dialog box prompt. Based on the prompt, confirm to reset the X-axis and Y-axis coordinate values of the current position to zero (click on the X-axis and Y-axis coordinate columns in the coordinate window). As shown in the figure below:

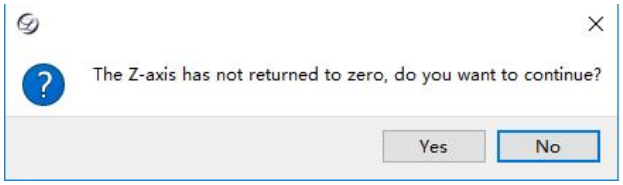


Figure 7-1 Set the Workpiece Origin

7.2 Fixed tool setting

Firstly, it is necessary to set the mechanical coordinates of the tool setting gauge in parameter management.

After performing a fixed tool setting, the system will automatically move to the corresponding mechanical coordinates of X-axis and Y-axis, and then start the Z-axis tool setting.

Fixed tool setting can be divided into the first tool setting and the tool setting after tool changing. Users should pay attention when using it. The first tool setting is the tool setting before machining, through which the workpiece coordinates when the tool nose contacts the fixed feeler block can be determined. Through the tool setting after tool changing, the Z-axis workpiece coordinate can be restored to the value set in the first tool setting action when the tool nose contacts the fixed feeler block. During the fixed tool setting, in order to protect the tool, a speed segmentation method is adopted, which is divided into two types: fast tool setting speed and tool setting speed. The speed can be set through parameters. When the tool nose is about to approach the tool setting gauge, a fast tool setting speed can be used. After the tool nose does not come into contact with the tool setting gauge, a normal tool setting speed can be used.

7.3 Floating tool setting

The tool setting mode can be switched through parameter modification. The system defaults to fixed tool setting mode. If the fixed tool setting is "No" and floating fixed tool setting is "Yes", it will be switched to the floating tool setting mode.

Floating tool setting allows users to easily determine the surface height of the workpiece and set the workpiece origin of the Z-axis. Similar to manual tool setting, since the workpiece origin of the Z-axis is usually at the center of the rotation axis, it is necessary to set the distance between the position of the feeler block and the center of the rotation axis to a common offset (alternatively, add the distance between the feeler block and the center of the rotation axis to the thickness of the feeler block). The specific operation is as follows: Place the feeler block on the workpiece surface, and manually move the tool nose above the workpiece origin. After clicking the "Measure Workpiece Surface" button, the machine tool will perform the tool setting

action. After the tool nose touches the feeler block, it will automatically lift up 10mm, and add the thickness of the feeler block to determine the Z-axis positioning.

Note:

(1) Before tool setting, users must ensure that the tool nose is located above the feeler block, that is, after lowering the tool, the tool nose can touch the feeler block; otherwise, it will continue to lower the tool, causing damage to the tool head and workpiece.

(2) The thickness of the feeler block can be set in the manufacturer parameters, and the workpiece origin coordinate in the Z-axis direction after tool setting will automatically compensate for it.

(3) Due to the existence of a certain gap between the workpiece surface and the feeler block, after the tool setting is completed, the gap can be compensated in the workpiece offset settings. The compensation difference depends on the size of the gap. If the workpiece surface is rough, the compensation difference can be larger. If the workpiece surface is smooth and the gap with the feeler block is smaller, the compensation difference can be set smaller. Generally, 0.1mm can be used.

(4) The tool setting speed can be determined in the parameter settings. The range of tool setting speed is 60-1000mm/min. If the tool setting speed exceeds the maximum value set by the parameter, it will wear the tool head or damage the feeler block.

Note:

7.4 Storage and selection of workpiece origin

Select the menu items [Save Workpiece Origin]/[Read Workpiece Origin] in the [Manual] menu to save the current workpiece origin. Click [Read Workpiece Origin] to set the saved workpiece origin as the current workpiece origin. Up to 10 workpiece origins can be saved.

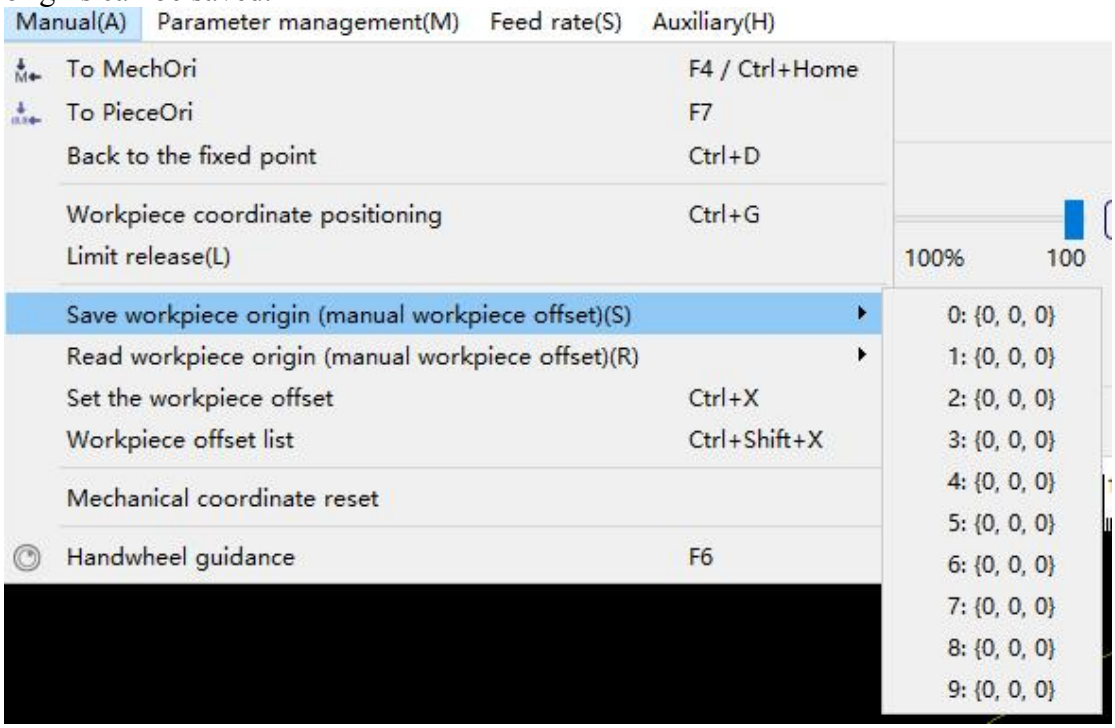


Figure 7-3 Save the Workpiece Origin

Note:

Users can choose the menu item "Save Workpiece Origin" in the "Manual" menu to save the current workpiece origin to the machining file system. In this way, users can save the frequently used workpiece origin as a preset value. When users set the

workpiece origin for the first time and want to quickly locate it in the future, they can use this function to save the coordinate value of the workpiece origin.

Chapter VIII Manual Machining

Manual machining means that the machine tool manually machines the machining program according to the parameters set by the user. There are four ways to manually operate a machine tool: handwheel pulse mode, continuous inching mode, incremental stepping mode, and custom step length mode.

Users can choose the manual operation mode to machine program files. After clicking the "Manual" button on the multi-functional window at the bottom right of the main interface, the window will display a manual operation interface, where you can perform corresponding manual operations. The manual button area in this window contains six manual buttons, corresponding to the positive and negative directions of the X, Y, and Z axes. The manual window provides an interactive operating environment for users to manually manipulate the machine tool.

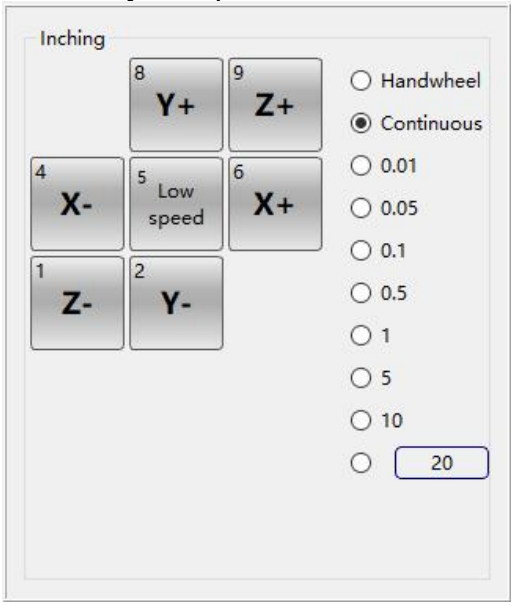


Figure 8-1 Manual Machining Window

8.1 Handwheel pulse mode

Users can choose the handwheel pulse method for continuous machining. When the radio button on the right side of the manual machining window is switched to the "handwheel" option, the movement of the machine tool is determined by the handwheel input.

1. There is a pulse rate option on the handwheel: X1, X10 and X100 gears, which represent different pulse multiples of the handwheel.
2. There is an axis option on the handwheel: Users can select the X, Y, and Z axes to feed.
3. There is a step direction option on the handwheel: Each axis has positive and negative directions. After selecting the stepping axis on the handwheel, the handwheel can be moved in both positive and negative directions, as indicated by the "+/-" arrow on the corresponding direction knob on the handwheel.
4. When the handwheel is in gears X1, X10, and X100, the distance corresponding to each grid of the handwheel can be set through parameters.
5. The acceleration of the handwheel can be set separately.
6. The handwheel supports two modes: 1. Strict pulse counting, 2. Non-strict pulse counting. In the strict pulse counting, the distance traveled by the machine tool is strictly equivalent to the number of manually operated pulses (note: in this mode, if the manually operated pulses are too fast, it may cause too many buffered pulses, and when the handwheel is stopped, the machine tool will also travel a considerable

distance). In the non-strict pulse counting mode, the number of manually operated pulses is not strictly equivalent to the distance traveled by the machine tool. When the handwheel stops, the machine tool immediately begins to slow down and stop.

Note:

Before performing the operation, check that the external handwheel device is properly connected. This mode is mainly used for rapid positioning of machine tools.

8.2 Continuous inching mode

Select the "Continuous" radio button on the right side of the window with the mouse to enter the continuous inching machining mode. In this mode, users can click the manual button with the mouse. When the manual button is pressed on the corresponding axis with the mouse, the machine tool moves; once released, the machine tool stops moving.

In the inching mode, the path display window will display the relevant machining path.

8.3 Stepping mode

Similar to the continuous inching mode, the stepping mode is another manual operation mode of the machine tool. Unlike the continuous inching mode, the stepping mode can accurately control the feed distance of the machine tool's motion axis.

Before using this mode for operation, a suitable step length must be set, and the distance for each inching feed must be set by modifying the inching step length.

The step length can be set in the following ways:

Enter the manual page, click 

with the mouse to pop up a dialog box, and fill in the step length value. As shown in the figure below:

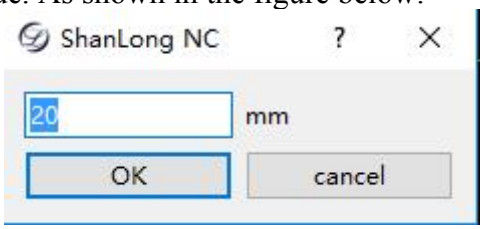


Figure 8-2 Custom Step Length Window

Note:

It is necessary to avoid setting the inching step length value in the Z direction too large to prevent the damage to the machine tool due to misoperation.

Chapter IX Machining Operation

9.1 Load file

Click "Program Management" to switch to the program management window, select the file to be machined, and click "Load" or double-click the mouse. After loading, the file loading success will be displayed on the right side of the toolbar above the main interface, and the loaded file name will be displayed in the title bar below the main interface. At this time, it will automatically switch to the automatic machining window state.

