NcStudio V10 Glass Cutting CNC System User Manual

Version: 2018.11 10th Version
Author: Document Department

1 Check Configuration and Wiring

To use NcStudio V10 glass cutting CNC system, checking configuration and wiring is required.

1.1 Prerequisites

In bus type configuration mode, please check the hardware configuration as following list:

- Lambda 5M controller
- Servo drives
- Servo motors
- PM95A control card
- Host computer (with PCIE slot)
- EX31A terminal board
- Optional: handwheel

In non-bus type configuration mode, please check the hardware configuration as following list:

- Lambda 3S/3L/4S/5S/5E controller
- Servo drives
- Servo motors
- PM95A control card
- Host computer (with PCIE slot)
- EX27A/EX31A terminal board (For Lambda 3L controller, a terminal board is not required.)
- Optional: handwheel

Lambda 3S/4S controller matches EX27A terminal board, and Lambda 5S/5E controller matches EX31A terminal board.

1.2 Steps

To check configuration and wiring, do the following:

1. Confirm space for installation
2. Install control card
3. Hardware wiring
4. Wiring of I/O port

1.2.1 Confirm Space for Installation

To install the hardware device, please check the specification of the product and ensure that there is sufficient space for installation.

Please refer to Related Dimensional Drawing for details.

1.2.2 Install Control Card

To install control card, do the following:

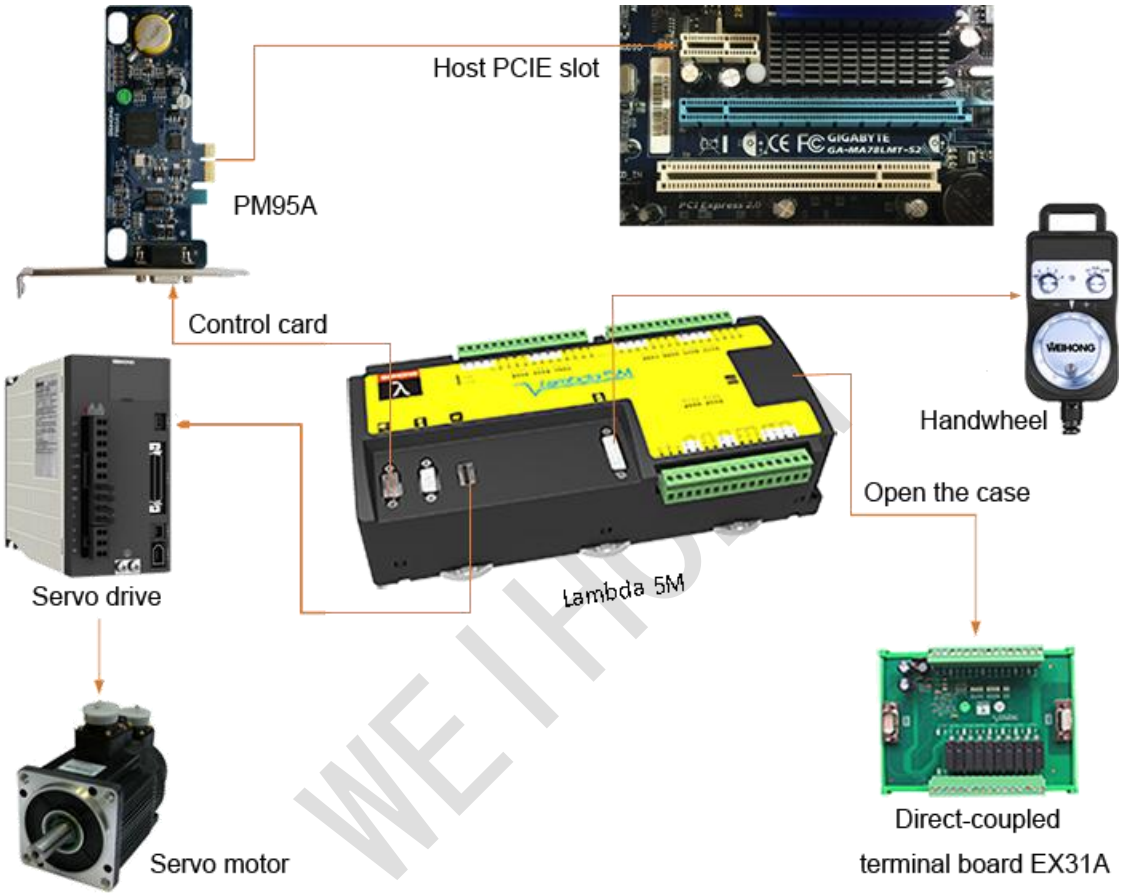
1. Turn off the power supply of host computer, and open the case.
2. Insert the motion control card into PCIE slot.
3. Tighten the mounting screw, and close the case.
4. Restart the computer.

1.2.3 Hardware Wiring

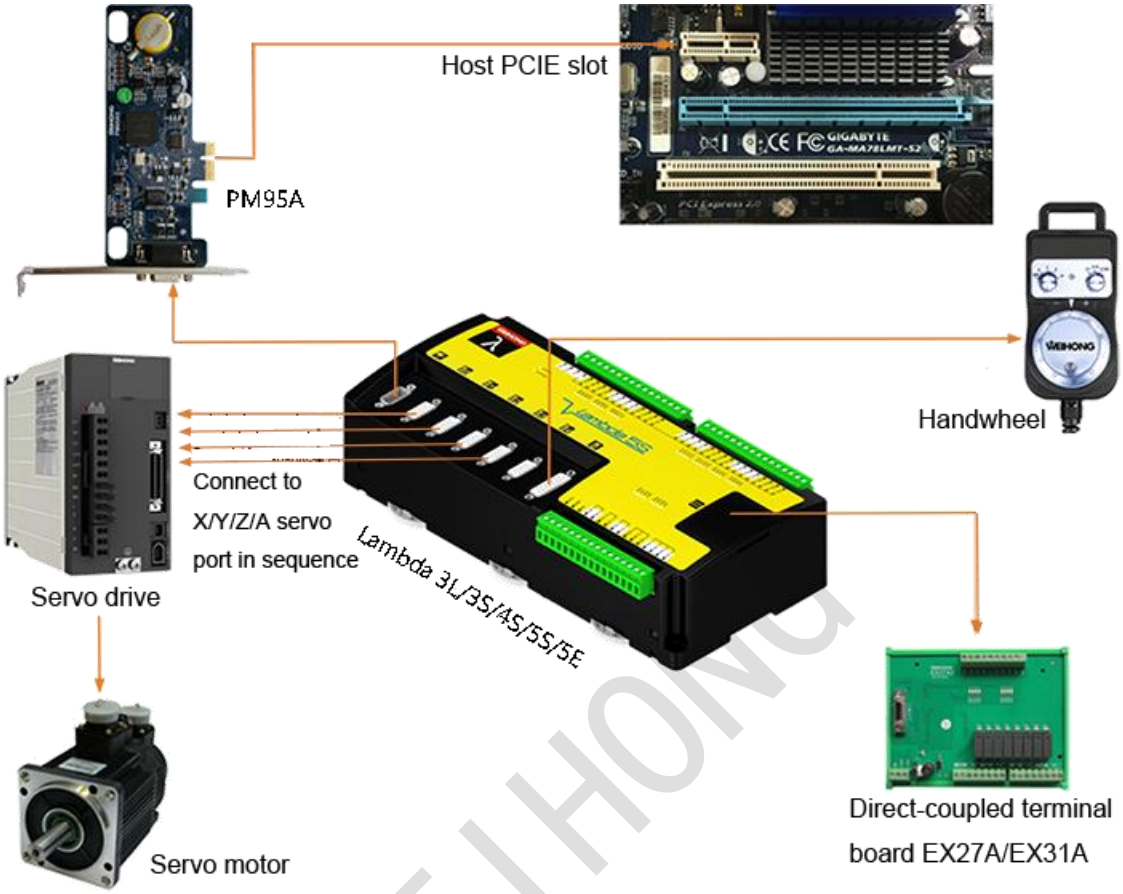
This part introduces hardware wiring in bus type and non-bus type configurations.

Taking PM95A control card as an example, connect as follows:

In bus type configuration:



In non-bus type configuration:



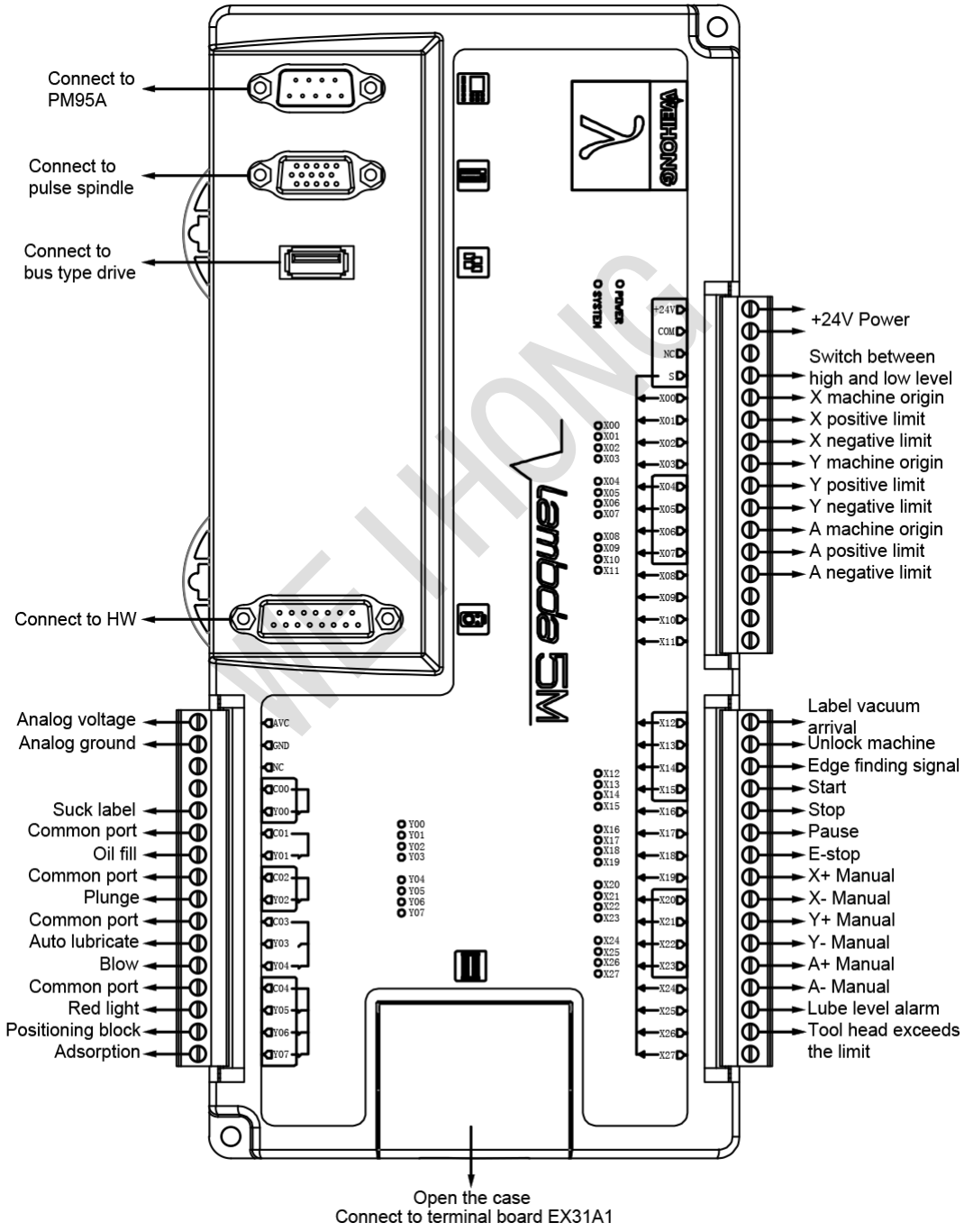
1.2.4 Wiring of I/O Port

This part introduces wiring of I/O port in bus type and non-bus type configurations.

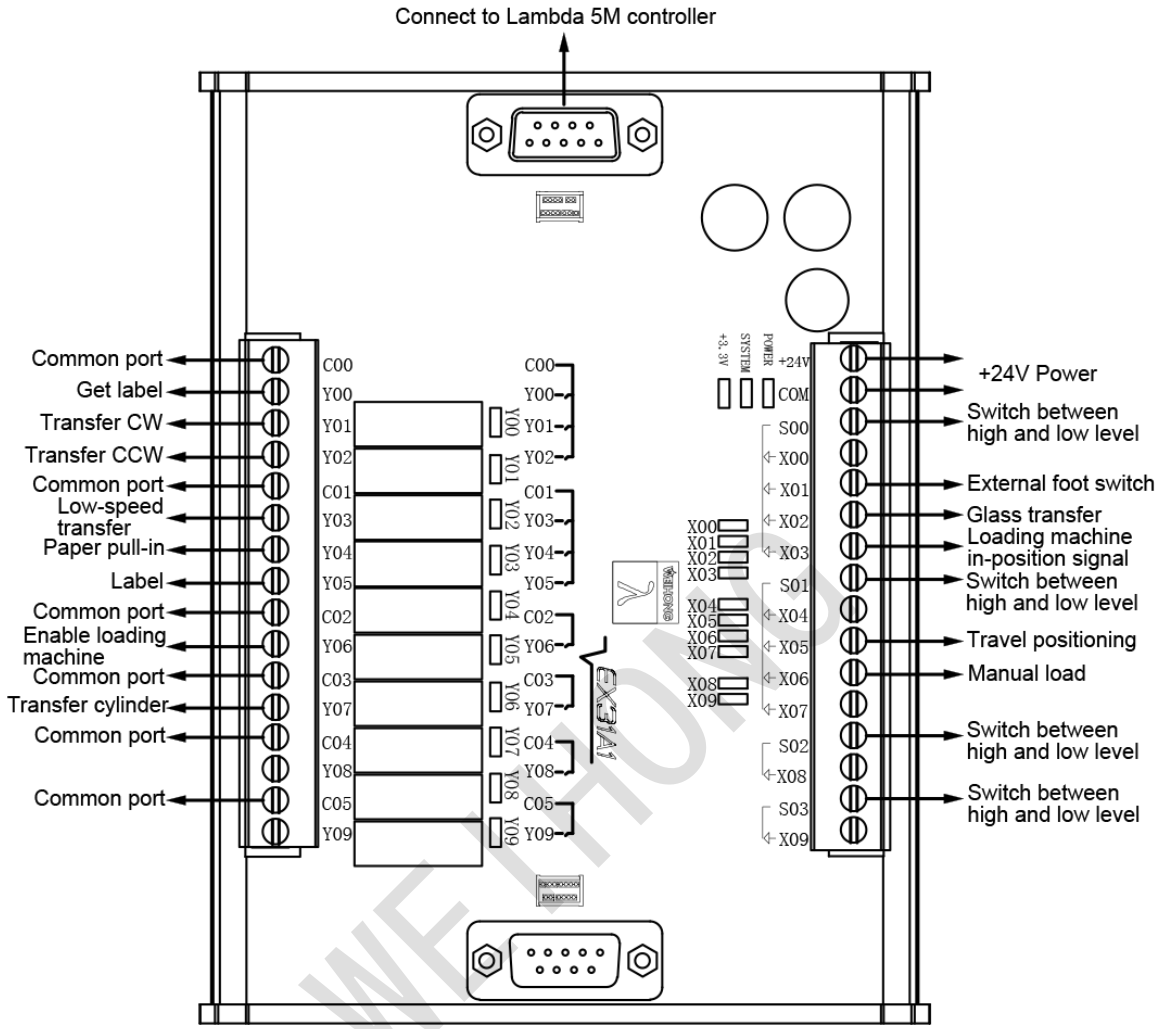
Connect the product as follows:

In bus type configuration:

- Wiring diagram of I/O port of Lambda 5M controller is as follows:

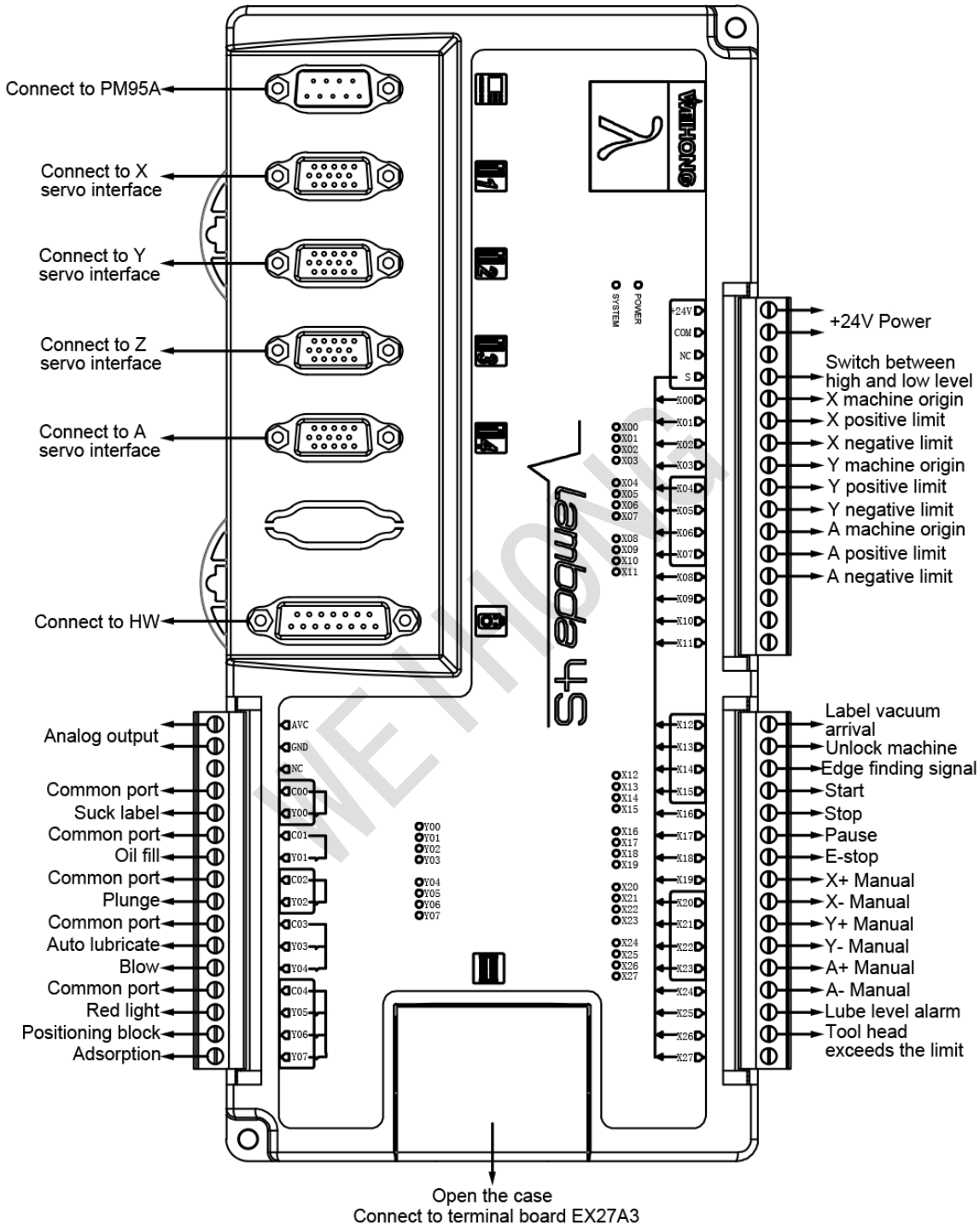


- Wiring diagram of I/O port of EX31A terminal board is as follows:

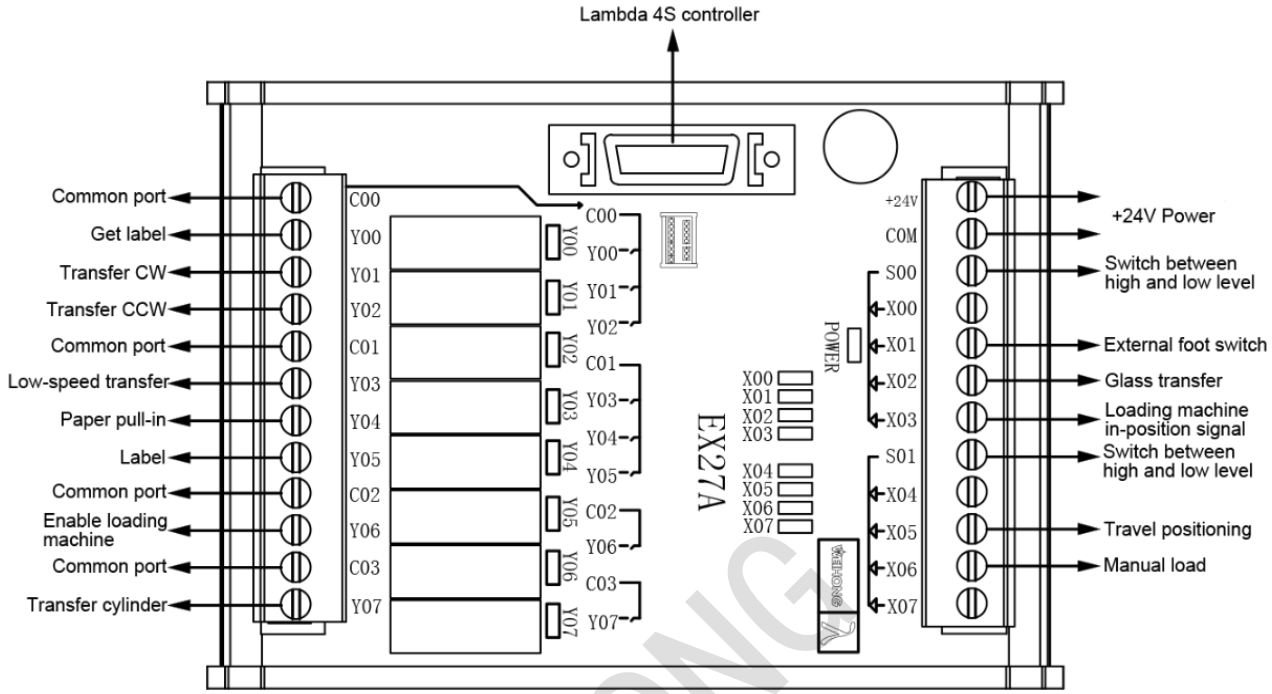


In non-bus type configuration:

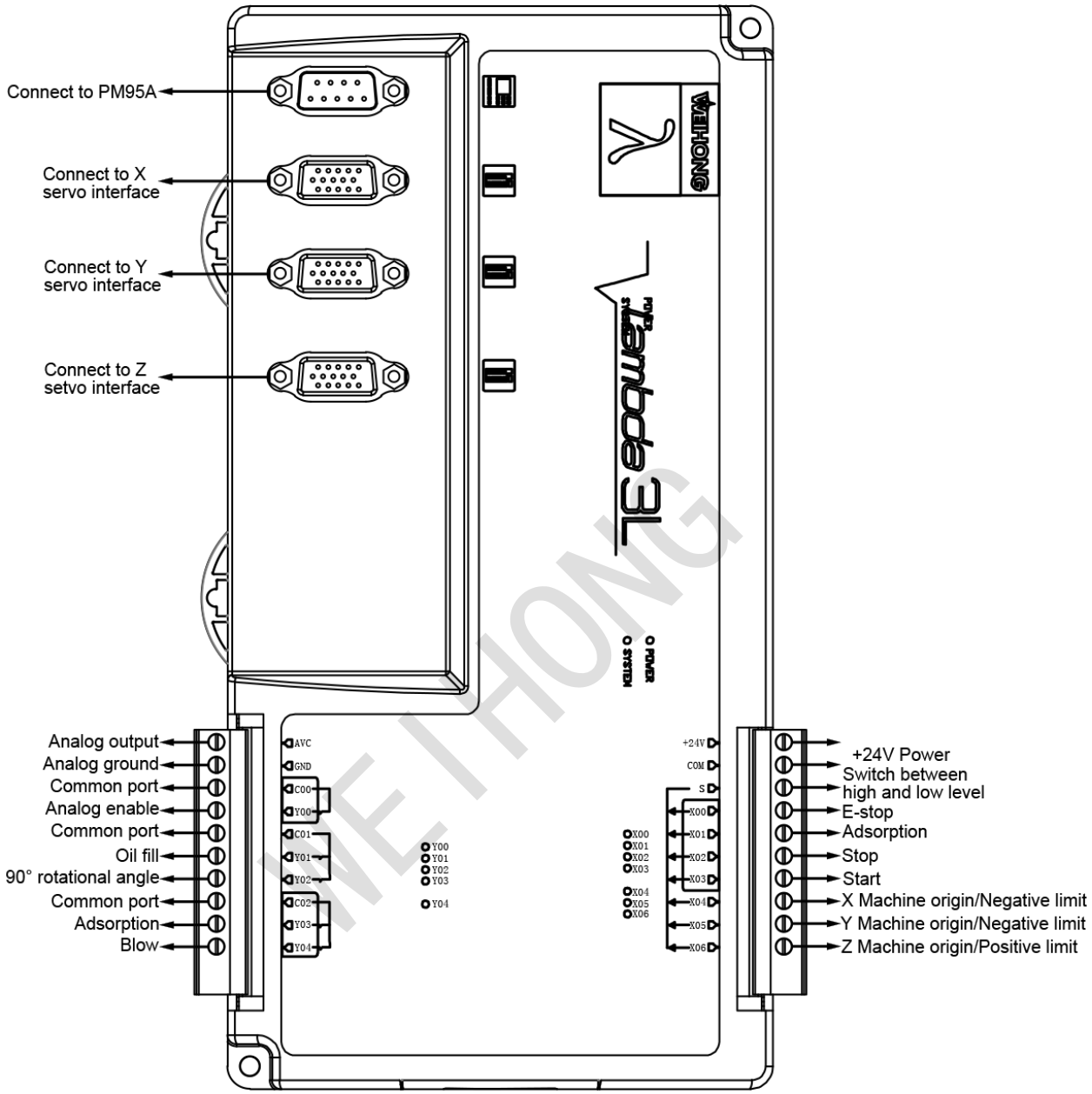
- Wiring diagram of I/O port of Lambda 4S controller is as follows:



- Wiring diagram of I/O port of EX27A terminal board is as follows:



- Wiring diagram of I/O port of Lambda 3L controller is as follows:



2 Install Software

To use NcStudio V10 glass cutting CNC system, installing NcStudio software is required.

2.1 Prerequisites

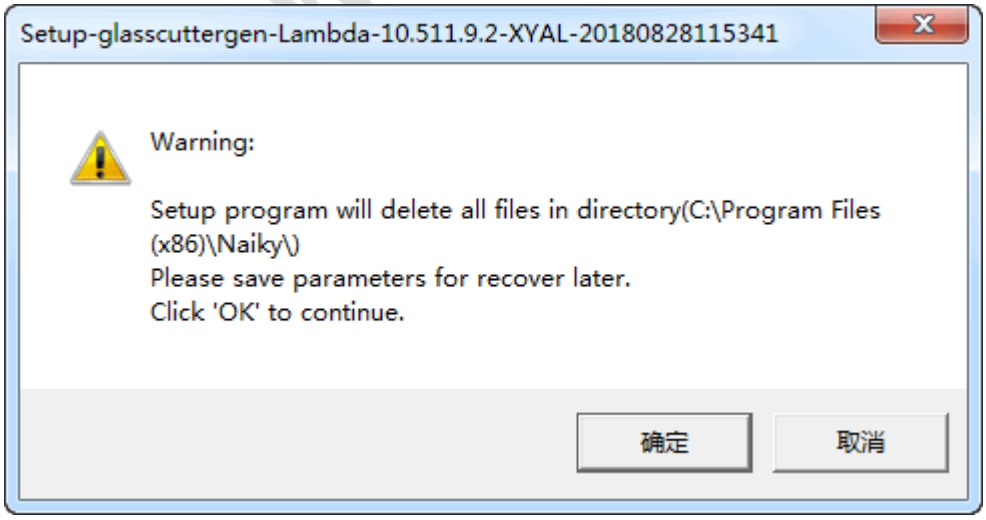
Before installing NcStudio, please confirm the configuration of the computer meets the requirements in the table below:

Name	Requirement
CPU	Main frequency 1G or above
Memory	512M or above
Hard disk	20G or above
Graphic card	1024*768 at least
Display	VGA 14" or above
CD-ROM	4X or higher (optional)
Mainboard expansion slot	More than 1 PCIE slot (with PM95A control card)

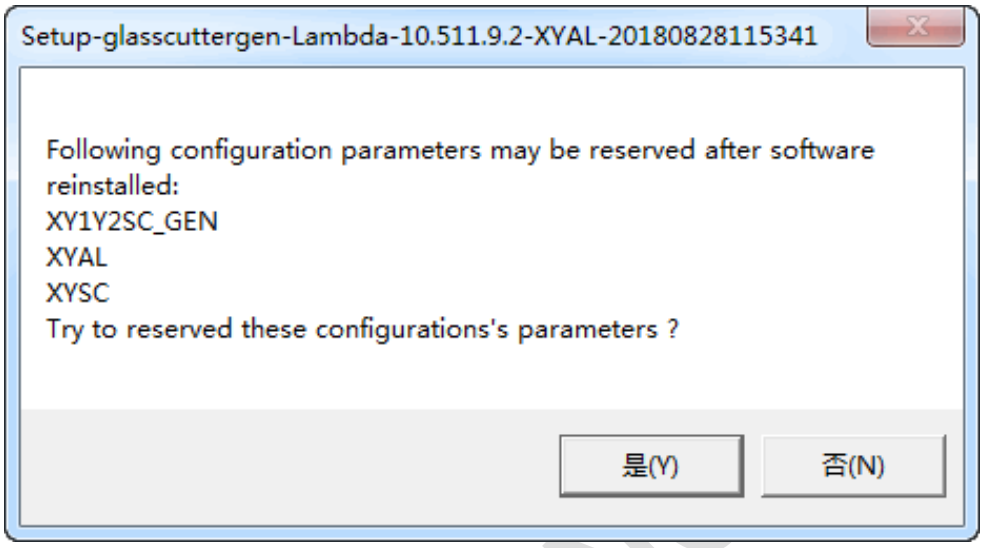
2.2 Steps

To install NcStudio, do the following:

1. Double click  .
2. Select the display language.
3. Click **OK** to save parameters in the following dialog box:



- If it is your first time to install NcStudio, a dialog box will pop up to prompt that installation completes.
- If it is not your first time to install NcStudio, a dialog box will pop up:



Select whether to save the configuration parameters according to actual situation.