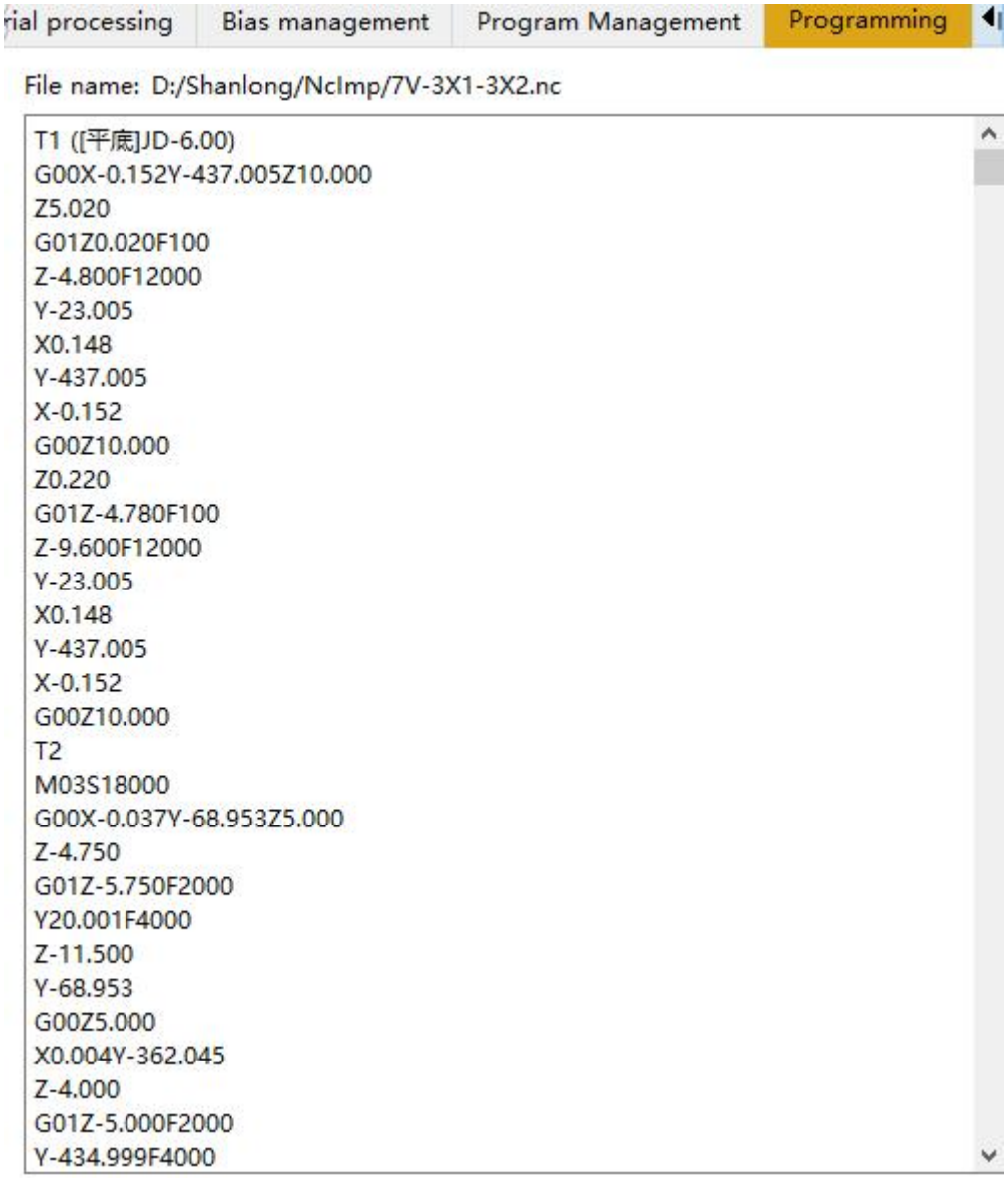


Figure 9-1 "Load" Machining Files

9.2 Set the workpiece origin

Refer to Chapter VII for details. If the workpiece origin has been set, there is no need to reset it.

9.3 Automatic machining

The "Automatic" menu contains items related to automatic machining.

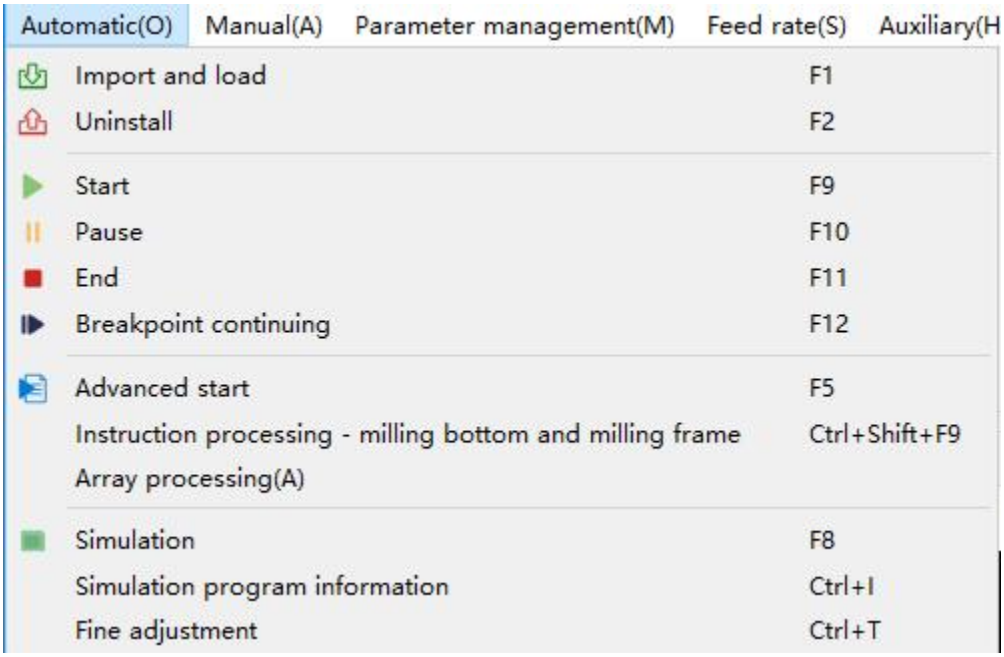


Figure 9-2 "Automatic" Menu

9.3.1 Start

After selecting the machining file, users can click on the "Start" button icon in the tool and status bar at the bottom of the main interface, 

or the shortcut key "F9", and the machine tool will start automatic machining from the first line according to the selected machining file. In the machining path window, it can be seen that the corresponding machining path is displayed based on the tool movement; In the automatic machining window, it can be seen that the program is machined line by line, and the cursor will automatically track to the current code. The highlighted red code is continuously scrolling downwards, and users can view the current machining program code information through this window.

Note:

To set parameters, users must perform the operation of returning to the mechanical origin before machining, and the system will prompt you to return to the mechanical origin first. Automatic machining instructions cannot be executed without returning to the mechanical origin

Figure 9-3 Automatic Machining Interface Status

Figure 9-4 Current Automatic Machining Status Prompt in the Toolbar

Note:

The system will perform syntax check on automatically machined files while machining them, and syntax check is performed earlier than automatic machining (i.e. syntax check has a "forward-looking" function). If the system detects a syntax error in a certain line of the program in the machining file, the error statement will be

highlighted in the automatic machining window and an alarm will be given, and the automatic machining will stop at the same time. Users can perform syntax and semantic checks on erroneous statements, modify and edit them. After editing, click "Save", and then click the "Breakpoint Continuing" button on the upper left toolbar of the main interface again, at this time, the program will continue automatic machining from the modified area.

During the automatic machining, new machining files cannot be loaded. The status bar "Machining Information Window" will display information on the start machining time, machined time, and tool number currently in use of the current file. It is convenient for users to view the operation status of automatic machining.

In the feed rate area, the slider can be adjusted or the feed rate setting value can be changed to reset the feed rate; In the spindle speed area, the slider can be adjusted or the spindle speed setting value can be changed to reset the spindle speed. The value change will take effect immediately.

The start and end information of automatic machining is saved in the system log file. The system log records important actions and events that occur for users, who can not only browse the log information that has occurred since this startup from the system log window, but also review the historical information records that have occurred through this window. If a system fault occurs, users can conduct system analysis and diagnosis through this function.

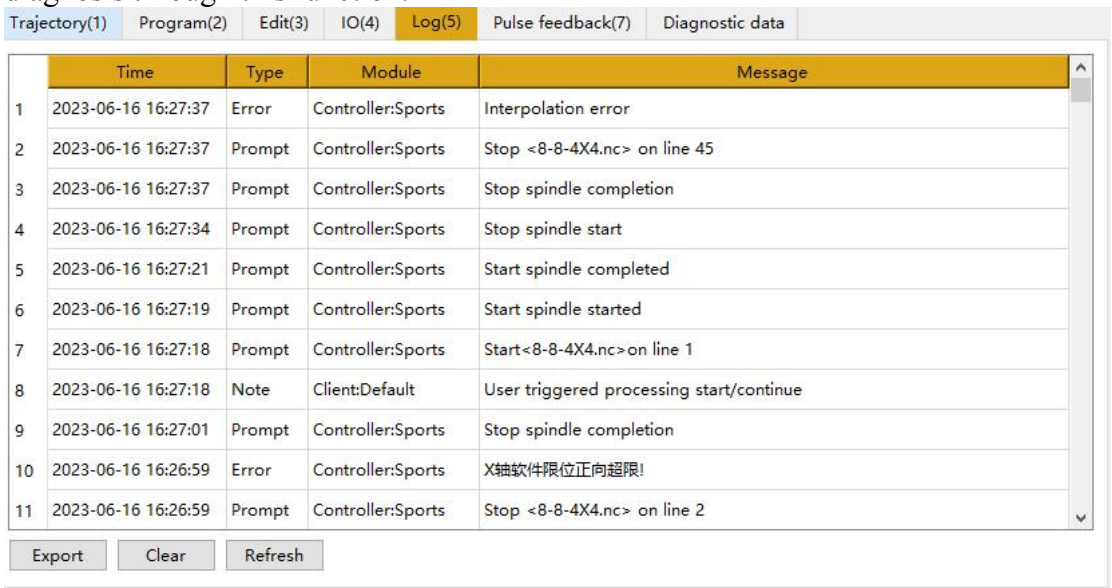


Figure 9-5 System Log Function

The current log information recorded by the system includes:

(1) Start and end information of automatic machining;

(2) Changes in workpiece coordinates;

(3) System alarm information;

(4) Completed machining information of the files;

(5) Other system information.

Note:

The system log can only store 1000 pieces of information.

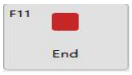
9.3.2 Pause

After "Start" automatic machining, in order to pause machining, users can click the "Pause" button icon on the upper left toolbar of the main interface, 

or the

shortcut key "F10", and the machine will slow down from the current speed until the speed reaches zero.

9.3.3 Stop

After "Starting" automatic machining, in order to stop machining the file, users can click on the "Stop" button icon in the upper left toolbar of the main interface, 

or the shortcut key "F11", and the machine will slow down from the current speed until the speed is zero and the tool is lifted. When the system stops, breakpoints will be automatically saved.

During automatic machining, if the system is in simulation mode, click the "Stop" button to stop the simulation, and exit the simulation mode. At this time, users can analyze the simulation results.

9.3.4 Fine tuning

During the machining, press "Fine Tuning" in the automatic menu to pop up the interface as shown in the figure:

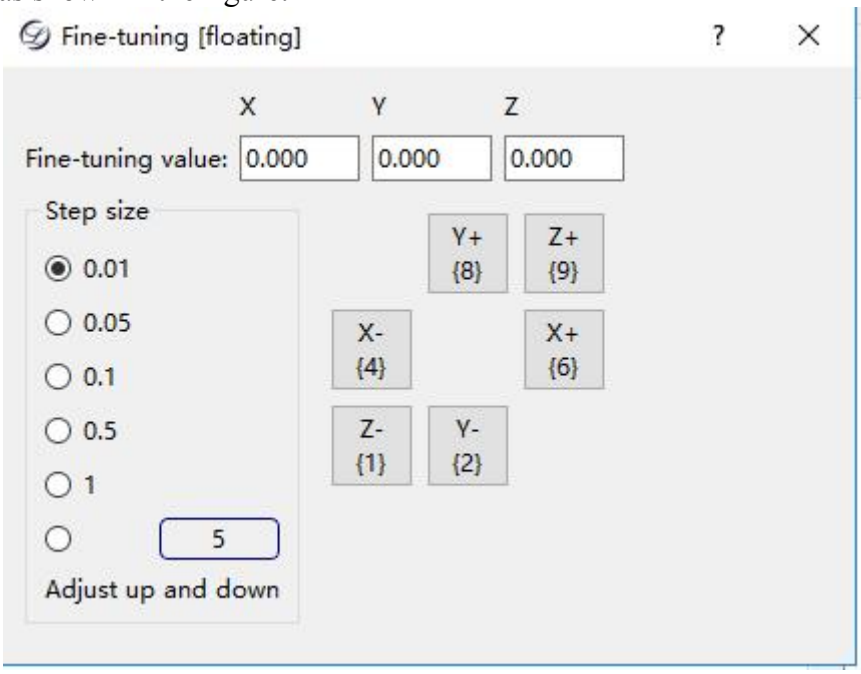
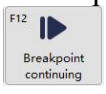


Figure 9-6 Fine Tuning during Machining

The step length can be selected, and the step length value is the fine tuning value. As shown in the interface, the number keys correspond to fine tuning in different directions of the X, Y, Z, and A axes.

9.3.5 Breakpoint continuing

In order to continue machining the workpiece from the last stop, users can click on the "Breakpoint Continuing" button icon in the upper left toolbar of the main interface, 

or the shortcut key "F12". If the breakpoint continuing is caused by a power outage during the machining of the file, it must be returned to the mechanical origin before continuing.

If there is a sudden power outage during machining, users can click the "Breakpoint Continuing" button to restore the scene. The system will continue machining from the point where the last machining interruption occurred. "Breakpoint Continuing" can also be used to continue the simulation.

9.3.6 Advanced start

If the entire file does not need to be machined, as long as it starts from a specified line in the machining file and ends at the specified line number, it is the "Skip Segment Execution" in automatic machining. It can also be used to check whether a certain program in the machining file is correct.