Installation completes.

3 Interface Overview

NcStudio V10 glass cutting software includes the following:

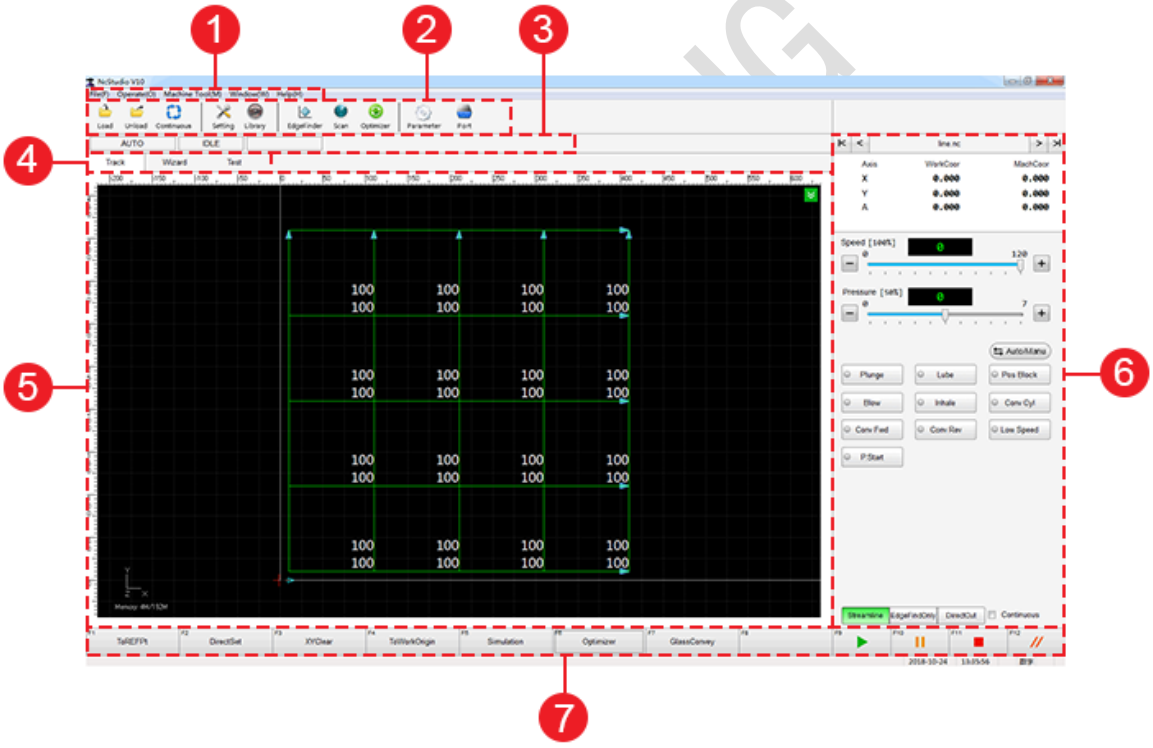
- NcStudio
- NcEditor

After the system is started, two software interfaces pop up:

- NcStudio interface: Used to control machining process, like starting the system, stopping the system, loading file, etc.
- NcEditor interface: Used to edit tool path and post-process CAM.

3.1 NcStudio Interface

The interface of NcStudio is as follows:



1. Menu bar
2. Tool bar
3. CNC info bar

   It includes three kinds of CNC information:

   - Operation mode
   - Status
   - Status info

4. Window switch buttons

   It is used to switch among **Track** window, **Wizard** window and **Test** window.

5. Function window

   It includes three kinds of function windows:

   - Track
   - Wizard
   - Test

6. Machine control area

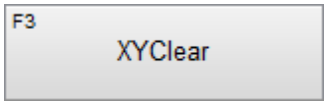
   It shows the information of the machine, including coordinate, parameters, auto mode, manual mode, port, etc.

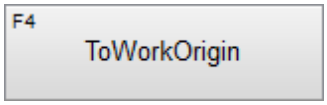
7. Buttons for common operations

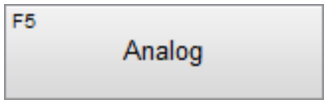
   It includes buttons for eleven common operations:

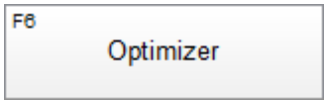
   - 

Return to machine origin
   - 

Set directly
   - 

XY clear
   - 

Homing
   - 

Analog
   - 

Optimize

- 

Convey
- 

Start
- 

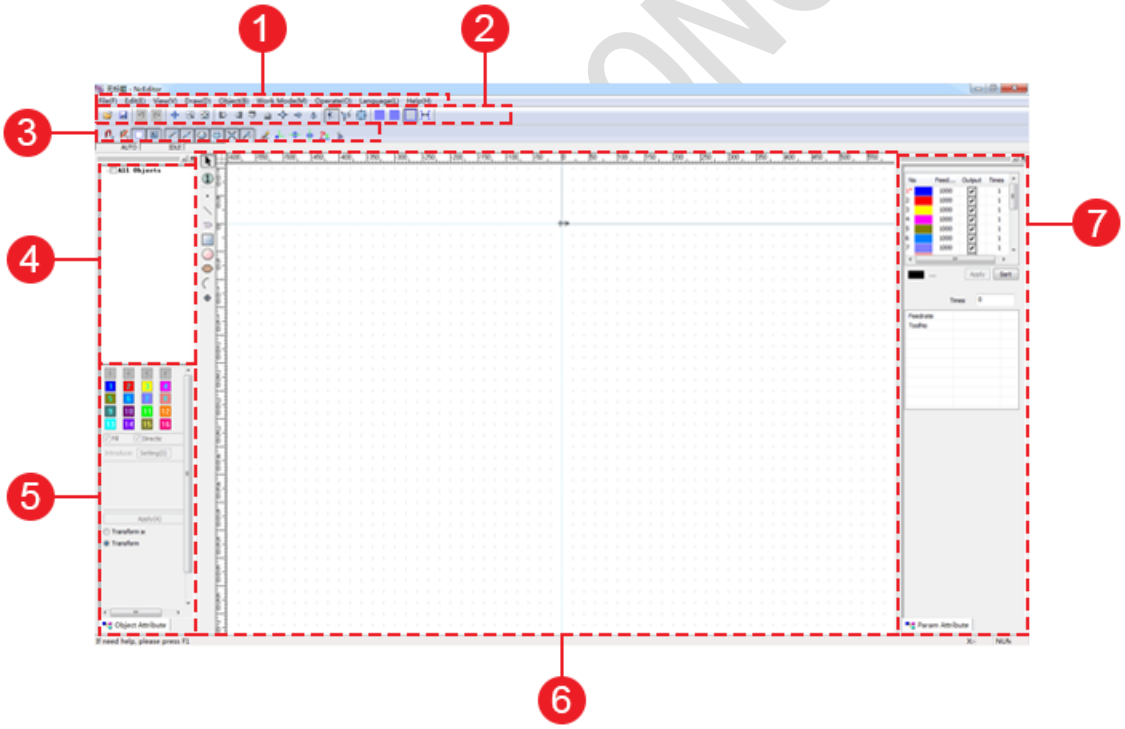
Pause
- 

Stop
- 

Reset

3.2 NcEditor Interface

The interface of NcEditor is as follows:



1. Menu bar
2. Tool bar
3. Drawing toolbar

   The tools in drawing toolbar correspond to instructions in the pull-down list below **Draw** in menu bar.

4. Object list area

   It shows the objects that are displayed in edit window.

5. Object attribute area

   It is used to set attribute of current selected object.

6. Edit window

   It shows target object.

7. Parameter attribute area

   It includes two parts:

   – Color area

   There are 16 colors with different numbers. Each color corresponds to one parameter.
   The objects with the same color shares the same parameters.

   – Parameter attribute area

   After selecting a color, the attribute of the parameter that corresponds to the color will be shown in parameter attribute area.

4 Quick Start

This part introduces NcStudio V10 glass cutting system machining process.

The machining process includes the following:

1. Commission before returning to origin.
2. Return to machine origin.
3. Detect I/O port.
4. Set speed and eccentricity.
5. Import machining file.
6. Set workpiece origin.
7. Execute simulation and analog.
8. Execute auto machining.

4.1 Commission Before Returning to Origin

To commission before returning to origin, do the following:

1. Adjust pulse equivalent.
2. Check axis direction of machine.
3. Set workbench stroke.

4.1.1 Adjust Pulse Equivalent

The smaller pulse equivalent is, the higher the machining accuracy and surface quality will be. While the larger, the faster feedrate will be.

Therefore, lower pulse equivalent should be set under the condition of meeting the demand of feedrate.

To adjust pulse equivalent, do the following:

1. Click **Parameter**. A dialog box named **Parameter** pops up.
2. On **Machine** interface, click **Manufacturer**. Double click parameters **N10010~10013 Axis Pulse Factor**. An input box pops up.
3. Enter the value of each parameter in the input box.
4. Click **F1 OK**.

The adjustment takes effect after the system is restarted.

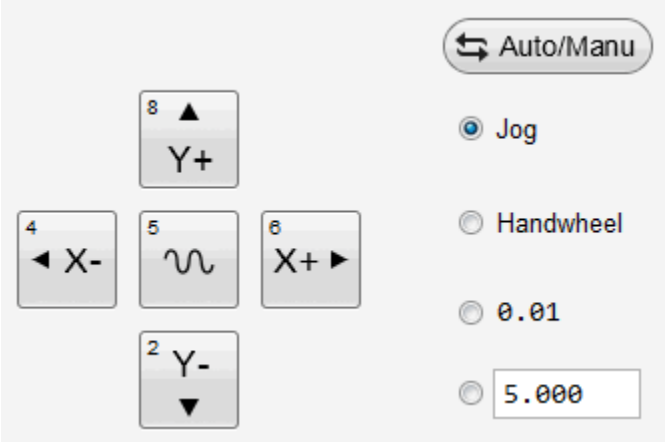
4.1.2 Check Axis Direction of Machine

Checking axis direction of machine is required before homing.

Before checking axis direction of machine, set **Pulse equivalent** parameter to confirm that the direction of the axis is the same as actual situation.

To check axis direction of machine, do the following:

1. Click **Parameter**. A dialog box named **Parameter** pops up.
2. On **Machine** interface, click **Manufacturer**. View the values of parameters **N10000~10003 Axis Direction**.
3. In machine control area, click Auto/Manu to switch to **Manual** mode.



4. To control the machine moves in the direction of the selected axis, do one of the following:
   - Select **Jog** mode:

     Select the axis direction in machine control area or on the keypad.

     If you press the direction button, the machine will move in the direction at jog feedrate until you release the button.

     If you click the direction key 6 X+ / 4 X- / 8 Y+ / 2 Y- and 5 at the same time, the machine will move at rapid jog feedrate.

   - Select **Handwheel** mode:

     Decide axis direction and override on the handwheel, and rotate it. The machine will move in the direction.

- Select step length **0.01** mode or custom step length mode:

  Click the direction key. The machine will move the step length in the direction.

5. Check if the actual direction of axis is the same as the set direction of parameters **N10000~10003 Axis direction** parameter:

   - Same: The axis direction is correct.
   - Opposite: Set the value of parameter **N10000~10003 Axis direction** to the opposite value.

4.1.3 Set Workbench Stroke

To use soft limit function in NcStudio V10, please set workbench stroke according to actual size of the machine.

To set workbench stroke, do the following:

1. Click **Parameter**. A dialog box named **Parameter** pops up.
2. On **Machine** interface, click **Manufacturer**.
3. Double click parameters **N10020~10023 Workbench Lower Limit** and **N10030~10033 Workbench Upper Limit**.
4. Enter the value of each parameter according to actual size of the machine.
5. Click **F1 OK**.

The setting takes effect after the system is restarted.

Note: If it is your first time to set the upper and lower limits of the workbench stroke, confirm the effective machining range of the machine.

4.2 Return to Machine Origin

To confirm the machining position is correct, please execute returning to machine origin.

The following functions can be used after executing returning to machine origin:

- Enable soft limit
- Set fixed point
- Tool change

To execute returning to machine origin, do the following:

1. Click 

, a dialog box named **BKRef** pops up.
2. In the dialog box, do one of the following according to actual situation:
   - Click **All Axis**.

     If it is X1X2 or Y1Y2 configuration, click **Detect Y. REF** before clicking **All Axis**.
   - Click **Set Directly**.

     Set current position as machine origin.
   - Click **X-axis/Y-axis/A-axis**.

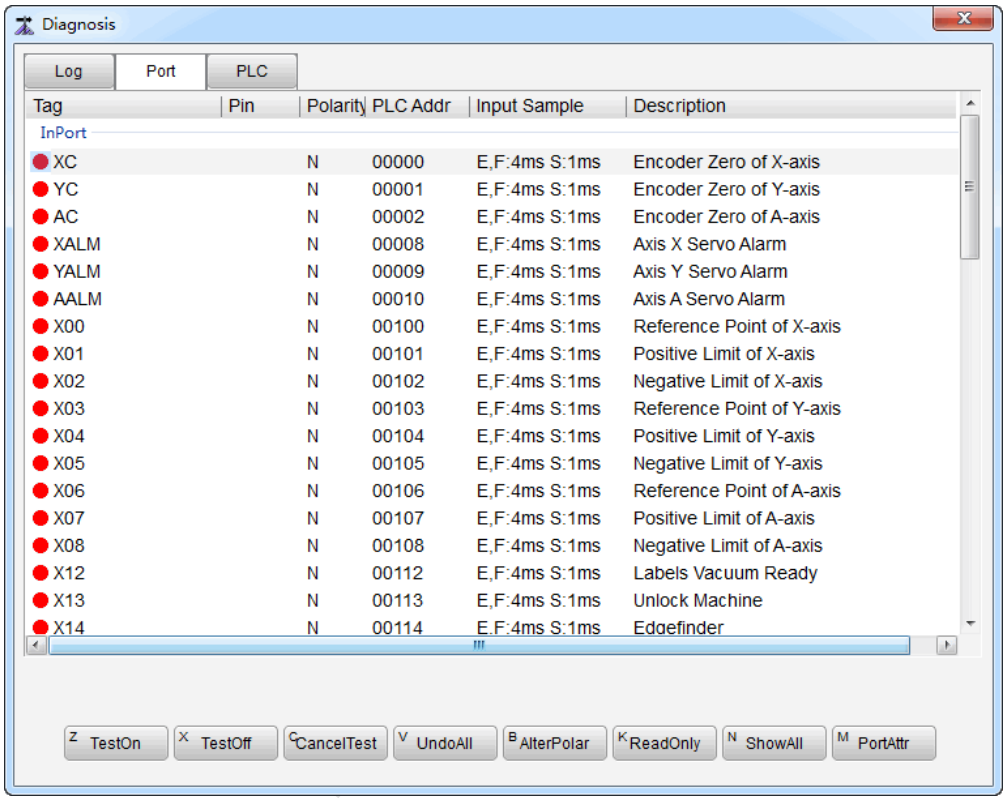
   After returning to machine origin,  appears on operation panel.

4.3 Detect I/O Port

Polarity of I/O port is determined by the type of the switch. For a normally closed switch, the polarity is N (Normally closed) by default; for a normally open switch, the polarity is P (Normally open) by default.

To detect I/O port, do the following:

1. Click **Port**. A dialog box named **Diagnosis** pops up:



   Current status of each I/O port displays in the dialog box. It is helpful for monitoring and troubleshooting.

2. Select the target port, and click **Alter Polar**. A dialog box named **NcStudio** pops up.

3. Click **OK** to restart the software. Modification takes effect.

4. Select the target **Input port**, and click **Port Attribute**. A dialog box named **Port Attributes** pops up.

5. Set **Sampling Interval**, check **Filter** and **Port Enable** and modify **Port Name** and **Port Desc** in the dialog box.

6. Click **OK**.

Note: If the port polarity is wrong, modify it; otherwise, alarm prompt or other failure will occur.

4.4 Set Speed and Eccentricity

Setting speed parameters and eccentricity parameters are required before machining.

4.4.1 Set Speed Parameters

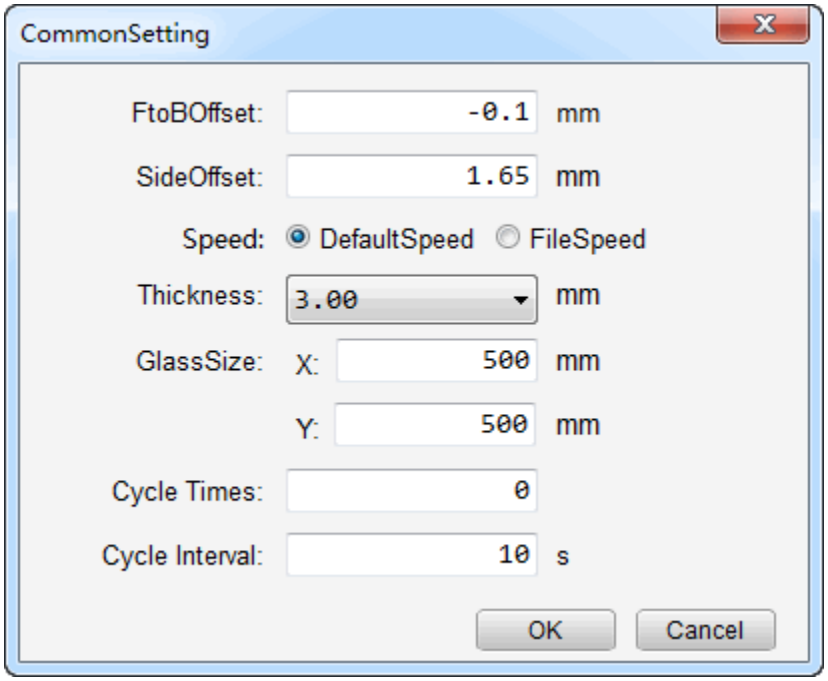
During auto machining, adjust current feed override to control feed speed.

Calculation formula:

Actual value of feed speed = Current feed override * Set value of feed speed

To set speed parameters, do one of the following:

- Click **Setting**. A dialog box name **Common Setting** pops up:



  Select **Speed** according to actual situation.

- Click **Parameter**. A dialog box named **Parameter** pops up.

  Under **Machine** interface, click **Manufacturer**. Double click parameter **N72001 SPEED ASSIGN TYPE**. Enter the value in the input box. Click **F1 OK**.

  Please refer to Set Parameters for details.

4.4.2 Set Eccentricity Parameters

There exists relative offset distance between cutting tool (work point) and control point. To solve this problem, the distance is shown as fore-and-aft eccentricity parameter, and the system compensates toolpath automatically via eccentricity value.

Side eccentricity refers to X-axis offset of the tool. Fore-and-aft eccentricity refers to Y-axis offset of the tool.

Before setting eccentricity parameters, measurement is required. See following figures for eight kinds of eccentricities:

<table>
<tr>
<td>No fore-and-aft eccentricity</td>
<td>

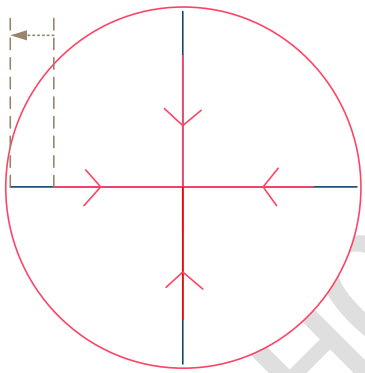



Fig. (1)
As shown in Fig. (1), the side offset exists and its value is negative, while the fore-and-aft eccentricity does not exist.

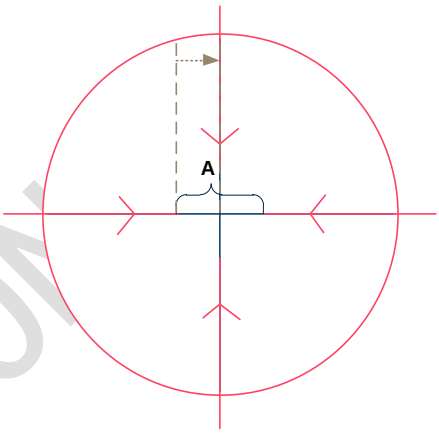


Fig. (2)
As shown in Fig. (2), the side offset exists. Its value is positive and equals ½ the length of line segment A. The fore-and-aft eccentricity does not exist.
</td>
</tr>
<tr>
<td>No side eccentricity</td>
<td>

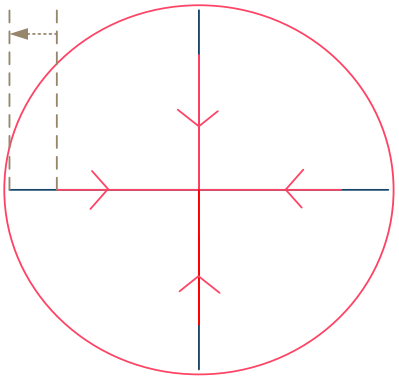



Fig. (1)
As shown in Fig. (1), the side offset exists and its value is negative, while the fore-and-aft eccentricity does not exist.

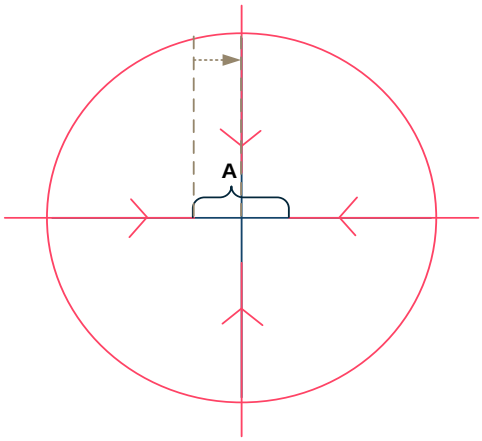


Fig. (2)
As shown in Fig. (2), the side offset exists. Its value is positive and equals ½ the length of line segment A. The fore-and-aft eccentricity does not exist.
</td>
</tr>
</table>

Both fore-and-aft eccentricity and side eccentricity exist

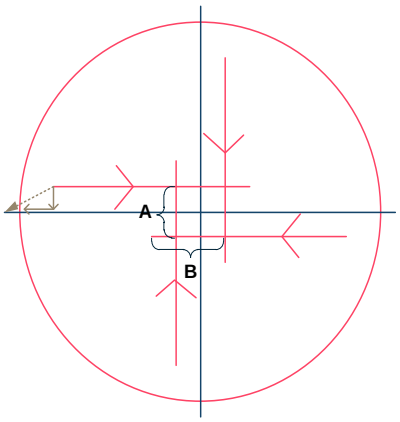


Fig. (5)

As shown in Fig. (5), both the side offset and fore-and-aft eccentricity exist, and their values are both negative. The value of fore-and-aft eccentricity equals the length of line segment A. The value of the side offset equals the length of line segment B minus the length of line segment A.

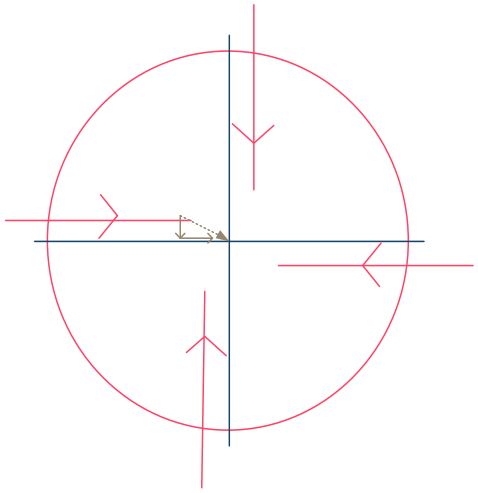


Fig. (6)

As shown in Fig. (6), both the side offset and fore-and-aft eccentricity exist. The value of side offset is positive and the value of fore-and-alt eccentricity is negative.

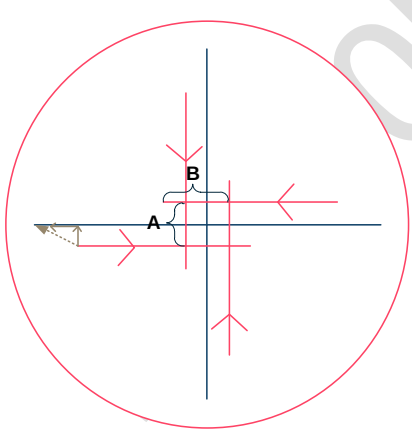


Fig. (7)

As shown in Fig. (7), both the side offset and fore-and-aft eccentricity exist. The value of side offset is negative while the value of fore-and-aft eccentricity is positive. The value of fore-and-aft eccentricity equals ½ the length of line segment A. The value of the side offset equals the length of line segment B minus the length of line A.

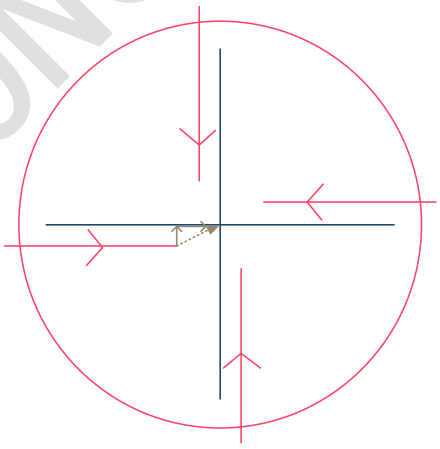
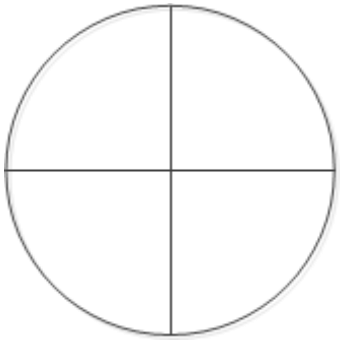


Fig. (8)

As shown in Fig. (8), both the side offset and fore-and-aft eccentricity exist, and their values are both positive.

To set eccentricity parameters, do the following:

1. Click **Window** → **NcEditor** to switch to **NcEditor**. Draw the following graphic:



2. Click **File** → **Save** to save the graphic.

3. In **NcStudio**, click 

.

   The graphic and the machining track will be displayed in **Track** window.

4. Drag the ruler on the left and above sides of **Track** window. Decide eccentricity according to the ruler.

5. Click **Setting**. A dialog box named **Common Setting** pops up.

6. Enter eccentricity values in the dialog box.

7. Click **OK**.

4.5 Import Machining File

To import machining file, do one of the following:

- Load machining file in NcStudio.
- Import machining file in NcEditor.
- Draw machining object in NcEditor.

4.5.1 Load Machining File in NcStudio

To load a machining file in NcStudio, do the following:

1. To find the machining file, do one of the following:
   - Click **File** → **Open and Load** in menu bar.
   - Click **Load** in the toolbar.
2. Select and **Open** the target file.

Optional: Convert the file into a NCE file by setting parameter **N65000**.

4.5.2 Import Machining File in NcEditor

To import a machining file in NcEditor, do the following:

1. Click **File** → **Import G Code File**.
2. Select and **Open** the target file.

Optional: The importing file will cover the original one. To save the original one, click **Edit** → **Insert G Code File**.

4.5.3 Draw Machining Object in NcEditor

To draw a machining object in NcEditor, do the following:

1. To select drawing tools, do one of the following:
   - In menu bar, click **Draw**. Select drawing tools in the pull-down list.
   - In drawing toolbar, click drawing tools.
2. Draw the machining object.
3. Click **File** → **Save** to save the graphic.