Click 

, or use the shortcut key "F5", or select the "Advanced Start" menu item from the "Automatic" menu. This function implements the functions of "Skip Segment Execution" and "Nearest Point Machining". After selecting this function, the system will pop up the "Execute (Advanced Options)" dialog box, as shown in Figure 9-7:

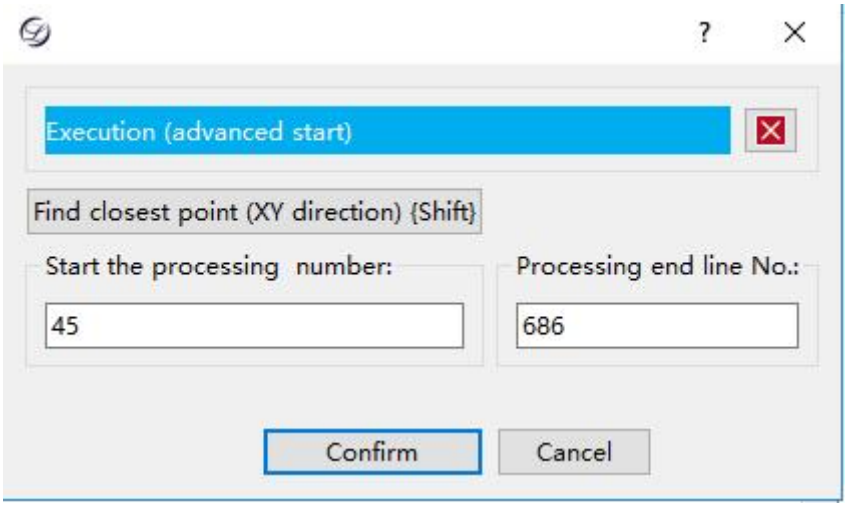


Figure 9-7 Advanced Start

Users can set the starting and ending line numbers of the file in the dialog box, and then click the "OK" button, and the machine tool will only execute the specified program segments in the entire machining program according to the requirements.

Note:

If users choose to start from the file to the end of the file, the entire program file will be machined, which is the maximum range of "Skip Segment Execution".

If the X-axis and Y-axis are moved after stopping machining, to continue machining from the stop point, users can move to the stop point and click [Find the Nearest Point] to continue machining

9.3.7 Array machining

Users can perform array machining on files, and click to select files that require array machining. After entering the parameters of array rows, array columns, array row spacing, and array column spacing, click to generate the machining file, which will generate the array file in the program management directory. The file name after the array is named according to the following rules: Original file name array rows * Number of array columns

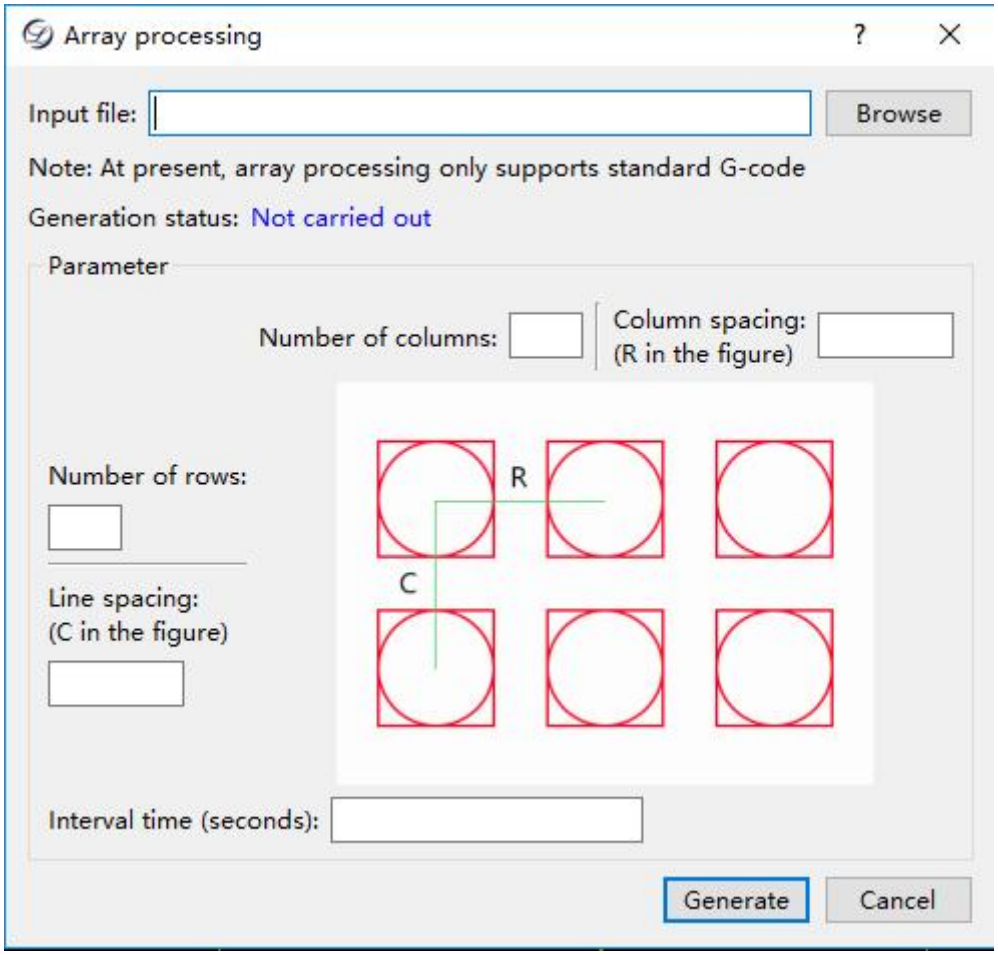


Figure 9-8 Select Array Files

9.4 Handwheel guided machining

The system supports handwheel guided machining and needs to enter handwheel guided mode, as shown in the following figure:

Figure 9-10

Select the "Handwheel Guided" option in the toolbar at the bottom of the interface, and click the "Start" button icon. As it is in handwheel guided mode, the machine tool will not move.

After shaking the handwheel, the machine tool will adjust the speed according to the speed of the handwheel movement along the machining path. When the handwheel is stopped, the machine tool will also stop. When continuing to shake the handwheel, the machine tool will continue machining along the original path. The entire machining process is controlled by a handwheel.

Note:

During the handwheel guided machining, it is only by shaking it in the positive direction of the handwheel that the machining can be guided; otherwise, it is invalid.

Chapter X Check Machining File

When the machining files have been loaded and the current system status is "idle", users can choose the "Simulation Program Information" option in the "Automatic" menu (or click the "Simulation" button icon in the upper left toolbar of the main interface, 

or the shortcut key "F8") to perform high-speed simulation on the loaded machining files.

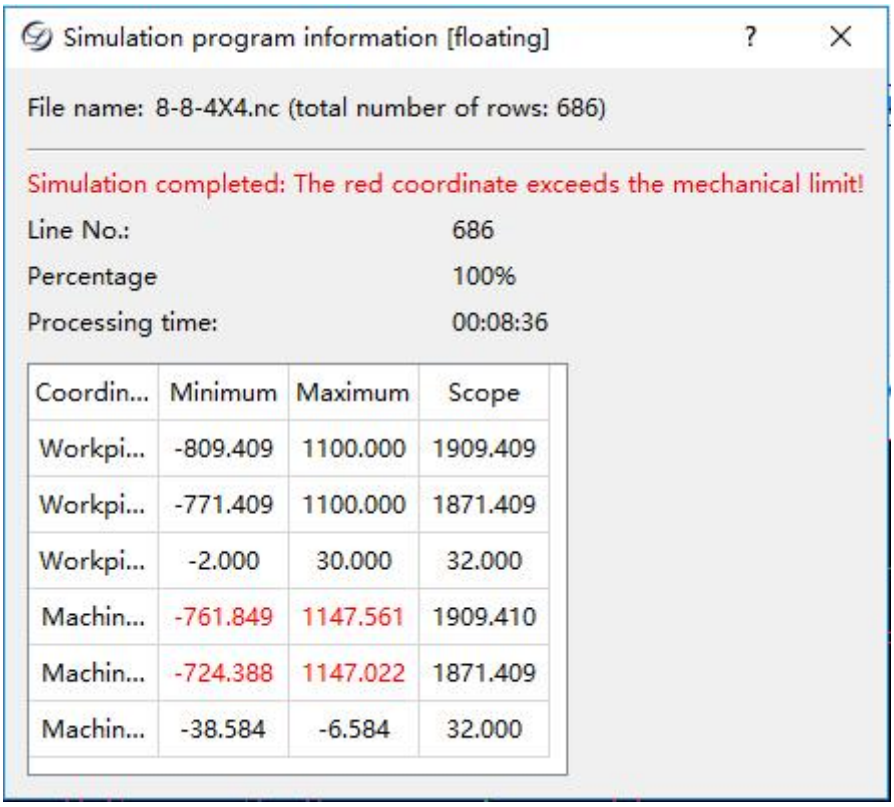


Figure 10-1 Simulation Program Information

Simulation provides users with a fast and realistic simulation machining environment. After the simulation starts, the system no longer emits pulses to drive the machine tool movement, but only tracks and displays the actual effect of tool machining in the window at a high speed. Through simulation, users can understand the movement and machining effect of the machine tool, as well as other additional information in advance to prevent errors in editing the machining program from causing damage to the machine tool. Once the simulation process begins, the menu item will change to "Stop Simulation and Exit Simulation Mode", through which the simulation will immediately terminate.

Note:

The simulation information includes:

(1) When the simulation limit set in the parameters is valid, the system will check whether it will exceed the travel during the simulation. If there is an overtravel prompt during simulation, the actual machining will also overtravel without changing the workpiece origin.

(2) During the simulation, syntax check will be performed on the G code, and syntax errors will be reported.

In the machining path, right-click to enter "Simulation Settings".

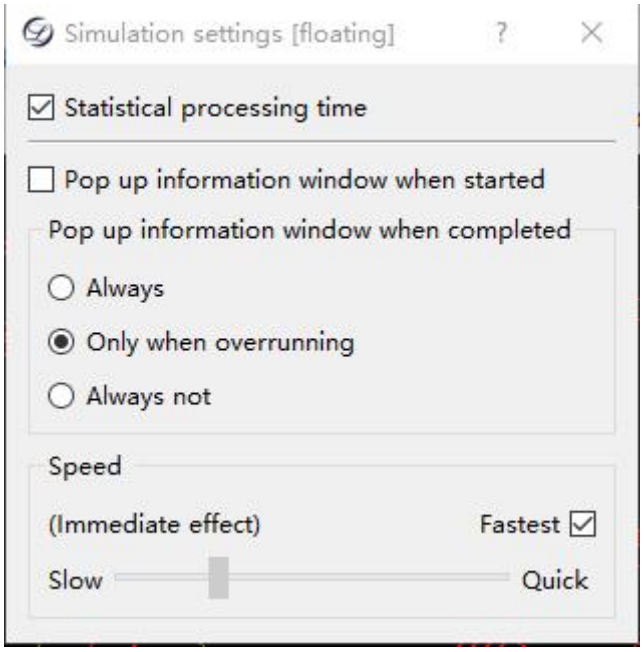


Figure 10-2 Simulation Settings

Enable controller simulation: When the controller simulation is enabled, it will display "total file line number", "machining time", and "machining percentage". If it is not enabled, it will not be displayed.

A message prompt window will pop up after starting: When the controller simulation machining is enabled, the information window that pops up after starting cannot be cancelled, which can only be cancelled after canceling controller simulation machining

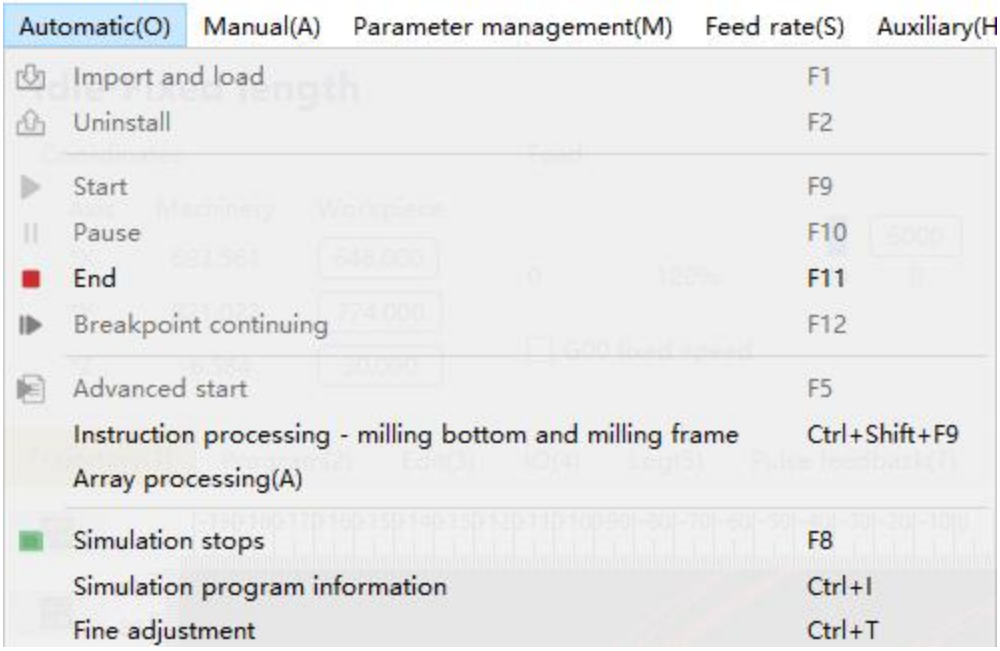
A message window will pop up after completing: When the controller simulation machining is enabled, it will always pop up. When the controller simulation machining is cancelled, the information window can only pop up "only when exceeding the limit"

Plotting speed: The speed setting of the progress bar will only take effect when the "fastest" is cancelled

Chapter XI Milling Bottom and Frame Operations

When users need to perform simple milling bottom and frame operations, there is no need to manually write G code or generate machining files using CAM/CAD software. On the [Automatic] - [Execute Machining Instructions] interface, the system provides the function of executing machining instructions, and it can be completed by inputting a few parameters.

"Execute Machining Instructions" have functions such as rectangular milling bottom, circular milling bottom, milling rectangular frame, and milling circular frame.



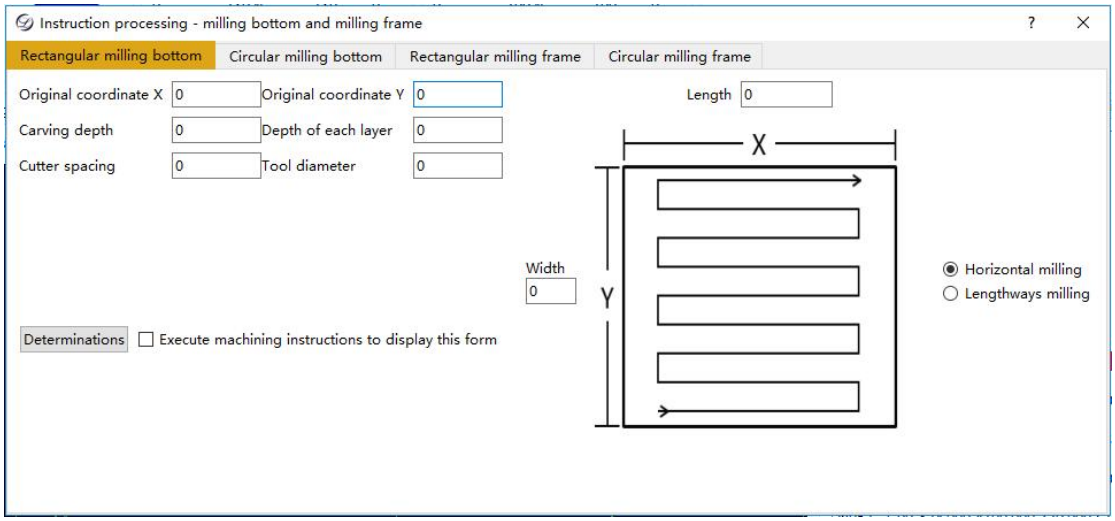


Figure 11-1 Circular Milling Bottom Windo

Chapter XII Program Management

Click[Program Management] to switch to the program management window, where you can create, edit, delete, rename, load, uninstall, import, import and load, browse folders, and refresh machining program files.

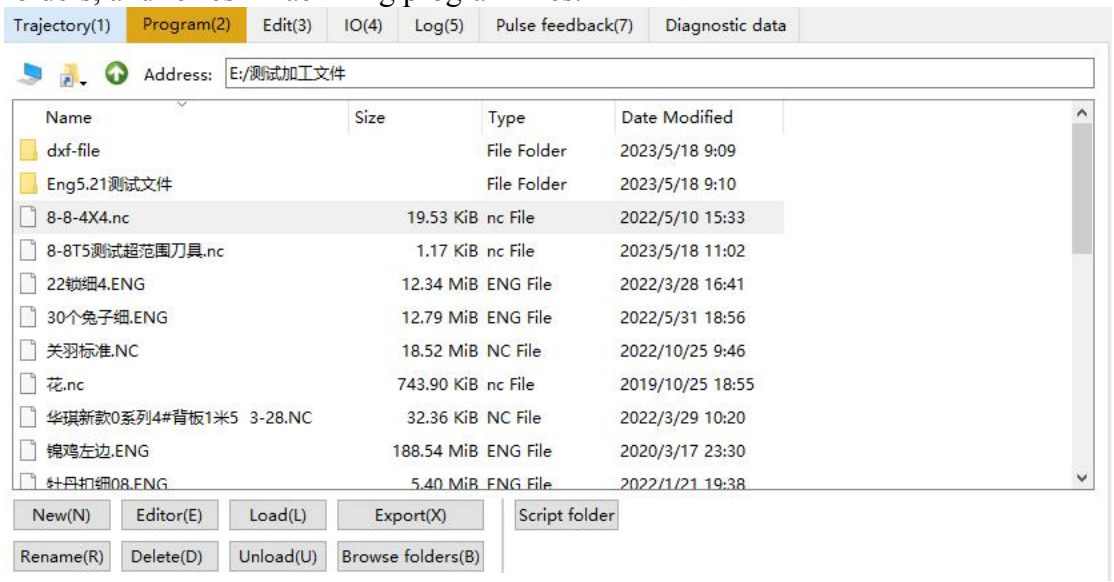


Figure 12-1 "Program Management" Window

12.1 Create

After clicking the "Create" button at the bottom of the [Program Editing] window, a blank new file with the default name (New File 1. nc) will be created in the window.

12.2 Edit

Select an existing file or create a new one, and click the "Edit" button at the bottom of the window to edit and modify the file. Alternatively, click [Program Editing] to switch to the program editing window, and right-click in the editing window to pop up a context menu. Users can easily copy, paste, cut, and other functions to quickly edit and modify the program. After editing, users can right-click and select "Save" or "Close before Saving".

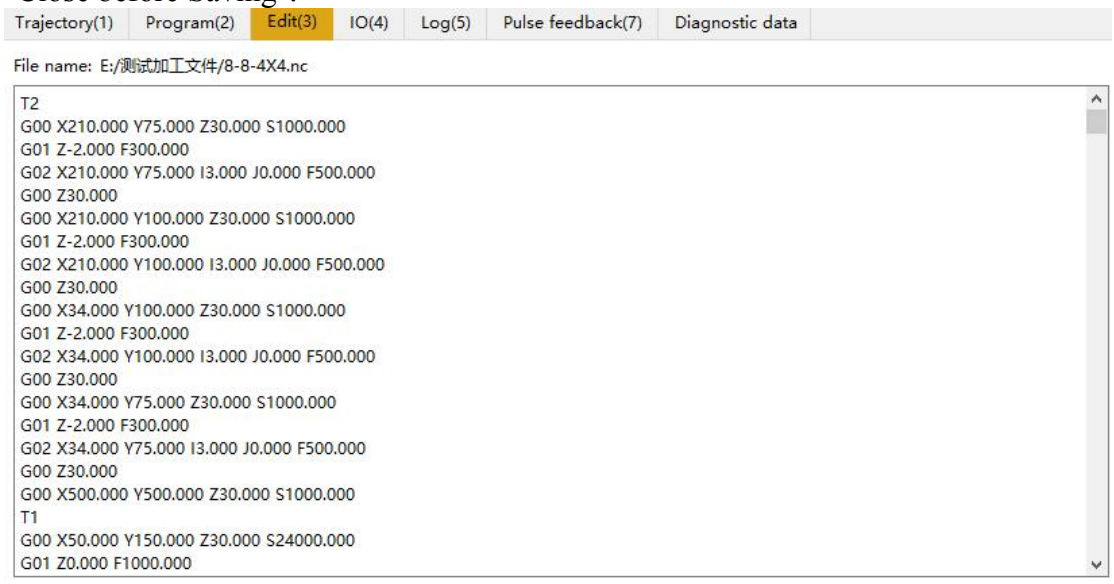


Figure 12-2 "Program Editing" Window

For newly created or edited files, the system will automatically perform a syntax check before saving. Users must edit files according to our programming standards;

otherwise, the system will report an error. Refer to *Programming Specification* for details.

Note:

In this editing window, users can edit machining programs smaller than 10 megabytes in size. In case of files larger than 10 megabytes in size, please use an editor on your PC.

In the editing window, users can input any text. After the input is completed, the system will automatically perform a syntax check to ensure that the machine tool will not execute incorrect instructions and cause damage to the machine tool.

12.3 Delete

Click or move the "Up and Down" keys of the keyboard to select the file to be deleted, and click the "Delete" key to delete the selected file. Certain files can also be selected for batch deletion.

12.4 Load

Select the machining file and click "Load" or double-click the mouse to load it. After the file is successfully loaded, the loaded file name will be displayed in the upper right corner of the interface.

Chapter XIII Parameter Management

The [Parameter Management] menu includes items related to parameters, as shown in the figure:

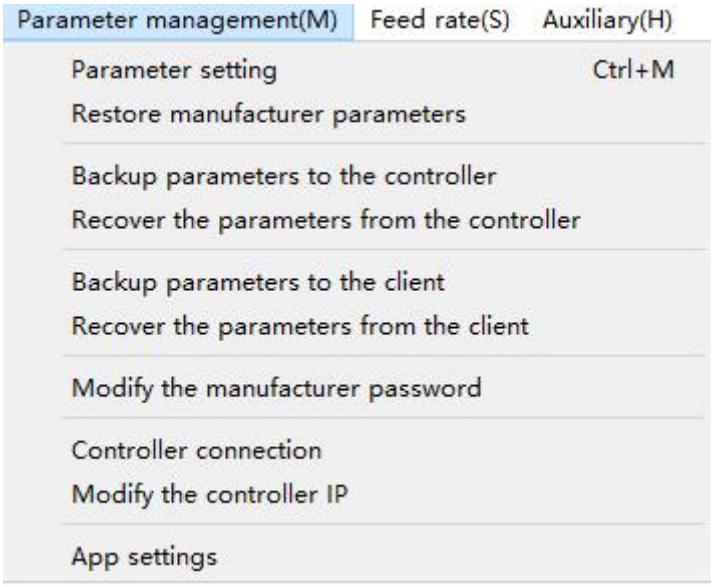


Figure 13-1 Parameter Management Menu

13.1 Setting parameters

Select the menu item [Set Parameters] in the [Parameter Management] menu to pop up the following window for parameter settings under parameter permissions. It is divided into two parts: user parameters and manufacturer parameters. The opening of tool parameters depends on whether [N81661] supports wheel driven tool change settings. Yes: it supports wheel driven tool change and opens tool parameters; No: it does not support wheel driven tool change, and tool parameters are shielded.

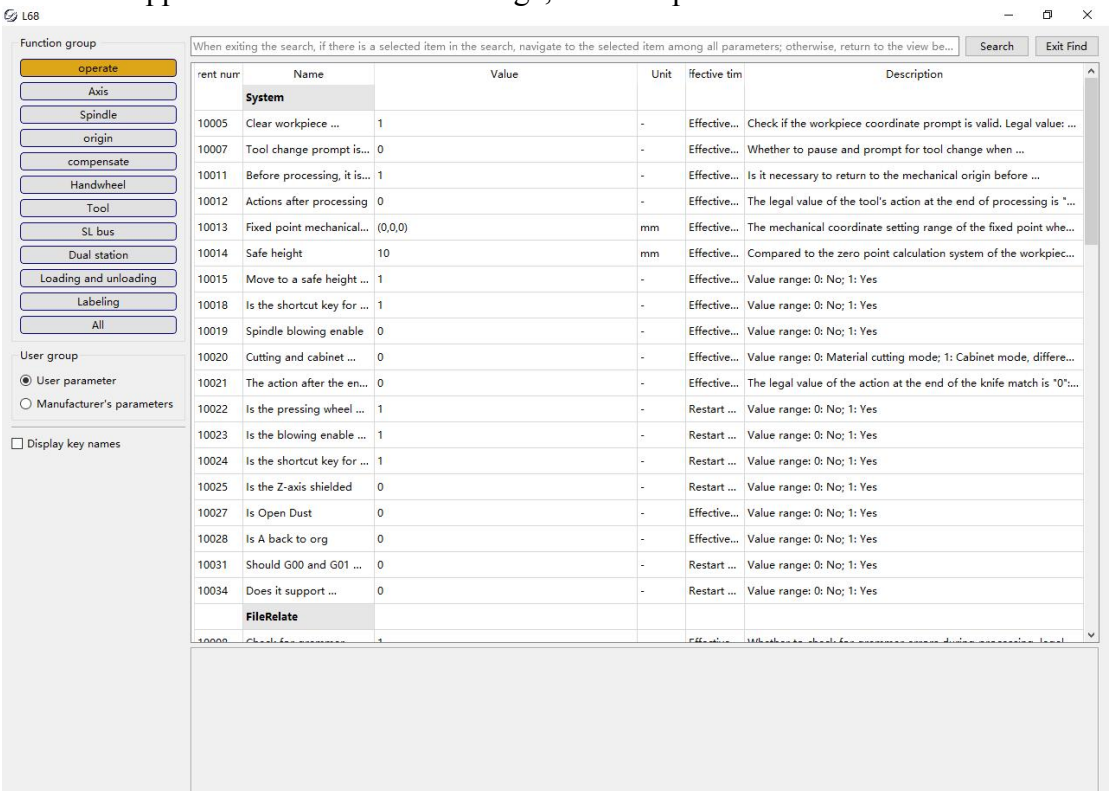


Figure 13-2 Setting Parameter Window

"User parameter" setting: To set user parameters, select the option "User Parameters" in the bottom left corner of the window, and then set the operation parameters, feed axis parameters, spindle parameters, origin parameters, compensation parameters, and tool parameters in the parameter categories. After setting, all user parameters will take effect.
"Manufacturer parameter" setting: Firstly, select the parameter permissions. To set manufacturer parameters, select the option "Manufacturer Parameters" in the bottom left corner of the window, and then set the operation parameters, feed axis parameters, spindle parameters, origin parameters, compensation parameters, and tool parameters in the parameter categories. After setting, all manufacturer parameters will take effect.
Note:
In general (default state), the parameter section displays user parameters for general machining. In order to modify machine performance related parameters, such as pulse equivalent, maximum spindle speed, etc., users need to enter a password and enable the manufacturer parameters for modification.