The file is saved as a NCE file which will be loaded in NcStudio automatically.

4.6 Set Workpiece Origin

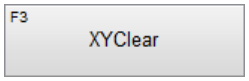
Setting workpiece origin is required before machining.

To set workpiece origin, do one of the following:

- Set in NcStudio.
- Set in NcEditor.

4.6.1 Set Workpiece Origin in NcStudio

To set workpiece origin in NcStudio, do the following:

1. Move each axis of the machine to the target position.
2. Click 

to reset the coordinate of current position.

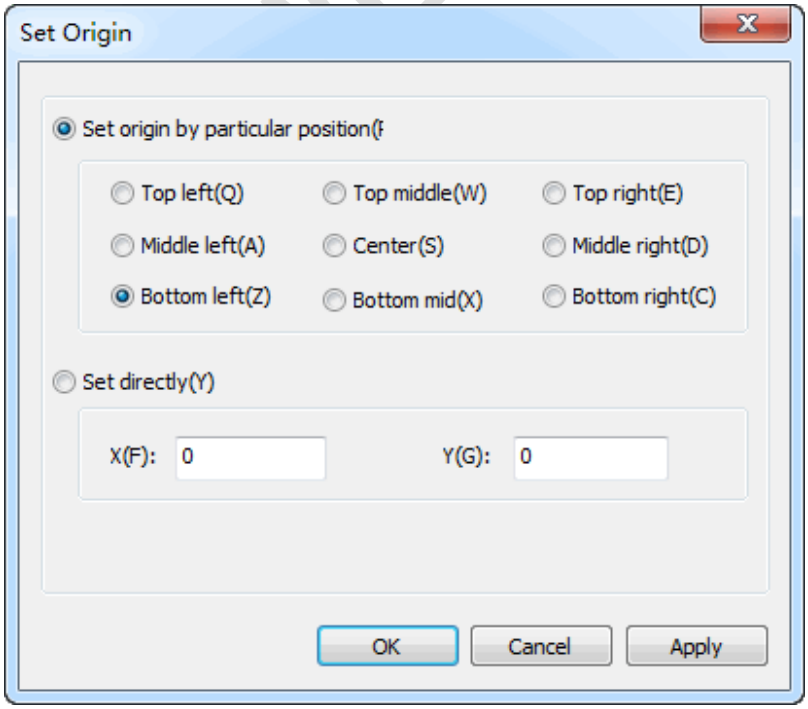
   The current position will be the workpiece origin during machining.

4.6.2 Set Workpiece Origin in NcEditor

To set workpiece origin in NcEditor, do the following:

1. To open **Set Origin** dialog box, do one of the following:
   - In menu bar, click **Draw** → **Set Origin**.
   - In toolbar, click  .

   A dialog box named **Set Origin** pops up:



2. Set in the dialog box as actual situation.

3. Click **OK**.

4.7 Execute Simulation and Analog

After setting workpiece origin, to learn the motion forms of machine in advance to prevent the damage of machine caused by programming errors, execute simulation or analog.

4.7.1 Execute Simulation

In simulation process, the system only shows the machining path of tools in **Track** window and does not drive the machine to take any machining actions.

To execute simulation, do the following:

1. Click **Operate** → **Simulation Mode** in the menu bar.
2. Click 

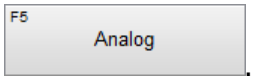
. Simulation starts automatically.
3. Click **Operate** → **Stop Simulation and withdraw the simulation Mode** to stop simulation.

4.7.2 Execute Analog

In analog process, the system will drive the tool to simulate toolpath on the XY two-dimensional surface in a safe height without opening the air cylinder and driving Z-axis.

To execute analog, do the following:

1. Click 

to switch to **Auto** mode.
2. Click 

.
3. Click 

.

In analog process, the air cylinder does not descend, so the glass will not be cut. There are also no port outputs. The machining time in analog is the same as that in normal machining.

4.8 Execute Auto Machining

Auto machining process including four parts:

- Start
- Pause
- Stop
- Breakpoint resume

4.8.1 Start Auto Machining

To start auto machining, do one of the following:

- Click **F9** on the keyboard.
- Click 

.
- Click **Operate** → **Start** in menu bar.

4.8.2 Pause Auto Machining

To pause auto machining, do one of the following:

- Click **F10** on the keyboard.
- Click 

.
- Click **Operate** → **Pause** in menu bar.

4.8.3 Stop Auto Machining

To stop auto machining, do one of the following:

- Click **F11** on the keyboard.
- Click 

.
- Click **Operate** → **Stop** in menu bar.

4.8.4 Use Breakpoint Resume

Prerequisites:

- In emergency situation, like power interruption, E-stop, etc.
- The workpiece coordinate is correct.

Step:

- Click **Shift+F9** on the keyboard.
- Click **Operate** → **Breakpoint Resume** in menu bar.

5 Common Operations

This chapter mainly introduces the following common operations of NcStudio V10:

- Set parameters
- Latch
- Edge finding
- Scan edge
- Wizard
- Label
- Continuous machining
- Set absolute value

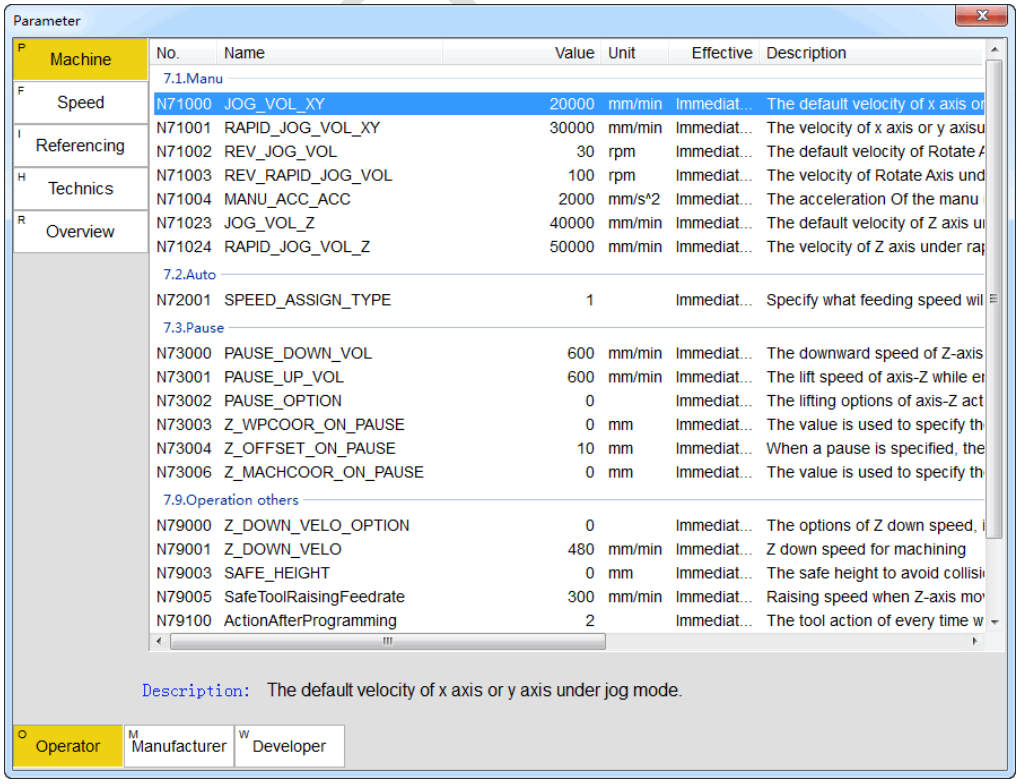
5.1 Set Parameters

Parameters are divided into three kinds:

- Operator parameters: to view operator parameters, click **Operator**.
- Manufacturer parameters: to view manufacturer parameters, enter password.
- Developer parameters: only WEIHONG personnel have permission to adjust and modify developer parameters.

To set parameters, do the following:

1. Click **Parameter** in tool bar. A dialog box named **Parameter** pops up:



2. Click **Operator/Manufacturer/Developer** to switch to corresponding page.
3. Double click target parameter.
4. Enter the value. Click **OK**.

Pay attention to **Effective** time:

- If it is **Immediately**, setting above takes effect immediately.
- If it is **After restarting**, setting above takes effect after the software is restarted.
- If it is **After reloading**, setting above takes effect after the program is reloaded.

5.2 Commission Encoder Latch

Encoder latch is used to adjust functions of returning to machine origin, edge finding and scan edge. It supports servo drives of both bus-type and pulse-type.

Before commissioning, do the following:

- Confirm that the drive is servo drive.
- To adjust returning to machine origin function, confirm that the encoder is incremental encoder.
- Before executing edge finding and scan edge, confirm that each axis has returned to machine origin and been set datum.

5.2.1 Commission Encoder Latch of Bus Type Servo Drive

To commission, do the following:

1. Establish communication among axes. Set station address and drive type.

   Please refer to Set Bus Function for details.

2. Set the following parameters according to actual situation:
   - N10000~10003 Axis direction
   - N11160~11163 Screw pitch
   - N11200~11203 Numerator of mechanical reducer ratio
   - N11200~11213 Denominator of mechanical reducer ratio
   - N16020~16023 Encoder digit
   - N16030~16033 Numerator of electronic gear ratio
   - N16040~16043 Denominator of electronic gear ratio
   - N11001 Encoder type
3. Modify the value of parameter **N11020 Enable encoder locker** to **Yes**.
4. Restart NcStudio software and drive.

After commissioning, returning to machine origin and edge finding will be executed by latch.

5.2.2 Commission Encoder Latch of Non-Bus Type Servo Drive

To commission, do the following:

1. Set drive parameters according to actual situation.

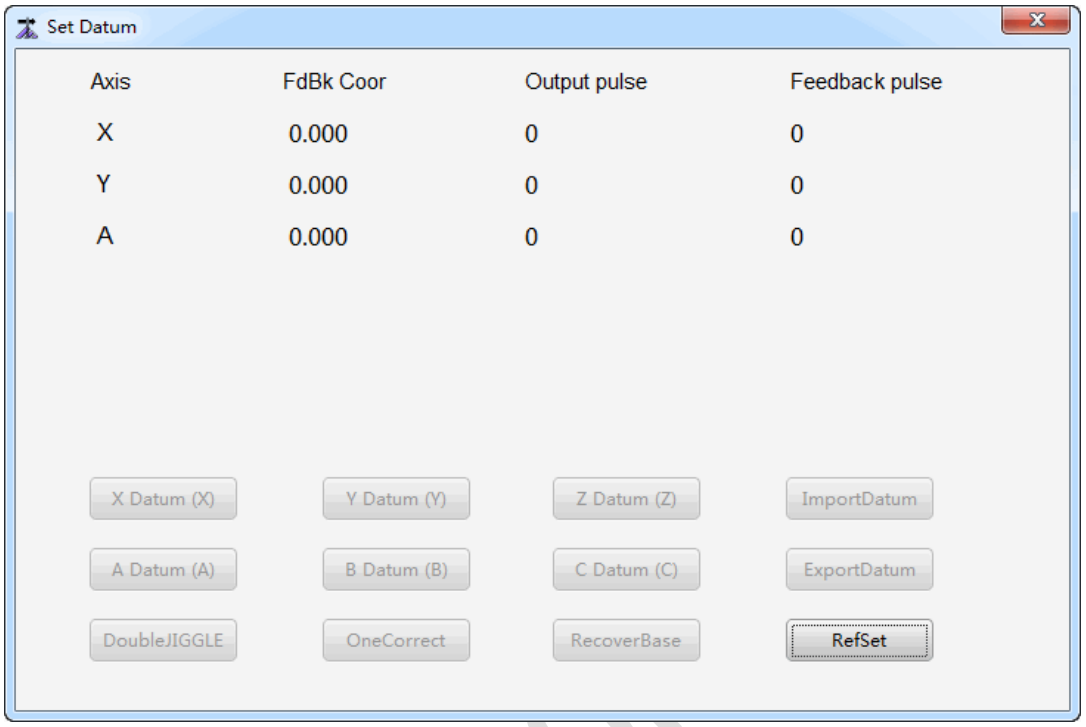
   Taking YASKAWA drive as an example, set parameters in the table below:

Drive parameter	Set value	Software parameter	Rule
PN000	0010(CCW) 0011(CW)	Encoder type	Modify according to set value of drive parameter.
PN002	0100(incremental type)	Axis direction	Modify according to right-handed coordinate rule.
PN200	0005(Pulse equivalent, Pulse direction, Negative logic)	Axis encoder direction	Opposite to axis direction.
PN20E	16777216(Numerator of electronic gear ratio)	Screw pitch	Modify according to the data of axis that corresponds to the machine.
PN210	Pitch/(Pulse equivalent*Mechanical reducer ratio)	Mechanical reducer ratio	Modify according to the data of axis that corresponds to the machine.
PN212	2500	PG frequency dividing ratio	10000; four times PN212 of the drive
PN50A	8100	Motor rotational mode(CCW, CW)	Modify according to the value of PN000 of the drive.
PN50B	6548	Whether the rotational directions of Y1-axis and Y2-axis motors are the same	Modify according to machine structure.

2. Execute returning to machine origin for all axes.

   Please refer to Return to machine origin for details.

3. Click **Machine Tool** → **Set Datum**. A dialog box named **Set Datum** pops up:



4. Enable latch for target axis.
5. Move the axis manually, and check whether machine coordinate and **Feedback coordinate** of the axis are the same.
   - Same: **Feedback coordinate** is correct.
   - Different: **Feedback coordinate** is incorrect. Check whether the parameters are set according to the rule.
6. Set the value of parameter **N11020 Enable encoder locker** as **Yes**.

After commissioning, returning to machine origin, edge finding and scan edge will be executed by latch.