13.2 Restore manufacturer parameters

Used to restore the manufacturer parameters to their default values.
Select the menu item [Restore Manufacturer Parameters] in the [Parameter Management] menu to pop up the following window. Click "Yes" to enter the password (default is blank) and confirm to restore the manufacturer parameters to their default values.

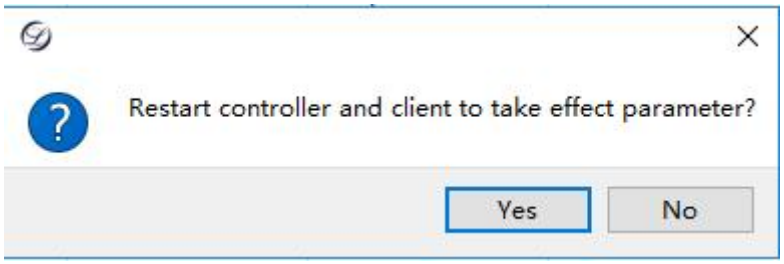


Figure 13-4 Restore Manufacturer Parameters

13.3 Backup parameters to internal controller

Used to back up system parameters internally for future use.
Select the menu item [Backup Parameters to Internal Controller] in the [Parameter Management] menu to pop up the following window, which is used to save system parameters in the form of a file to the internal system.

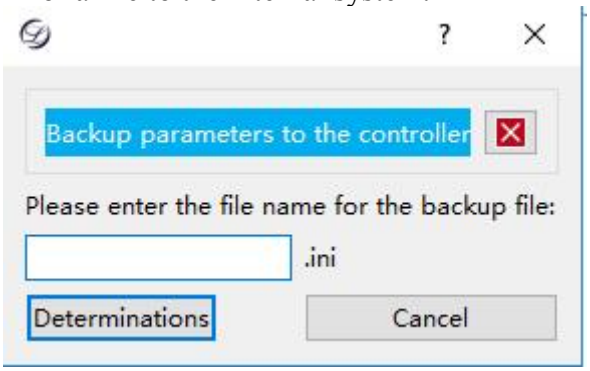


Figure 13-5 Backup Parameters To Internal Controller

13.4 Backup parameters to client path

Used to output system parameters in the form of files to a PC for data recovery.

Select the menu item [Backup Parameters to Client Path] in the [Parameter Management] menu to pop up the following window, which is used to output system parameters to a PC in the form of a file.

13.5 Restore parameters from the controller

Restore the parameters to the previously set values.

Select the menu item [Restore Parameters from the Controller] in the [Parameter Management] menu to pop up the following window, which is used to restore the parameters to the previously set values: Select the name of the parameter backup file in the pop-up window, and click the "OK" button. The system will restore the parameters to the values set at the time.

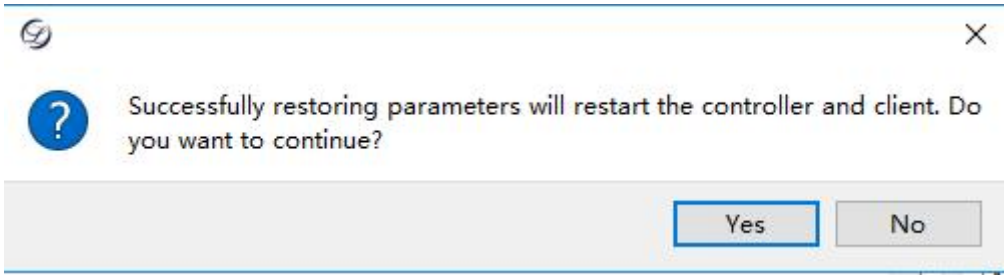


Figure 13-7 Restore Parameters from the Controller

13.6 Restore parameters from client path

Used to restore parameters saved on the PC to the system.

Select the menu item [Restore Parameters from Client Path] in the [Parameter Management] menu to pop up the following window, which is used to restore the parameters to the previously set values: Select the name of the parameter backup file in the pop-up window, and click the "OK" button. The system will restore the parameters to the values set at the time.

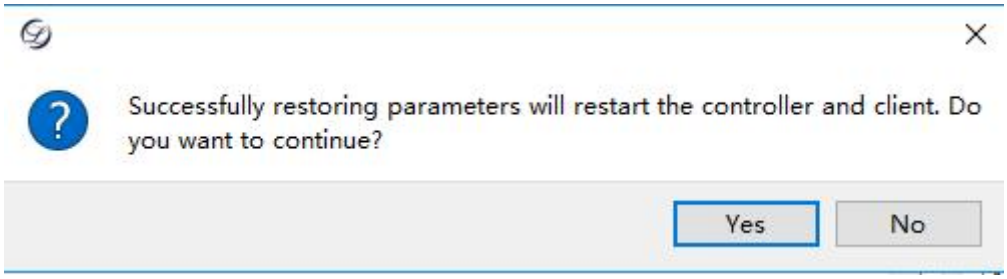


Figure 13-8 Restore Parameters from Client Path

13.7 Controller connection

If the controller is disconnected, users can click on [Controller Connection] in the [Parameter Management] menu to reconnect the controller. Click "Cancel" to exit the system

Figure 13-9 Controller Connection

13.8 Modify the password

Select the menu item [Modify the Manufacturer Password] in the [Parameter Management] menu to pop up the following window for modifying the manufacturer password, which can effectively protect the security of parameter settings.

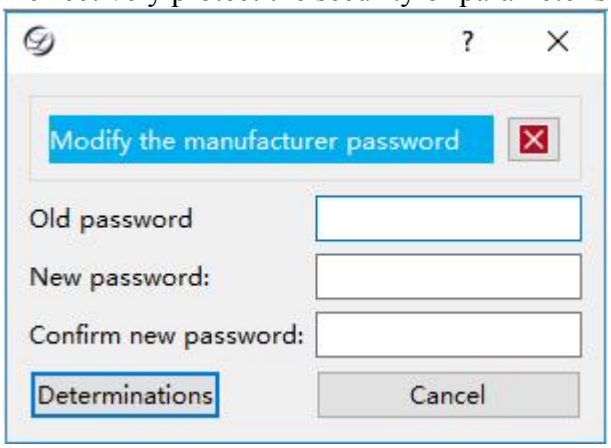


Figure 13-10 Modify the Password

The manufacturer password for the delivered equipment is blank. To modify the password, the "Old Password" does not need to be filled in. Enter the password to be modified in the "New Password", and then enter the new password to be modified again in the "Confirm New Password". After confirmation, the password will take effect immediately. When entering the manufacturer parameters again or restoring the factory parameters, a new password needs to be entered

There are many parameters involved in this system, which are divided into two categories: user parameters and manufacturer parameters. To modify and view a certain type of parameter, it is necessary to have the permissions to view and modify this type of parameter.

13.9 Modify the controller IP

Select the menu item [Modify the Control IP] in the [Parameter Management] menu to pop up the following window for modifying the controller IP, which can be modified to any IP (legal: compliant with TCP/IP protocol). After modification, click on [Restart Controller System] in the [Auxiliary] menu to restart, and make the modification take effect.

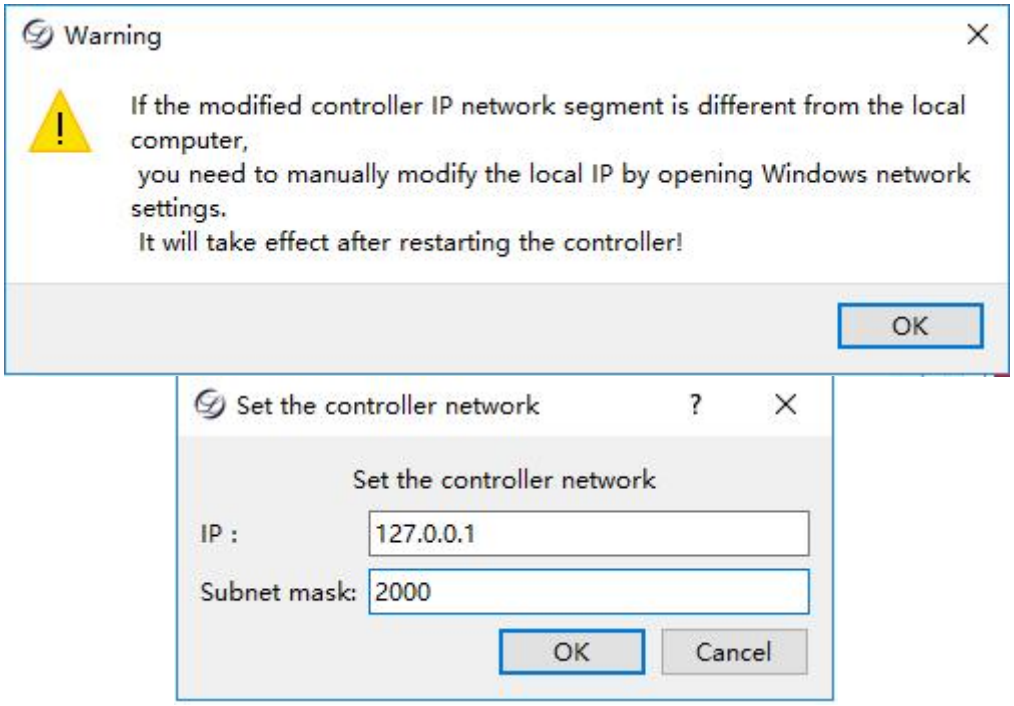


Figure 13-11 Modify the Controller IP

Note:

After modifying the controller IP, the PC IP must also be modified to be in the same network segment as the modified controller IP.

13.10 Client settings

Select the menu item [Client Settings] in the [Parameter Management] menu to pop up the following window for modifying the client interface layout, font style, and size. After modification, click [Restart Controller System] in the [Auxiliary] menu to restart, and made the modification take effect.

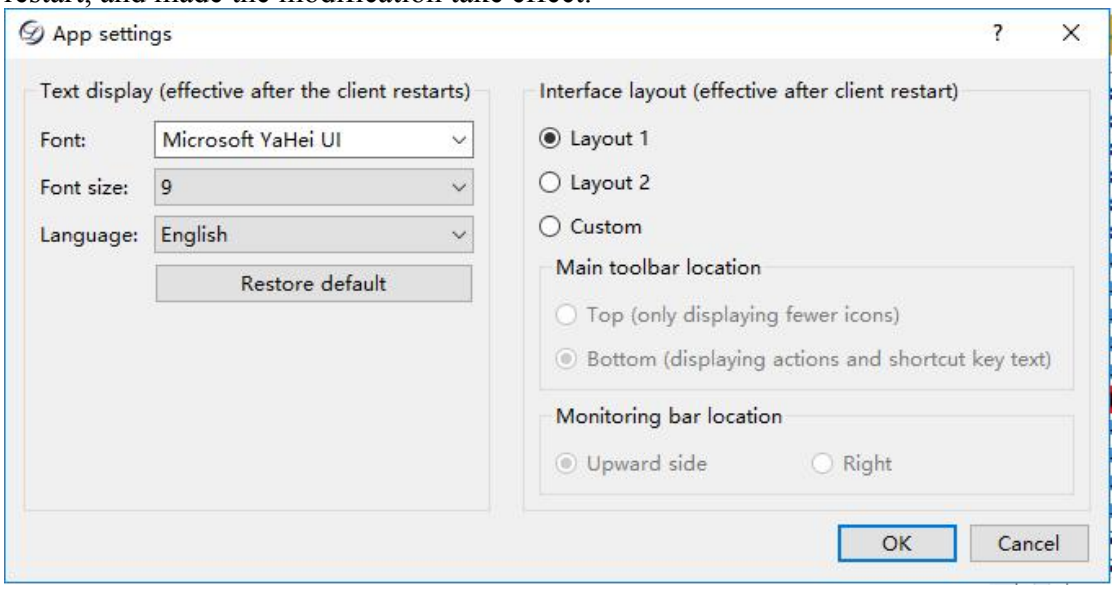


Figure 13-12 Client Settings

13.11 Parameter modification methods

The method to modify parameters is to double click on the row where the parameter is located with the mouse, pop up a dialog box, enter numerical values in the parameter area, and click to select "Yes" or "No" for "yes\no" type parameters.