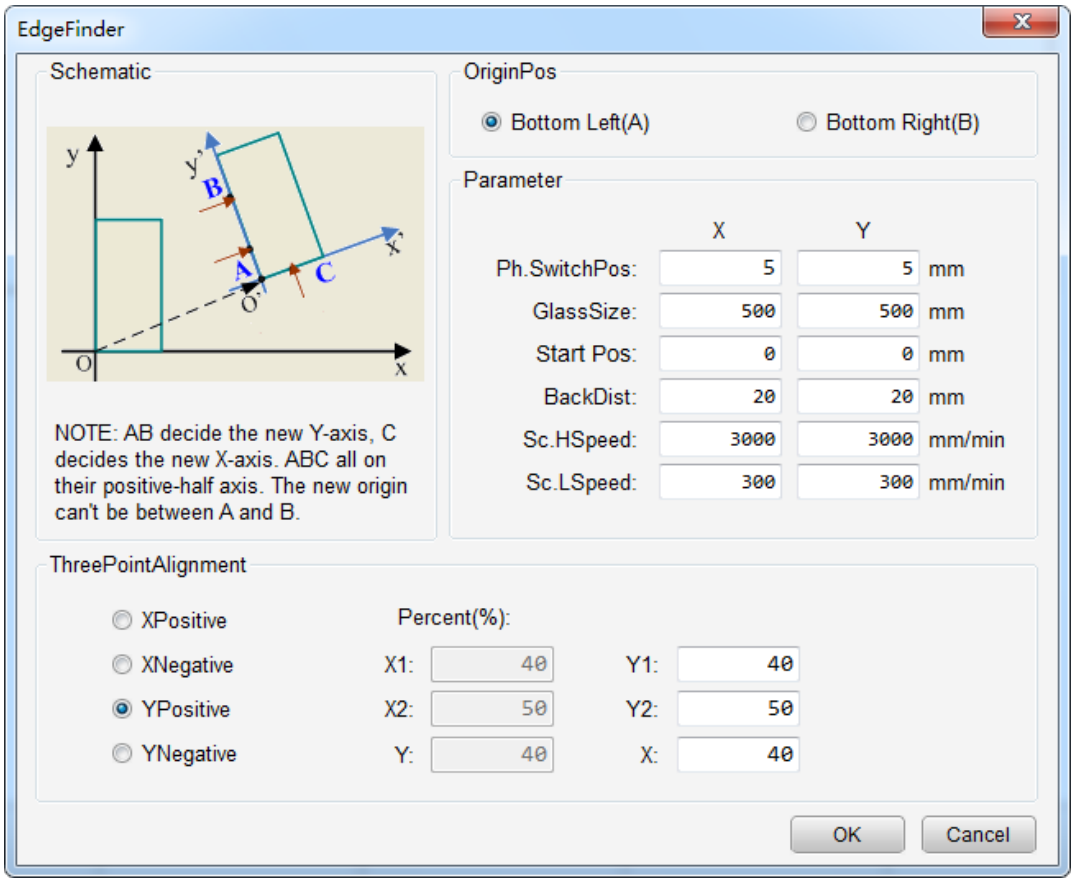
5.3 Execute Edge Finding

Edge finding function can be used in two situations:

- To adjust the coordinate of target graphic when the material is deflected.
- To confirm the position on workbench.

To execute edge finding function, do the following:

1. Click **EdgeFinder** in tool bar. A dialog box named **EdgeFinder** pops up:



   For successful edge finding, confirm the following:

   - The rotation angle between the new workpiece coordinate system and the original one cannot exceed 15°.
   - The new origin cannot be between point A and point B.

2. Enter the value in the dialog box. Click **OK**.
3. Click **Load** in tool bar, and select the toolpath file.
4. Click 

. Edge finding starts.

5.4 Execute Scan Edge

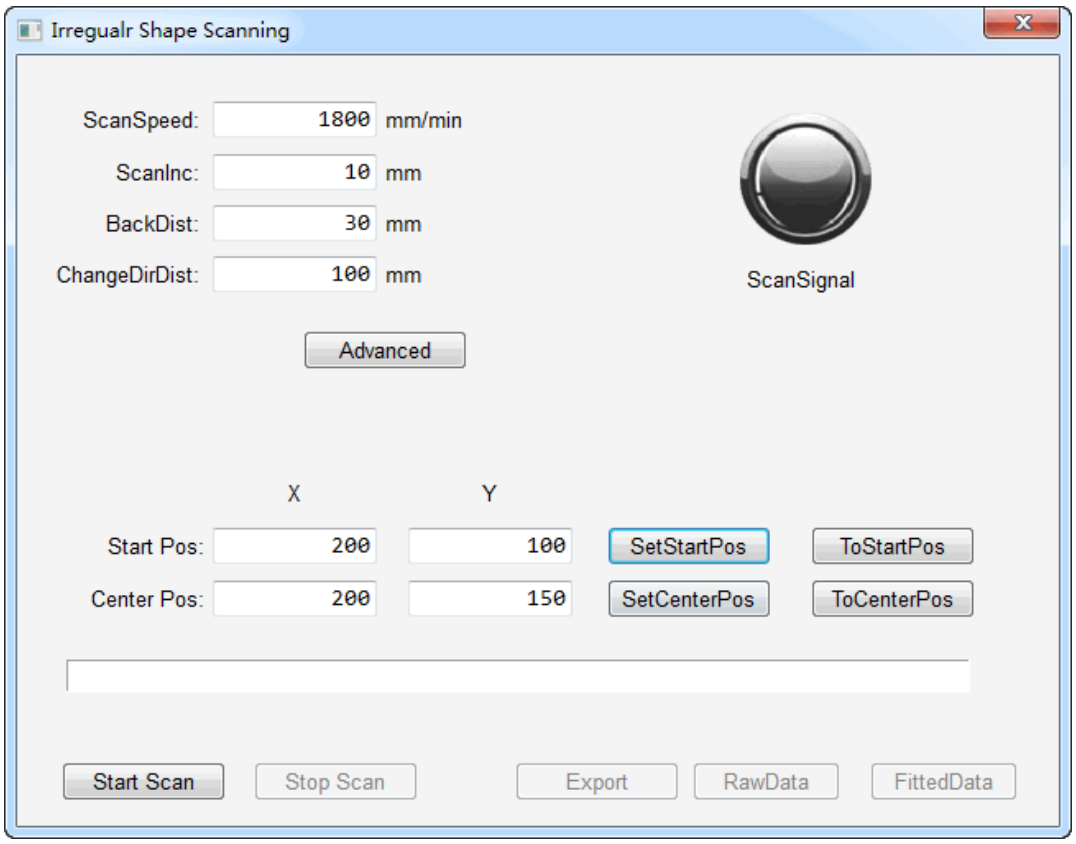
Scan edge is used to scan glass contour with laser head, and generate a DXF toolpath file which will be edited in CAD.
This function is beneficial to improve efficiency and save cost.

There are two kinds of scan edge function:

- Common function
- Advanced function

To execute scan edge function, do the following:

1. Place the workpiece on the machine.
2. Click **Scan** in tool bar. A dialog box named **Irregular Shape Scanning** pops up:



3. **Optional:** Click **Advanced**. Related parameters display in the dialog box.
4. Set **Scan Speed** and **Scan Inc** according to the size of the workpiece.

   To avoid triggering limit during scanning, confirm sufficient space for workpiece.

5. Click **To Center Pos** to check whether current midpoint is proper.
   - Proper: Click **Set Center Pos**.
   - Improper: Move the spindle of the machine to make the laser head shine in the middle of the workpiece. Click **Set Center Pos**.
6. Click **To Start Pos** to check whether current start point is proper.
   - Proper: Click **Set Start Pos**.
   - Improper: Move the spindle of the machine to make the laser head shine on the edge of the workpiece. Click **Set Start Pos**.
7. Click **Start Scan**. Scanning starts.
8. Export scanning result in one of the following ways:
   - **Export**: Output a DXF file in which all points are connected with lines.
   - **Raw Data**: Output a DXF file in which points are fitted to lines and arcs. Adjust by modifying **Cor Index** in **Advance** parameter.
   - **Fitted Data**: Output a DXF file in which all points are in it.

5.5 Use Wizard

There are three functions of wizard:

- Generating three kinds of toolpath files:
  - **Line** cutting
  - **Circle** cutting
  - **Region** cutting
- Using **Command** cutting
- Selecting graphics from **Gallery**

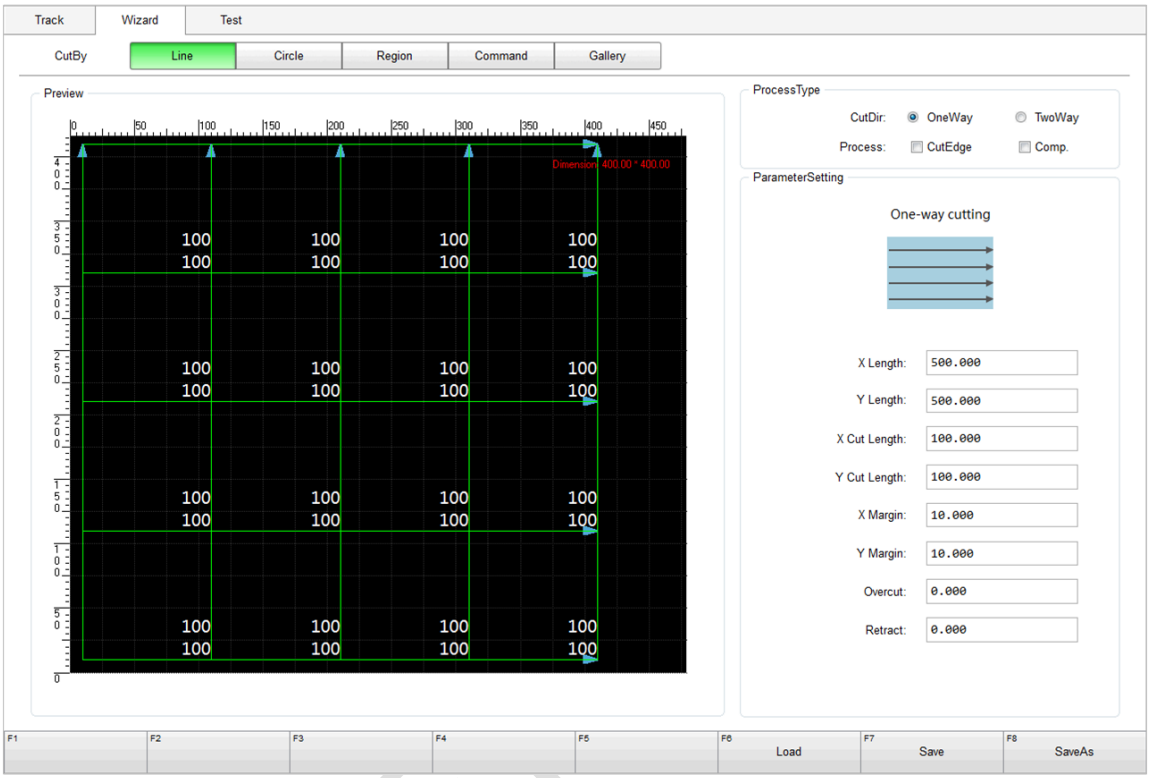
To use wizard function, do the following:

1. Click **Wizard** in function window to switch to wizard interface.
2. Select one of the following cut modes:
   - Line
   - Circle
   - Region
   - Command
   - Gallery
3. Set parameters in **Parameter Setting** area.
4. Click **Load** in tool bar.

5.5.1 Use Line Cut Mode

Line cut mode supports loading file to machine after editing line cutting toolpath.

The following figure is line cut mode wizard interface:

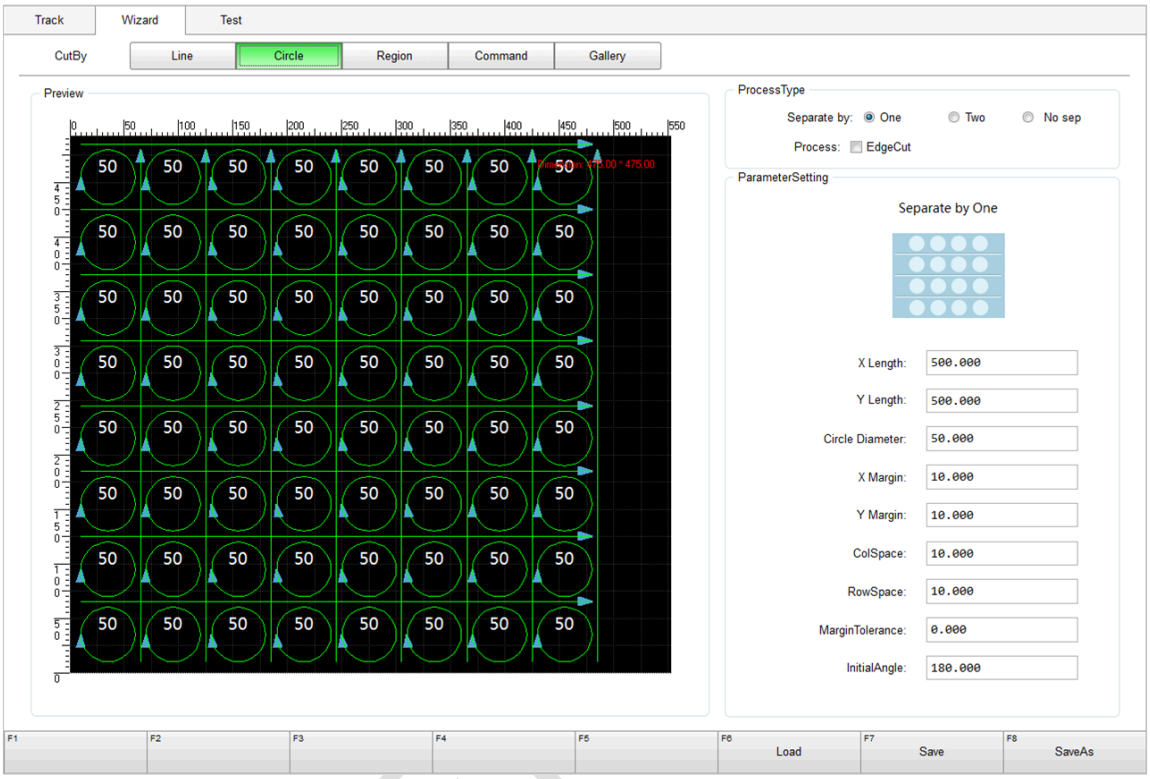


Parameter	Description
X/Y Length	Length of material in X-axis/Y-axis direction.
X/Y Cut Length	Per unit length of material in X-axis/Y-axis direction.
X/Y Margin	Distance between the first edge and workpiece.
Overcut	Cut allowance that exceeds effective distance. Used to decide the end point for tool and avoid damaging glass within effective distance.
Retract	Cut allowance ahead effective distance. Used to decide the start point for tool and avoid damaging glass within effective distance.

5.5.2 Use Circle Cut Mode

Circle cut mode supports loading file to machine after editing circle cutting toolpath.

The following figure is circle cut mode wizard interface:

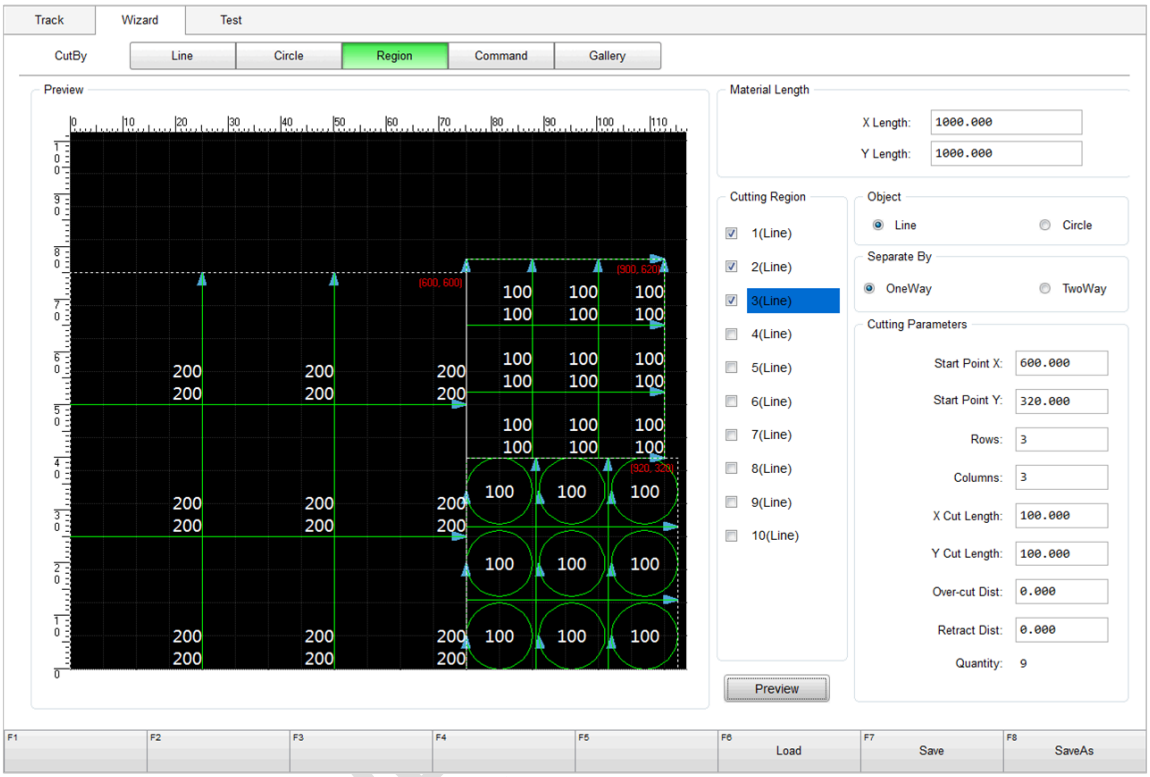


Parameter	Description
Circle Diameter	The diameter of circle that has been cut out from material.
X/Y Margin	Distance between the first edge and workpiece.
ColSpace	Distance between circles in X-axis direction.
RowSpace	Distance between circles in Y-axis direction.
Margin Tolerance	Cut allowance of material in Y-axis direction. X-axis single cutting line will be generated only when the distance between the edge of the last line of circles and workpiece origin.
Initial Angle	Angle offset of plunge position.

5.5.3 Use Region Cut Mode

Region cut mode supports editing different objects in different areas on material and loading to machine it.

The following figure is circle cut mode wizard interface:



Parameter	Description
Start Point X/Y	Start point of tool cutting.
Rows	Set cutting row number.
Columns	Set cutting column number.
X/Y Cut Length	Cut length in X-axis/Y-axis direction.
Over-cut Dist	Cut allowance that exceeds effective distance. Used to decide the end point for tool and avoid damaging glass within effective distance.
Retract Dist	Cut allowance ahead effective distance. Used to decide the start point for tool and avoid damaging glass within effective distance.

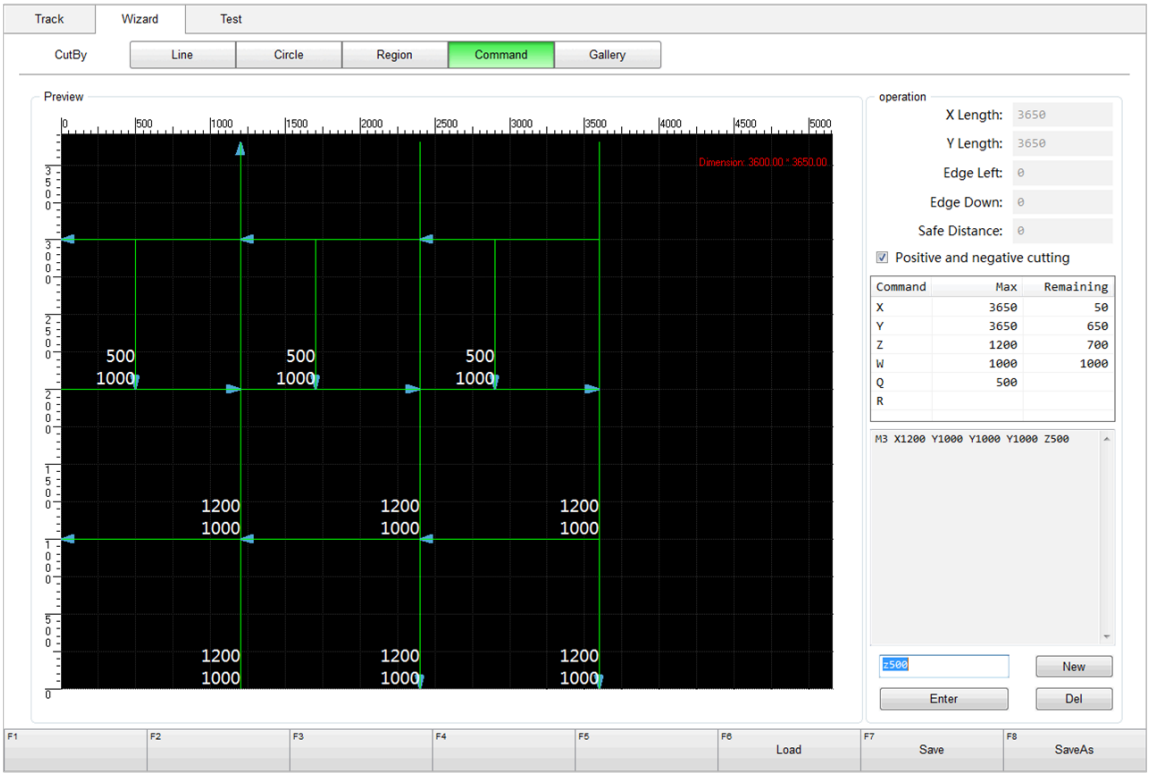
5.5.4 Use Command Cut Mode

Command cut mode assists users to generate machining toolpath.

Following commands are supported by NcStudio V10 software:

- Line cut command X, Y, Z, W, Q and R
- Multiplier command M
- Diagonal cutting command A, B, C, D

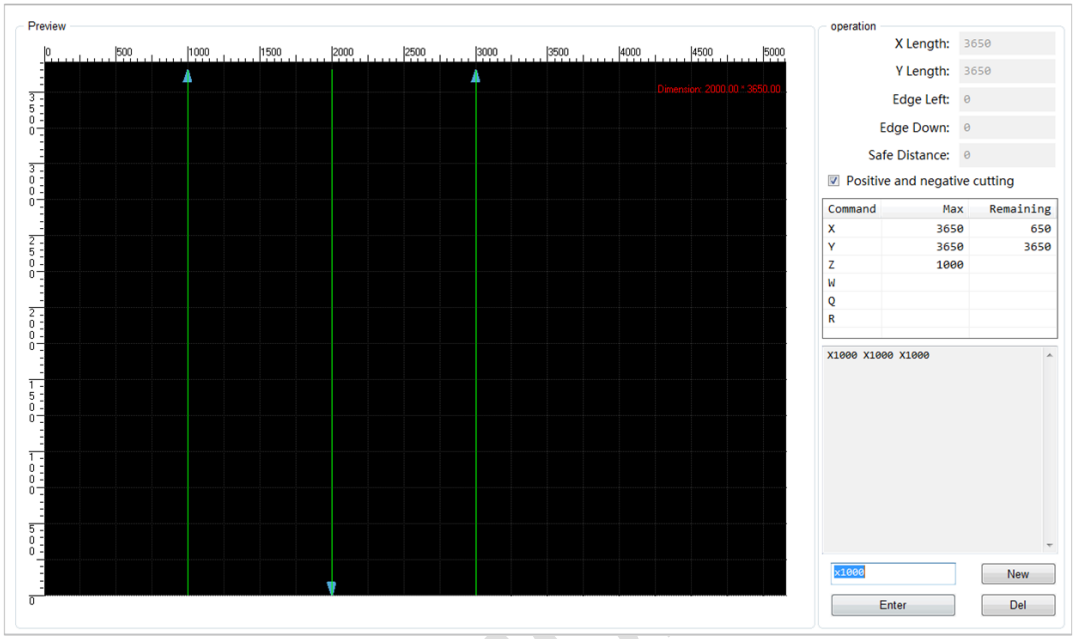
The following figure is circle cut mode wizard interface:



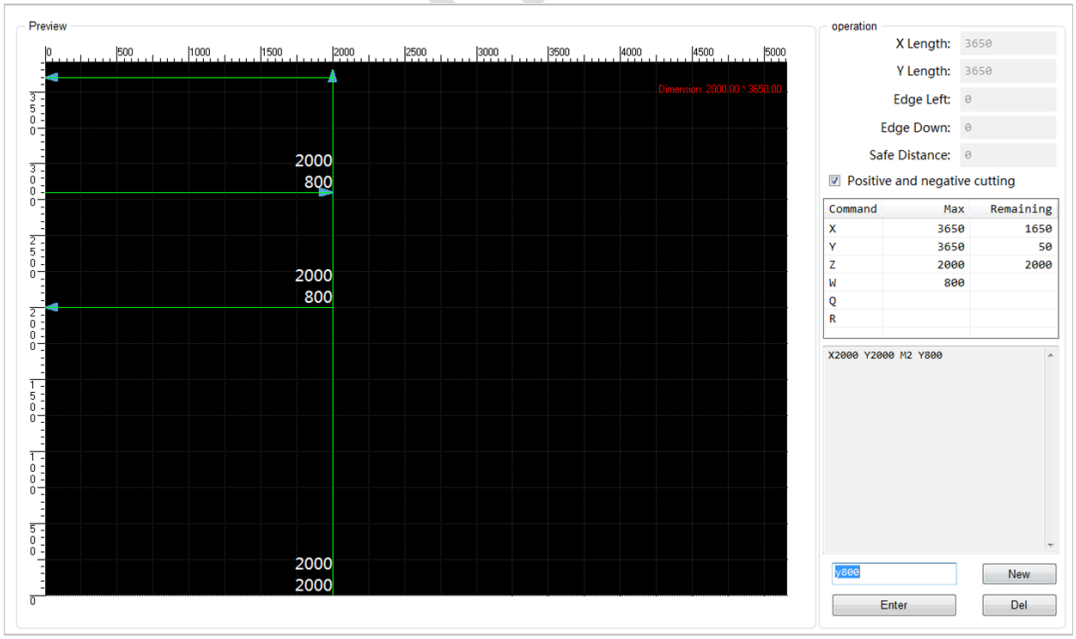
Parameter	Description
X/Y Length	Length of material in X-axis/Y-axis direction.
Edge Left	Reserved scrap area in the left of material.
Edge Down	Reserved scrap area at the bottom of material.
Safe Distance	Vertical distance between tool head and glass. Used to protect tool from plunging and collision during cutting.

Description of commands:

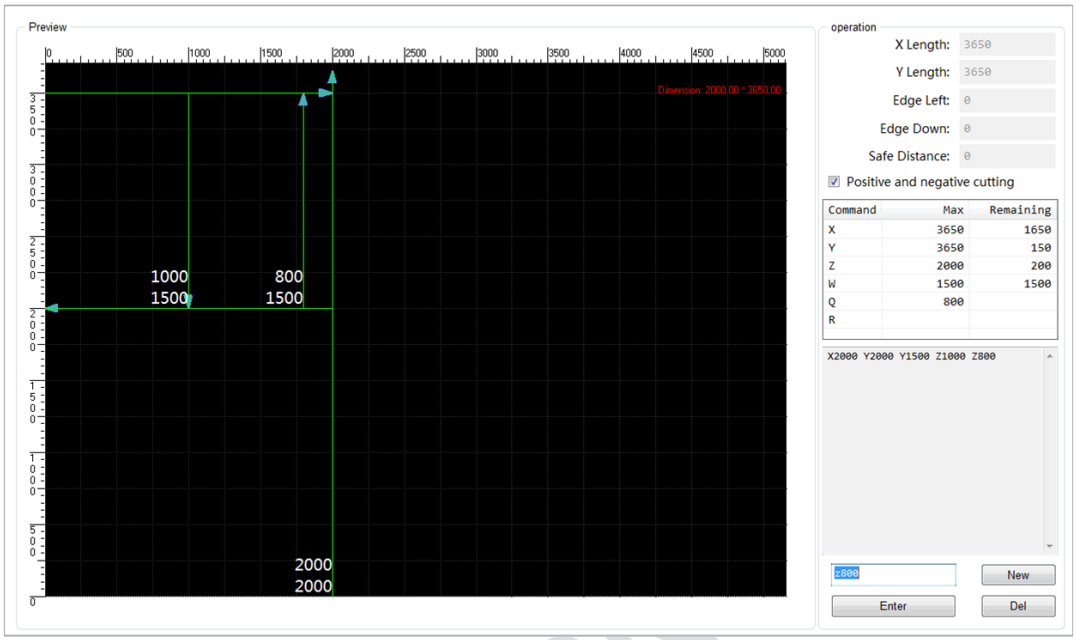
- X cut command: cut from the bottom edge of the material to the top edge vertically. Measure from the origin edge or last X cut line.