The effective time of parameters can be divided into immediate effective and effective after restart. For parameter modifications that take effect immediately, the modification will take effect once completed. For parameter modifications that take effect after restart, the modification will only take effect after the system automatically restarts.

Note:

All parameters cannot be modified in the machining state, and can only be modified after the machining is completed and before the next machining begins.

13.12 Overview of user parameters

Operating parameters

No.	Parameter name		Meaning	Setting range	Permission
1.1 Manual					
N11000	Low speed under manual mode		When manually moving the machine tool at low speed, directly use the direction keys to operate	Default to 3000.000mm/min, range (takeoff speed, high speed under manual mode)	User/manufacturer
N11001	High speed under manual mode		When manually moving the machine tool at high speed, usually using CTRL + direction keys	Default to 6000.000mm/min, range (takeoff speed, 30000)	User/manufacturer
N11002	Workpiece coordinate clear prompt is valid		Whether a prompt dialog box pops up when setting the workpiece origin	Default to "Yes", range (Yes, No)	User/manufacturer
N11003	Exclude Z-axis during workpiece origin operation		Whether the Z-axis coordinate is not restored when reading the workpiece origin	Default to 0, range (Yes, No)	User/manufacturer
N81579	Whether the origin is valid	X axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to	Default to "Yes", range (Yes, No)	User/manufacturer

			"No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.		
N81580		Y axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.	Default to "Yes", range (Yes, No)	User/manufacturer
N81581		Z axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.	Default to "Yes", range (Yes, No)	User/manufacturer
N81582		A-axis	Set to "Yes". The origin serves as a mechanical	Default to "Yes", range (Yes, No)	User/manufacturer

			origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.		
1.2 Automatic					
N12000	Actions after machining		The next actions performed by the machine tool after the machining is completed	Default to 0, range (0: Hold still, 1: Return to fixed point; 2: Return to workpiece origin)	User/manufacturer
N12001	Mechanical coordinates of fixed points	X axis	Set fixed point X-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12002		Y axis	Set fixed point Y-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12003		Z axis	Set fixed point Z-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12004	Interpolation algorithm selection		Selection of machining effect and efficiency	Default to 2, range (0: High efficiency; 1: Quality first; 2: Balancing quality and efficiency)	User/manufacturer
N12005	Safety height		Reference height for machine tool stop and machining idle line	Default to 10mm, range (5, 500)	User/manufacturer

		elevation		
N12006	Idling speed	G00 command motion speed	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N12007	Default feed speed	Command motion speed for G01, G02, G03, etc.	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N12008	Use default feed speed	Whether to use parameter settings for speed, "Yes" to use parameter speed, "No" to use file internal speed.	Default to "Yes", range (Yes, No)	User/manufacturer
N12009	Z-axis cutting speed	Speed limit of the Z-axis downward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12010	Z-axis cutter lifting speed	Speed limit of the Z-axis upward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12011	Approach speed	To protect the tool, the speed at which it is prepared to come into contact with the workpiece surface	Default to 300.000mm/min	User/manufacturer

		during machining		
N12012	Approaching distance	To protect the tool, the distance starting to use the approach speed, which is the distance from the workpiece surface.	Default to 5.000mm/min	User/manufacturer
N12013	Tool change prompt is valid	Whether to pause and prompt for tool change when encountering a tool change command during the machining.	Default to "Yes", range (Yes, No)	User/manufacturer
N12014	Whether to use cycle machining	Whether to run the cycle machining. If Yes, the cycle parameters on the main interface can be set. If No, the cycle parameters on the main interface cannot be set	Default to "No", range (Yes, No)	User/manufacturer
N12015	Whether to check syntax error during machining	Whether to report errors when encountering unrecognized codes during machining. Set to "Yes" to stop machining and prompt. Set to "No" to ignore the unrecognized code and	Default to "No", range (Yes, No)	User/manufacturer

		continue machining.		
N12016	Whether to check syntax error during simulation	Whether to report an error when encountering unrecognized code during the simulation. Set to "Yes" to stop simulation and prompt. Set to "No" to ignore the unrecognized code and continue simulation.	Default to "No", range (Yes, No)	User/manufacturer
N12017	Idling acceleration	Acceleration from the takeoff speed to the maximum speed during idle motion.	Default to 650.000mm/s^2, range (1, 3000)	User/manufacturer
N12018	Idle acceleration jerk	Set the change rate of idle acceleration to alleviate the sudden acceleration and deceleration of the machine tool, avoiding excessive vibration caused by the machine tool.	Default to 10000.000/s^3, range (1, 30000)	User/manufacturer
N81578	Whether G00 fixed speed is valid	Set whether G00 fixed speed is valid	Default to "No", range (Yes, No)	User/manufacturer
N13000	Action parameter selection during pause	Action of the machine tool when machining	Default to 1, range (0: Hold still; 1: Lift to a safe height; 2:	User/manufacturer

		pauses.	Lift to the set tool lifting height)	
N13001	Z-axis tool lifting amount during pause	Parameter set for the use of tool lifting when the tool is lifted according to the tool lifting amount during pause.	Default to 10.000mm, range (1, 1000)	User/manufacturer
N13002	Tool lifting method when stopping	Action of the machine tool when machining stops.	Default to 1, range (0: Hold still; 1: Lift to a safe height; 2: Lift to the reference point)	User/manufacturer
N13003	Z-axis tool lifting when stop	Parameter set for the use of tool lifting when the tool is lifted according to the tool lifting amount during stop.	Default to 10.000mm, range (1, 1000)	User/manufacturer
N14500	Floating feeler block thickness	Set the thickness of the floating feeler block, and automatically compensate for the thickness value when performing floating tool setting	Default to 0.000mm, range (0, 500)	User/manufacturer
N14501	Whether the floating tool setting is valid	Set to "Yes". The floating tool setting on the interface is operable and executable. Set to "No". The floating tool setting on the	Default to "No", range (Yes, No)	User/manufacturer

			interface cannot be operable and executable.		
N14502	Fixed tool setting gauge position	X axis	Set the fixed tool setting position and X-axis value	Default to 0.000mm	User/manufacturer
N14503		Y axis	Set the fixed tool setting position and Y-axis value	Default to 0.000mm	User/manufacturer
N14504		Z axis	Set the fixed tool setting position and Z-axis value	Default to 0.000mm	User/manufacturer
N14505	Whether the fixed tool setting is valid		Set to "Yes". The fixed tool setting on the interface is operable and executable. Set to "No". The fixed tool setting on the interface cannot be operable and executable.	Default to "No", range (Yes, No)	User/manufacturer
N14506	Fixed tool setting gauge thickness		Set the height of the fixed tool setting gauge relative to the table	Default to 0.000mm, range (0, 1000)	User/manufacturer
N14507	Tool setting speed		Speed at which the machine tool moves from the set fixed tool setting gauge position to the tool setting gauge surface.	Default to 60mm/min, range (0, 1000)	User/manufacturer
N14508	Fast fixed tool setting speed		Speed at which the Z-axis of the machine tool moves to the	Default to 300mm/min, range (0, Z-axis cutting speed)	User/manufacturer

		Z position set by the fixed tool setting gauge.		
Others				
N17000	Whether the limit is detected during simulation	Select "Yes" to consider whether the file exceeds the software limit during the simulation. If it exceeds the limit, stop the simulation and report an error; Select "No" to not consider the file machining range during the simulation.	Default to "No", range (Yes, No)	User/manufacturer

Spindle parameters

No.	Parameter name	Meaning	Setting range	Permission
N31000	Use the default spindle speed	Whether to use parameter settings for speed, "Yes" to use parameter speed, "No" to use file internal speed.	Default to "Yes", range (Yes, No)	User/manufacturer
N31001	Stall during pause	Set to "Yes". The spindle stops rotating when encountering a pause command; Set to "No". The spindle does not stop rotating when encountering a pause command.	Default to "Yes", range (Yes, No)	User/manufacturer
N31002	Stall during stopping	Set to "Yes". The spindle stops rotating when encountering a stop	Default to "Yes", range (Yes, No)	User/manufacturer

		command; Set to "No". The spindle does not stop rotating when encountering a stop command.		

Origin parameters

No.	Parameter name		Meaning	Setting range	Permission
N41000	Before machining, it is necessary to return to the mechanical origin first		Set to "Yes". The machine must return to the mechanical origin before machining. If not executed, the system will prompt an error during machining. Set to "No". Regardless of whether to perform a return to the mechanical origin, the system can perform machining.	Default to "No", range (Yes, No)	User/manufacturer
N41503	Coarse positioning stage direction	X axis	Set to 1. The X-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The X-axis quickly searches for the mechanical origin	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			direction as the negative direction.		
N41504		Y axis	Set to 1. The Y-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Y-axis quickly searches for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41505		Z axis	Set to 1. The Z-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Z-axis quickly searches for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41506	Coarse positioning stage speed	X axis	Operating speed of the X-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer

N41507		Y axis	Operating speed of the Y-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41508		Z axis	Operating speed of the Z-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41509	Fine positioning stage direction	X axis	Set to 1. The X-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The X-axis searches again for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41510		Y axis	Set to 1. The Y-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The Y-axis searches again for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

N41511		Z axis	Set to -1. The Y-axis searches again for the mechanical origin direction as the negative direction. Set to -1. The Z-axis searches again for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41512	Fine positioning stage speed	X axis	Operating speed of the X-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41513		Y axis	Operating speed of the Y-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41514		Z axis	Operating speed of the Z-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41515	Retrace distance	X axis	Distance to retreat after the X-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41516		Y axis	Distance to retreat after the Y-axis completes searching for the mechanical	Default to 4.000mm, range (-1000, 1000)	User/manufacturer

			origin		
N41517		Z axis	Distance to retreat after the Z-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41518	Permissible movement direction during origin limit	X axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41519		Y axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41520		Z axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

13.13 Overview of manufacturer parameters

Operating parameters

No.	Parameter name		Meaning	Setting range	Permission
1.1 Manual					
N11000	Low speed under manual mode		When manually moving the machine tool at low speed, directly use the direction keys to operate	Default to 3000.000mm/min, range (takeoff speed, high speed under manual mode)	User/manufacturer
N11001	High speed under manual mode		When manually moving the machine tool at high speed, usually using CTRL + direction keys	Default to 6000.000mm/min, range (takeoff speed, 30000)	User/manufacturer
N11002	Workpiece coordinate clear prompt is valid		Whether a prompt dialog box pops up when setting the workpiece origin	Default to "Yes", range (Yes, No)	User/manufacturer
N11003	Exclude Z-axis during workpiece origin operation		Whether the Z-axis coordinate is not restored when reading the workpiece origin	Default to 0, range (Yes, No)	User/manufacturer
N81579	Whether the origin limit is valid	X axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to	Default to "1", range (Yes, No)	User/manufacturer

			"No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.		
N81580		Y axis	Set to "Yes". The origin serves as a mechanical origin reference point and a limit; Set to "No". The origin does not serve as a limit during machining, but only as a reference point for the mechanical origin.	Default to "1", range (Yes, No)	User/manufacturer
N81581		Z axis	Same as above	Default to "1", range (Yes, No)	User/manufacturer
N11500	Manual direction	X axis	Manual direction 1: Positive direction -1: Negative direction	Default to "-1", range (positive, negative)	User/manufacturer
N11501		Y axis	Manual direction 1: Positive direction -1: Negative direction	Default to "-1", range (positive, negative)	User/manufacturer
N11502		Z axis	Manual direction 1: Positive direction -1: Negative direction	Default to "1", range (positive, negative)	User/manufacturer
1.2 Automatic					

N12000	Actions after machining		The next actions performed by the machine tool after the machining is completed	Default to 0, range (0: Hold still, 1: Return to fixed point; 2: Return to workpiece origin)	User/manufacturer
N12001	Mechanical coordinates of fixed points	X axis	Set fixed point X-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12002		Y axis	Set fixed point Y-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12003		Z axis	Set fixed point Z-axis coordinates	Setting range (lower limit of table travel, upper limit of table travel)	User/manufacturer
N12004	Interpolation algorithm selection		Selection of machining effect and efficiency	Default to 2, range (0: High efficiency; 1: Quality first; 2: Balancing quality and efficiency)	User/manufacturer
N12005	Safety height		Reference height for machine tool stop and machining idle line elevation	Default to 10mm, range (5, 500)	User/manufacturer
N12006	Idling speed		G00 command motion speed	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N12007	Default feed speed		Command motion speed for G01, G02, G03, etc.	Default to 6000.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N12008	Use default feed speed		Whether to use parameter settings for	Default to "Yes", range (Yes, No)	User/manufacturer

		speed, "Yes" to use parameter speed, "No" to use file internal speed.		
N12009	Z-axis cutting speed	Speed limit of the Z-axis downward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12010	Z-axis cutter lifting speed	Speed limit of the Z-axis upward during the machining. At manual speed, the idling speed is not limited.	Default to 2500.000mm/min, range (takeoff speed, maximum speed of Z-axis)	User/manufacturer
N12011	Approach speed	To protect the tool, the speed at which it is prepared to come into contact with the workpiece surface during machining	Default to 300.000mm/min	User/manufacturer
N12012	Approaching distance	To protect the tool, the distance starting to use the approach speed, which is the distance from the workpiece surface.	Default to 5.000/min	User/manufacturer
N12013	Tool change prompt is valid	Whether to pause and	Default to "No", range (Yes, No)	User/manufacturer

		prompt for tool change when encountering a tool change command during the machining.		
N12014	Whether to use cycle machining	Whether to run the cycle machining. If Yes, the cycle parameters on the main interface can be set. If No, the cycle parameters on the main interface cannot be set	Default to "No", range (Yes, No)	User/manufacturer
N12015	Whether to check syntax error during machining	Whether to report errors when encountering unrecognized codes during machining. Set to "Yes" to stop machining and prompt. Set to "No" to ignore the unrecognized code and continue machining.	Default to "No", range (Yes, No)	User/manufacturer
N12016	Whether to check syntax error during simulation	Whether to report an error when encountering unrecognized code during the simulation. Set to "Yes" to stop simulation	Default to "No", range (Yes, No)	User/manufacturer

		and prompt. Set to "No" to ignore the unrecognized code and continue simulation.		
N12017	Idling acceleration	Acceleration from the takeoff speed to the maximum speed during idle motion.	Default to 650.000mm/s^2, range (1, 3000)	User/manufacturer
N12018	Idle acceleration jerk	Set the change rate of idle acceleration to alleviate the sudden acceleration and deceleration of the machine tool, avoiding excessive vibration caused by the machine tool.	Default to 10000.000/s^3, range (1, 30000)	User/manufacturer
N81578	Whether G00 fixed speed is valid	Set whether G00 fixed speed is valid	Default to "No", range (Yes, No)	Manufacturer
N12500	Takeoff speed	Starting speed, which can reach the set speed at the moment of startup, but if it is too high, the machine tool will vibrate	Default to 200.000mm/min, range (0, the minimum value among the maximum speeds of each axis)	Manufacturer
N12501	Individual axis acceleration	Acceleration or deceleration of the axis when moving in a straight line. The	Default to 500.000mm/s^2, range (0.01, 2000)	Manufacturer

			greater the acceleration is, the shorter the time to reach the same speed will be.		
N12502	Turning acceleration		Maximum acceleration of the feed axis on adjacent axes during curve interpolation.	Default to 800.000mm/s^2, range (1, 100000)	Manufacturer
N12503	Acceleration jerk		Change rate of idle acceleration to alleviate the sudden acceleration and deceleration of the machine tool, avoiding excessive vibration caused by the machine tool.	Default to 12000.000mm/s^3, range (0.01, 300000)	Manufacturer
N12504	Maximum speed of each axis	X axis	Maximum running speed of X-axis	Default to 6000.000mm/min	Manufacturer
N12505		Y axis	Maximum running speed of Y-axis	Default to 6000.000mm/min	Manufacturer
N12506		Z axis	Maximum running speed of Z-axis	Default to 6000.000mm/min	Manufacturer
N12507	Minimum speed for arc machining		Minimum speed when performing arc machining	Default to 250.000mm/min	Manufacturer
N12508	Maximum speed for arc machining		Maximum speed when performing arc	Default to 9000.000mm/min	Manufacturer

		machining		
N12509	Arc radius tolerance	In the IJK increment representation of G02 and G03, the radius of the circle is calculated twice, but generally, the values obtained from these two calculations are different. The difference between them is the arc radius tolerance, which is generally not significant. The recommended error is 0.1	Default to 1.000mm, range (0, 10)	Manufacturer
N12510	Whether the arc speed limit is valid	Select "Yes" to limit the speed during arc machining; Select "No" to not limit the line during arc machining.	Default to "Yes", range (Yes, No)	Manufacturer
N12511	Allowable chord height error during arc machining	During the machining, the points on the curve are calculated and finally fitted to determine the error value of the chord height	Default to 0.010mm, range (0, 0.1)	Manufacturer

		calculated between points.		
N12512	Reference circle radius	Reference for machining circular workpieces by machine tools when referring to circles	Default to 5mm	Manufacturer
N12513	Reference circle speed	Reference speed when performing arc machining	Default to 3000.000mm/min	Manufacturer
N12514	Z-direction slow deceleration speed	During the machining, the speed at which each contact occurs to protect the workpiece.	Default to 300mm/min	Manufacturer
N12515	Corner tolerance	For the overall smoothness of the workpiece, there is a slight accuracy error between every two segments of the program, and for the error within the set value range, the program can be operated.	Default to 0.020mm, range (0, 0.1)	Manufacturer
N12516	Smoothing time	The larger the parameter is, the smoother the machining surface will be. However, too large a	Default to 0.020 second, range (0, 0.06)	Manufacturer

		parameter can easily cause significant changes in the workpiece size. It is recommended to set the high-precision of molds to 0.01 and woodworking machines to 0.03		
1.3 Pause				
N13000	Action reference selection during pause	Whether to lift to a safe height during pause, 0: Hold still 1: Lift to a safe height 2: Lift to the set tool lifting height	Default to 1, range (0, 2)	User/manufacturer
N13001	Z-axis tool lifting amount during pause	Parameter set for the use of tool lifting when the tool is lifted according to the tool lifting amount during pause.	Default to 10.000mm, range (1, 1000)	User/manufacturer
N13002	Tool lifting method when stopping	Action of the machine tool when machining stops.	Default to 1, range (0: Hold still; 1: Lift to a safe height; 2: Lift to the reference point)	User/manufacturer
N13003	Z-axis tool lifting when stop	Parameter set for the use of tool lifting when the tool is lifted according to	Default to 10.000mm, range (1, 1000)	User/manufacturer

			the tool lifting amount during stop.		
N14500	Floating feeler block thickness		Set the thickness of the floating feeler block, and automatically compensate for the thickness value when performing floating tool setting	Default to 0.000mm, range (0, 500)	Manufacturer
N14501	Whether the floating tool setting is valid		Set to "Yes". The floating tool setting on the interface is operable and executable. Set to "No". The floating tool setting on the interface cannot be operable and executable.	Default to "No", range (Yes, No)	Manufacturer
N14502	Fixed tool setting gauge position	X axis	Set the fixed tool setting position and X-axis value	Default to 0.000mm	User/manufacturer
N14503		Y axis	Set the fixed tool setting position and Y-axis value	Default to 0.000mm	User/manufacturer
N14504		Z axis	Set the fixed tool setting position and Z-axis value	Default to 0.000mm	User/manufacturer
N14505	Whether the fixed tool setting is valid		Set to "No". The fixed tool setting on the interface cannot be	Default to "No", range (Yes, No)	User/manufacturer

		operable and executable. Set to "No". The fixed tool setting on the interface cannot be operable and executable.		
N14506	Fixed tool setting gauge thickness	Set the height of the fixed tool setting gauge relative to the table	Default to 0.000mm, range (0, 1000)	User/manufacturer
N14507	Tool setting speed	Speed at which the machine tool moves from the set fixed tool setting gauge position to the tool setting gauge surface.	Default to 60mm/min, range (0, 1000)	User/manufacturer
N14508	Fast fixed tool setting speed	Speed at which the Z-axis of the machine tool moves to the Z position set by the fixed tool setting gauge.	Default to 300mm/min, range (0, Z-axis cutting speed)	User/manufacturer
N15000	Whether the arc IJK increment is valid	Legal value, "Yes": Valid; "No": Invalid	Default to "Yes", range (Yes, No)	
Others				
N17000	Whether the limit is detected during simulation	Select "Yes" to consider whether the file exceeds the software limit during the simulation. If it exceeds the limit, stop the	Default to "No", range (Yes, No)	User/manufacturer

		simulation and report an error; Select "No" to not consider the file machining range during the simulation.		

Feed shaft parameters

No.	Parameter name		Meaning	Setting range	Permission
N21500	Impulse equivalent	X axis	Distance of machine tool X-axis movement for each pulse sent by the system	Default to 0.00100000mm/p	Manufacturer
N21501		Y axis	Distance of machine tool Y-axis movement for each pulse sent by the system	Default to 0.00100000mm/p	Manufacturer
N21502		Z axis	Distance of machine tool Z-axis movement for each pulse sent by the system	Default to 0.00100000mm/p	Manufacturer
N21503	Whether the table travel range check is valid	X axis	Set to "Yes". The table travel on the X-axis is valid. Set to "No". The table travel on the X-axis is invalid.	Default to "No", range (Yes, No)	Manufacturer
N21504		Y axis	Set to "Yes". The table travel on the Y-axis is valid. Set to "No". The table travel on the Y-axis is invalid.	Default to "No", range (Yes, No)	Manufacturer
N21505		Z axis	Set to "Yes". The table travel on the Z-axis is valid. Set to "No". The table travel on the Z-axis is invalid.	Default to "No", range (Yes, No)	Manufacturer

N21506	Lower limit of table travel (mechanical coordinates)	X axis	Minimum value for X-axis table operation.	Default to 0.000mm	Manufacturer
N21507		Y axis	Minimum value for Y-axis table operation.	Default to 0.000mm	Manufacturer
N21508		Z axis	Minimum value for Z-axis table operation.	Default to -100.000mm	Manufacturer
N21509	Upper limit of table travel (mechanical coordinates)	X axis	Maximum value for X-axis table operation.	Default to 2500.000mm	Manufacturer
N21510		Y axis	Maximum value for Y-axis table operation.	Default to 2500.000mm	Manufacturer
N21511		Z axis	Maximum value for Z-axis table operation.	Default to 0.000mm	Manufacturer

Spindle parameters

No.	Parameter name	Meaning	Setting range	Permission
N31000	Use the default spindle speed	Whether to use parameter settings for speed, "Yes" to use parameter speed, "No" to use file internal speed.	Default to "Yes", range (Yes, No)	User/manufacturer
N31001	Stall during pause	Set to "Yes". The spindle stops rotating when encountering a pause command; Set to "No". The spindle does not stop rotating when encountering a pause command.	Default to "Yes", range (Yes, No)	User/manufacturer
N31002	Stall during stopping	Set to "Yes". The spindle stops rotating when encountering a stop command; Set to "No". The spindle does not stop rotating when encountering a stop command.	Default to "Yes", range (Yes, No)	User/manufacturer

N31500	Maximum speed	Set the maximum speed corresponding to the spindle at 10V analog output	Default to 24000.000rpm, range (0, 100000)	Manufacturer
N31501	Default speed	Actual speed of the spindle	Default to 24000.000rpm, range (0, maximum speed)	Manufacturer
N31502	Spindle start delay	Time from spindle startup to maximum spindle speed	Default to 10.000 seconds, range (0.5, 300)	Manufacturer
N31503	Spindle stop delay	Time from maximum spindle speed to spindle stop	Default to 5.000 seconds, range (1, 300)	Manufacturer

Origin parameters

No.	Parameter name		Meaning	Setting range	Permission
N41000	Before machining, it is necessary to return to the mechanical origin first		Set to "Yes". The machine must return to the mechanical origin before machining. If not executed, the system will prompt an error during machining. Set to "No". Regardless of whether to perform a return to the mechanical origin, the system can perform machining.	Default to "No", range (Yes, No)	User/manufacturer
N41503	Coarse positioning stage direction	X axis	Set to 1. The X-axis quickly searches for the mechanical origin direction as	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			the positive direction; Set to -1. The X-axis quickly searches for the mechanical origin direction as the negative direction.		
N41504		Y axis	Set to 1. The Y-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Y-axis quickly searches for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41505		Z axis	Set to 1. The Z-axis quickly searches for the mechanical origin direction as the positive direction; Set to -1. The Z-axis quickly searches for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer

N41506	Coarse positioning stage speed	X axis	Operating speed of the X-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41507		Y axis	Operating speed of the Y-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41508		Z axis	Operating speed of the Z-axis in coarse positioning to find the origin	Default to 1200.000mm/min, range (takeoff speed, maximum speed of each axis)	User/manufacturer
N41509	Fine positioning stage direction	X axis	Set to 1. The X-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The X-axis searches again for the mechanical origin direction as the negative direction.	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41510		Y axis	Set to 1. The Y-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The Y-axis searches again for the	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			mechanical origin direction as the negative direction.		
N41511		Z axis	Set to 1. The Z-axis searches again for the mechanical origin direction as the positive direction; Set to -1. The Z-axis searches again for the mechanical origin direction as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41512	Fine positioning stage speed	X axis	Operating speed of the X-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41513		Y axis	Operating speed of the Y-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41514		Z axis	Operating speed of the Z-axis in fine positioning to find the origin	Default to 60.000mm/min, range (0.1, coarse positioning stage speed)	User/manufacturer
N41515	Retrace distance	X axis	Distance to retreat after the X-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer

N41516		Y axis	Distance to retreat after the Y-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41517		Z axis	Distance to retreat after the Z-axis completes searching for the mechanical origin	Default to 4.000mm, range (-1000, 1000)	User/manufacturer
N41518	Permissible movement direction during origin limit	X axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41519		Y axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the positive direction and -1 as the negative direction.	Default to 1, range (1: positive direction; -1: negative direction)	User/manufacturer
N41520		Z axis	When subjected to the origin limit, the direction of machine tool movement is set to 1 as the	Default to -1, range (1: positive direction; -1: negative direction)	User/manufacturer

			positive direction and -1 as the negative direction.		

Compensation parameters

No.	Parameter name		Meaning	Setting range	Permission
N51500	Effective compensation for screw error		This parameter can be set to determine whether to perform screw error compensation, including reverse clearance compensation and pitch compensation	Default to "No", range (Yes, No)	Manufacturer
N51501	The reverse clearance compensation is valid		Set the reverse clearance of each axis screw	Default to "No", range (Yes, No)	Manufacturer
N51502	The tool compensation is valid		Set to "No". The tool compensation parameters are invalid. Set to "Yes". The tool compensation is valid.	Default to "No", range (Yes, No)	Manufacturer
N51503	Reverse clearance	X axis	The clearance value in the X-axis direction will be compensated multiple times if it exceeds 0.2	Default to 0.000mm, range (0, 1)	Manufacturer
N51504		Y axis	The clearance value in the Y-axis direction will be compensated multiple times if it exceeds 0.2	Default to 0.000mm, range (0, 1)	Manufacturer
N51505		Z axis	The clearance value in the Z-axis direction	Default to 0.000mm, range (0, 1)	Manufacturer

			will be compensated multiple times if it exceeds 0.2		
N71002	Handwheel	Handwheel acceleration	The smaller the value is, the smoother the speed will be	Default to 200.000m, range (1, 6000)	Manufacturer
N71006		Strict handwheel pulse calculation	If strict handwheel counting is used, the system will move the distance specified; otherwise, the machine tool will only move when the handwheel is shaking	Default to 1, range (0, 1)	Manufacturer
N71007		Handwheel direction	Set the handwheel to control the moving direction, -1: Negative; 1: Positive	Default to 1, range (-1, 1)	Manufacturer

Chapter XIV Miscellaneous Function

The [Auxiliary] menu contains items related to non auxiliary functions. Click on the [Auxiliary] menu with the mouse to pop up the drop-down menu:

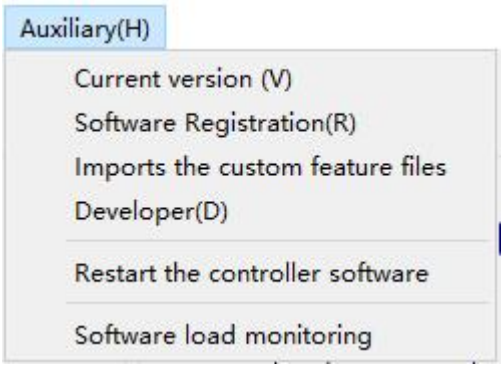


Figure 14-1 "Auxiliary" Menu on the Menu Bar

14.1 Current version

Select the menu item [Current Version] in the [Auxiliary] menu to provide relevant information such as the current system software version

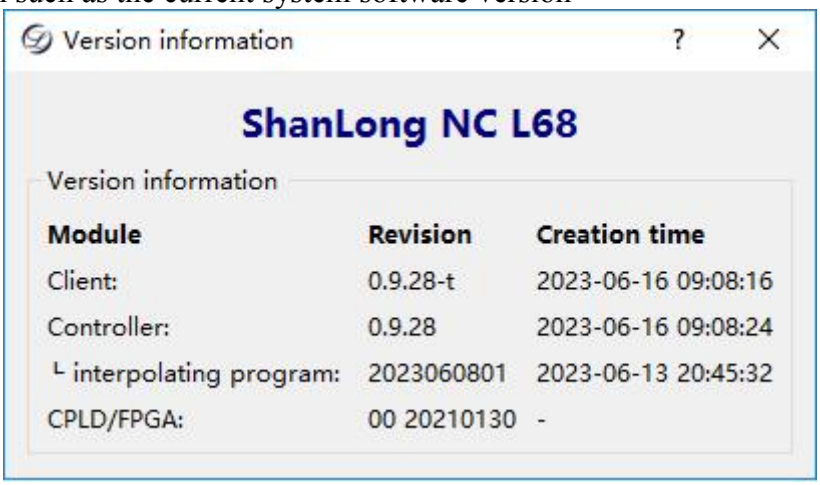


Figure 14-1 Version Information

14.2 Restart of controller software

Select the menu item [Restart Controller Software] in the [Auxiliary] menu to execute a client restart.

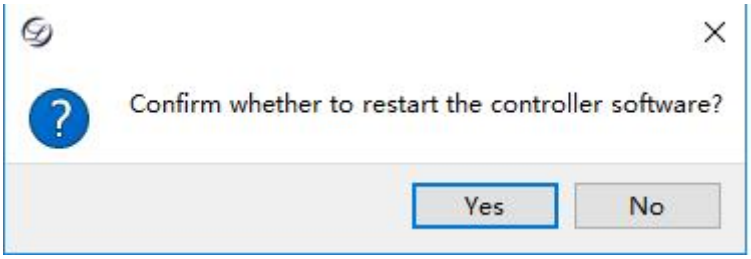


Figure 14-5 Restart Controller Software

Select the menu item [Restart Controller System] in the [Auxiliary] menu to execute the controller restart.

14.3 Program load monitoring

Select the menu item [Program Load Monitoring] in the [Auxiliary] menu to provide relevant information about the current system communication cycle

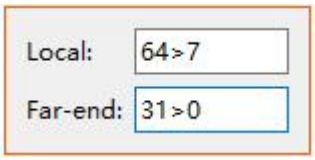


Figure 14-7 Program Load Monitoring

Chapter XV IO Status

There are 27 input IO channels and 17 output IO channels, which can be freely set; Master control IO:1-8 and expansion board IO:33-48 can be used once configured. If the IO port is configured as "0", it is not configured (not used).

The IO status window displays the input/output port description, port number, polarity, and current IO status. When IO is turned on, the IO turns red in the IO status; When IO is turned off, the IO turns green in the IO status. Users can configure ports and view the IO status window for input signals or signal outputs.

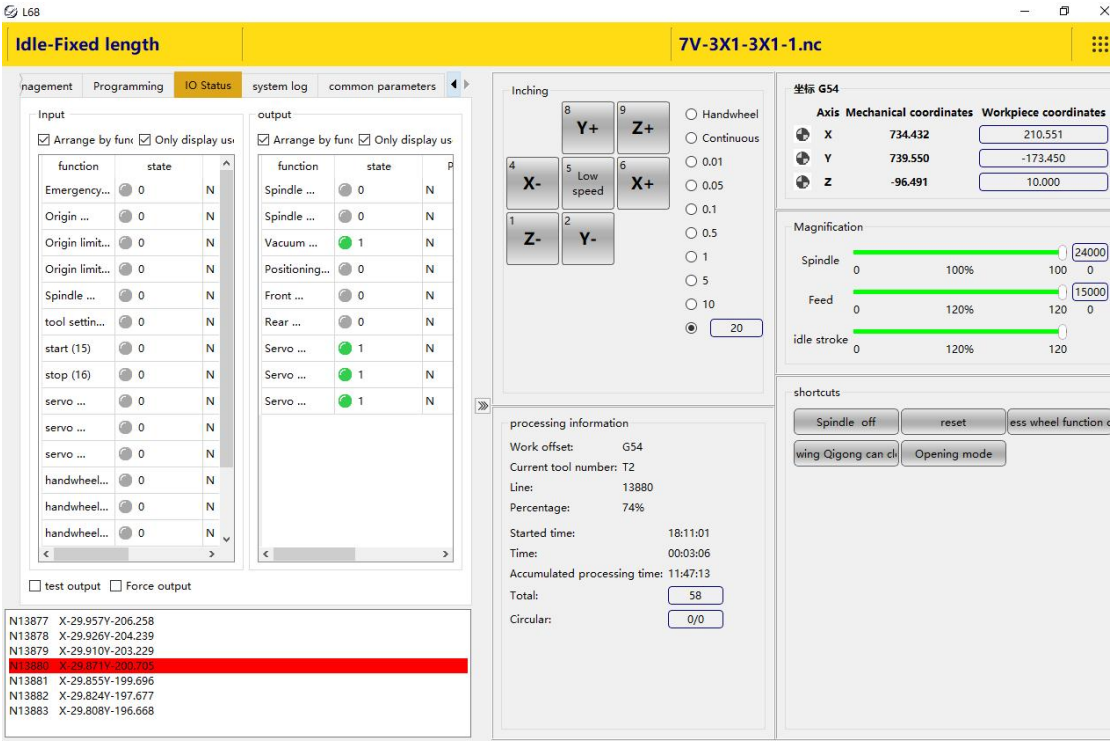


Figure 15-1 IO Status Window

15.1 Output IO test

Users can click on [Start Test] at the bottom of the interface to test the output port IO for signal output. Select the output port and click on the "Test Switch" to perform the test. Click "Cancel All" to cancel the testing of the selected port. After the test is completed, click "Exit Test" to exit the testing status.

15.2 Input/output IO configuration

Select an input/output port and double-click it or click [Set Port Number] under the main interface, enter the port number and click "OK". Click "OK" to pop up a dialog box, select "Yes" and restart the system to complete the port configuration. To modify the port number or reverse polarity for the first time, users need to enter the manufacturer password.

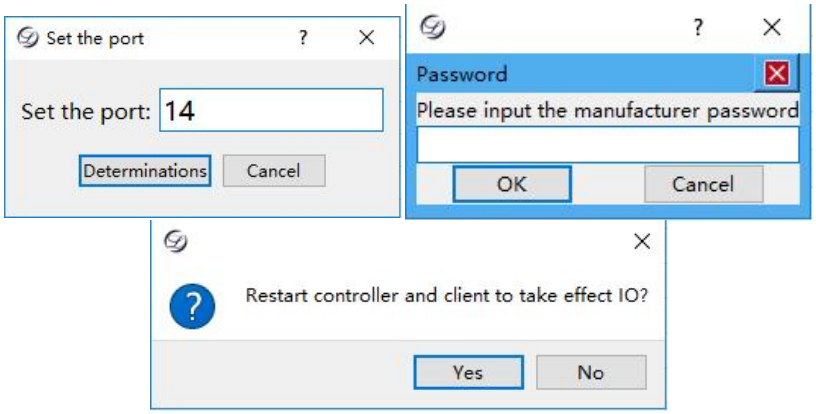


Figure 15-2 IO Configuration

Figure 15-3 Default Input IO

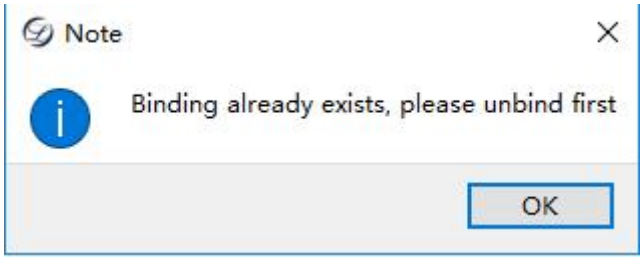
Polarity	Port	Description	Polarity	Port	Description
N	2	Emergency stop	N	4	Stop
N	16	X-axis origin limit	N	65	X-axis servo alarm
N	13	Y-axis origin limit	N	66	Y-axis servo alarm
N	10	Z-axis origin limit	N	67	Z-axis servo alarm
N	15	X-axis positive limit	N	79	Handwheel input X-axis
N	12	Y-axis positive limit	N	80	Handwheel input Y-axis
N	9	Z-axis positive limit	N	81	Handwheel input Z-axis
N	14	X-axis negative limit	N	85	Handwheel input X1
N	11	Y-axis negative limit	N	86	Handwheel input X10
N	8	Z-axis negative limit	N	87	Handwheel input X100
N	77	Spindle alarm	N	71	X-axis z-signal
N	1	Tool setting	N	72	Y-axis z-signal
N	3	Start machining	N	73	Z-axis z-signal

Figure 15-3 Default Output IO

Polarity	Port	Description	Polarity	Port	Description
N	5	Band-type brake	N	79	Spindle speed control
N	4	Red light	N	6	Expansion output 2
N	3	Green light	N	78	Spindle enable _ reverse
N	2	Cooling	N	65	X-axis servo enable
N	1	Automatic machining completed	N	66	Y-axis servo enable
N	77	Spindle enable	N	67	Z-axis servo enable
N	8	Yellow light	N	71	X-axis servo alarm cleared
N	7	Expansion	N	72	Y-axis servo alarm

		output 1			cleared
			N	73	Z-axis servo alarm cleared

If duplicate port numbers are configured, a prompt will appear as shown in the following figure:



15.3 Port polarity modification

Select an input/output port, click [Reverse Polarity] to pop up a dialog box, and click "OK" to pop up another dialog box. Select "Yes", restart the system, and complete the modification of port polarity.

Figure 15-3 Modify Port Polarity