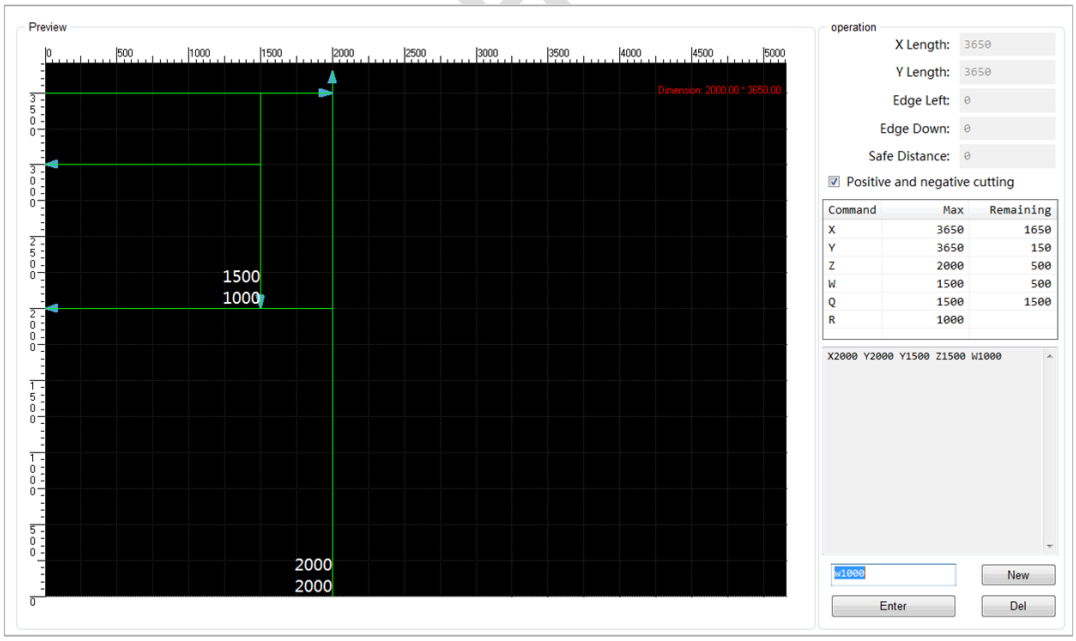
- Y cut command: cut lengthways. Measure from the origin edge and the first X cut line, or between two X cut lines.



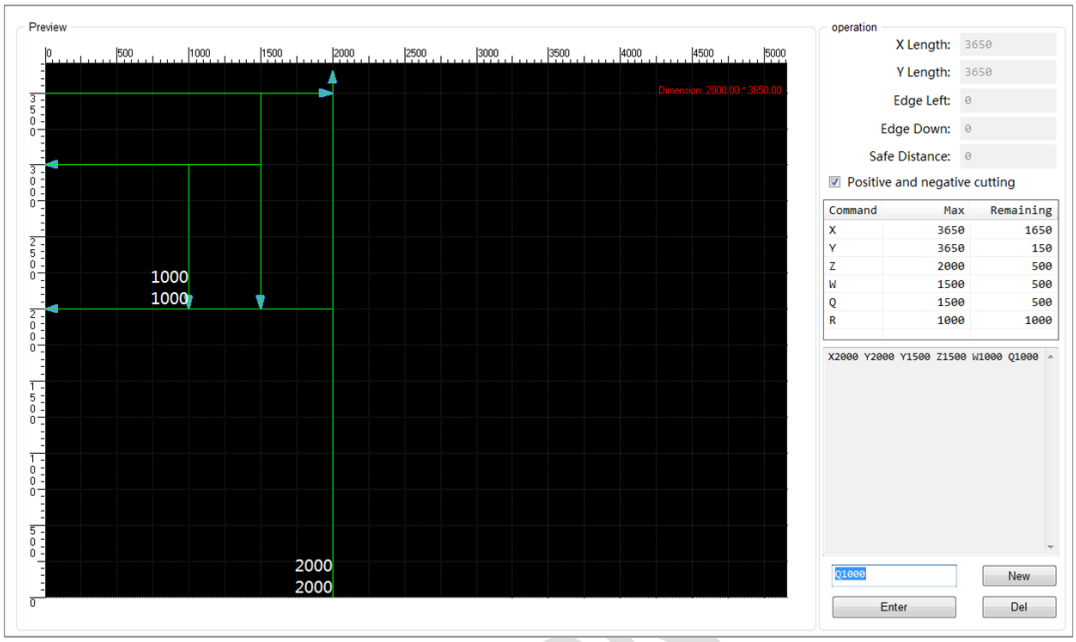
- Z cut command: cut vertically. Measure from the bottom edge and the first tool Y cut line of second material, or between two Y cut lines.



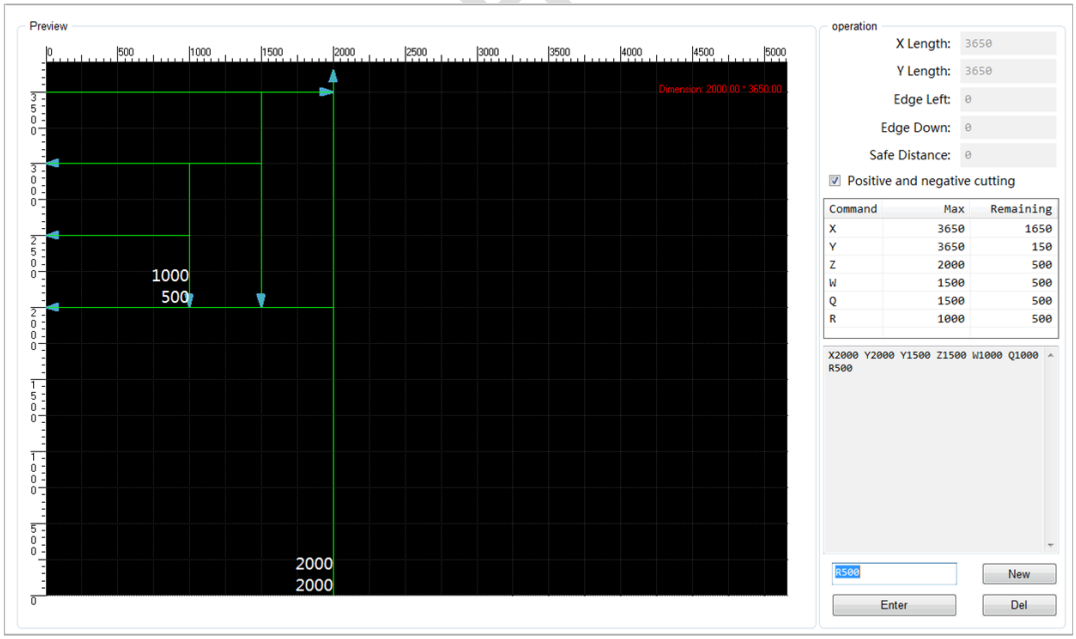
- W command: cut lengthways. Measure from the origin edge and the first tool W line cut, or between two W cut lines.



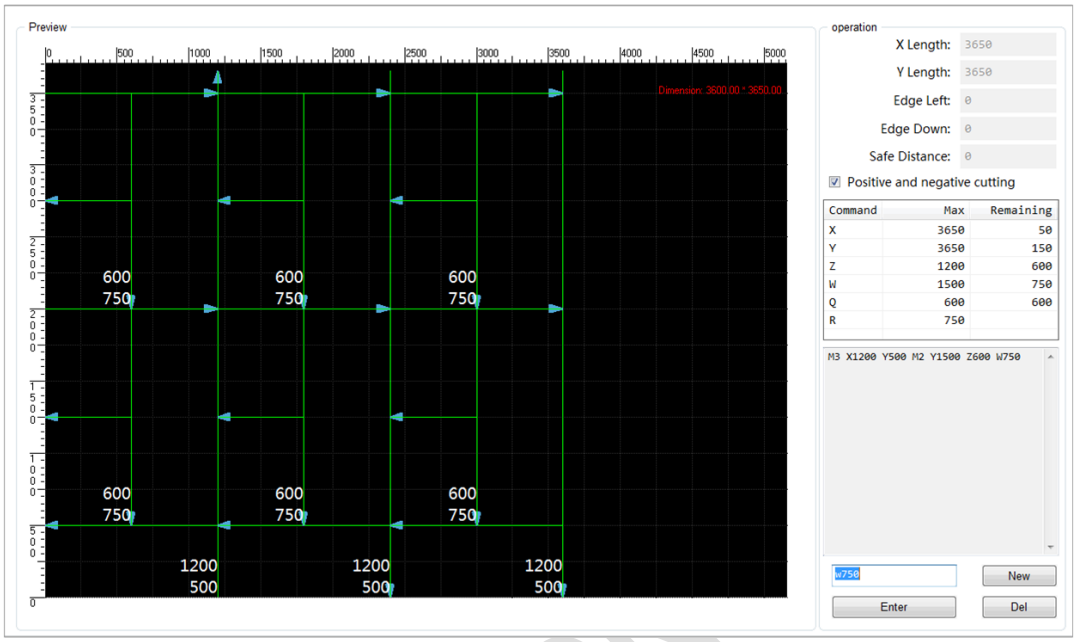
- Q command: cut vertically. Measure from the origin edge and the first tool W line cut, or between two W cut lines.



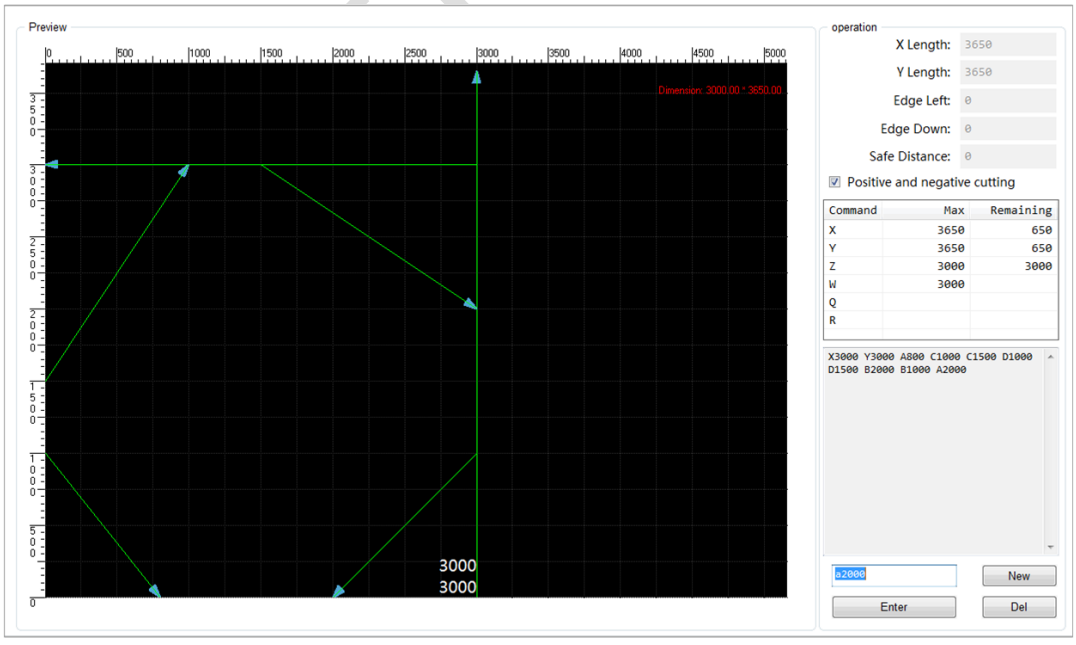
- R command: cut horizontally. Measure from the origin edge and first tool Q cut line, or between two Q cut lines.



- Multiplier command M: if you use multiplier before command X, Y, Z, W, Q, R, the other cut commands will be executed repeatedly.



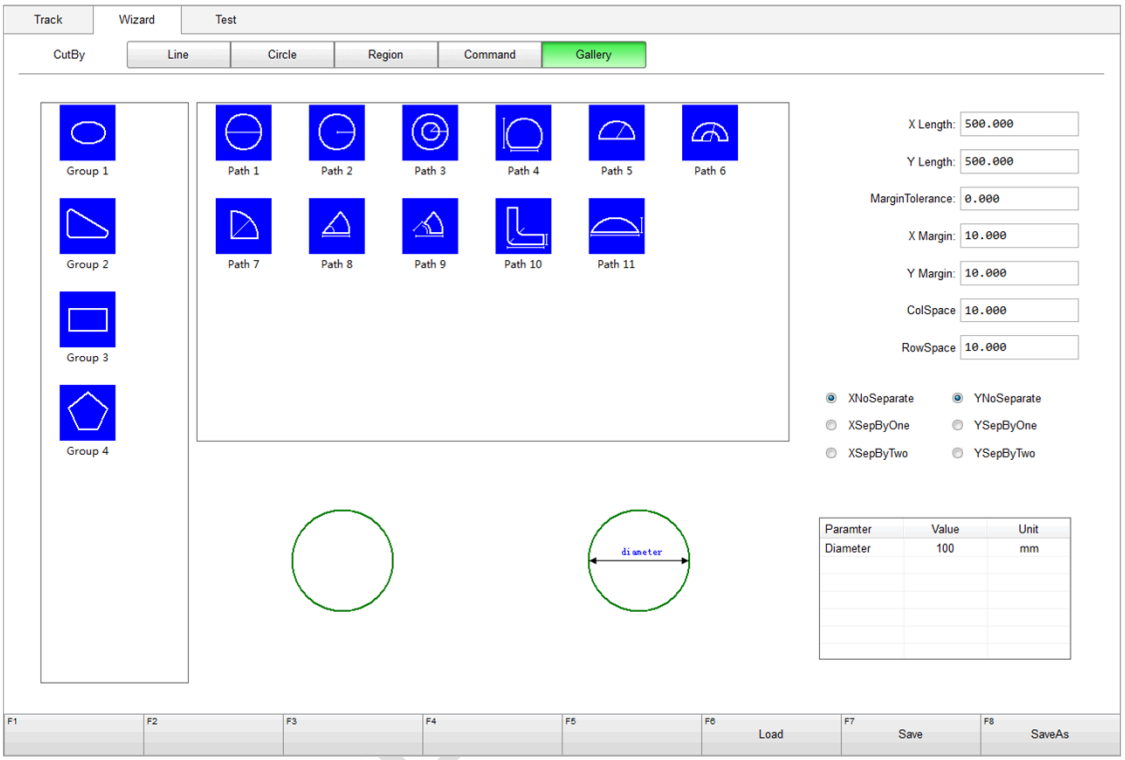
- Diagonal cutting command: diagonal cutting command A, B, C, D should be used in pair. Each command corresponds to an edge of fragment area. To diagonal cutting command A and D, the start point is on the left. While to diagonal cutting command B and C, the start point is at the bottom.



5.5.5 Use Gallery

Gallery function supports setting parameters for graphics, generating machining toolpath file and loading to machine it.

The following figure is circle cut mode wizard interface:



To know more about **X/Y Length**, **Margin Tolerance**, **X/Y Margin**, **ColSpace** and **Row Space**, please refer to Use Line Cut Mode or Use Circle Cut Mode.

Parameter	Description
X/Y No Separate	No split line between each two lines in X/Y direction.
X/Y Sep By One	One split line between each two lines in X/Y direction.
X/Y Sep By Two	One split line every two lines in X/Y direction.

5.6 Use Label

Label function supports labeling on each piece of glass.

Before labeling, check whether the printer has been installed and commissioned:

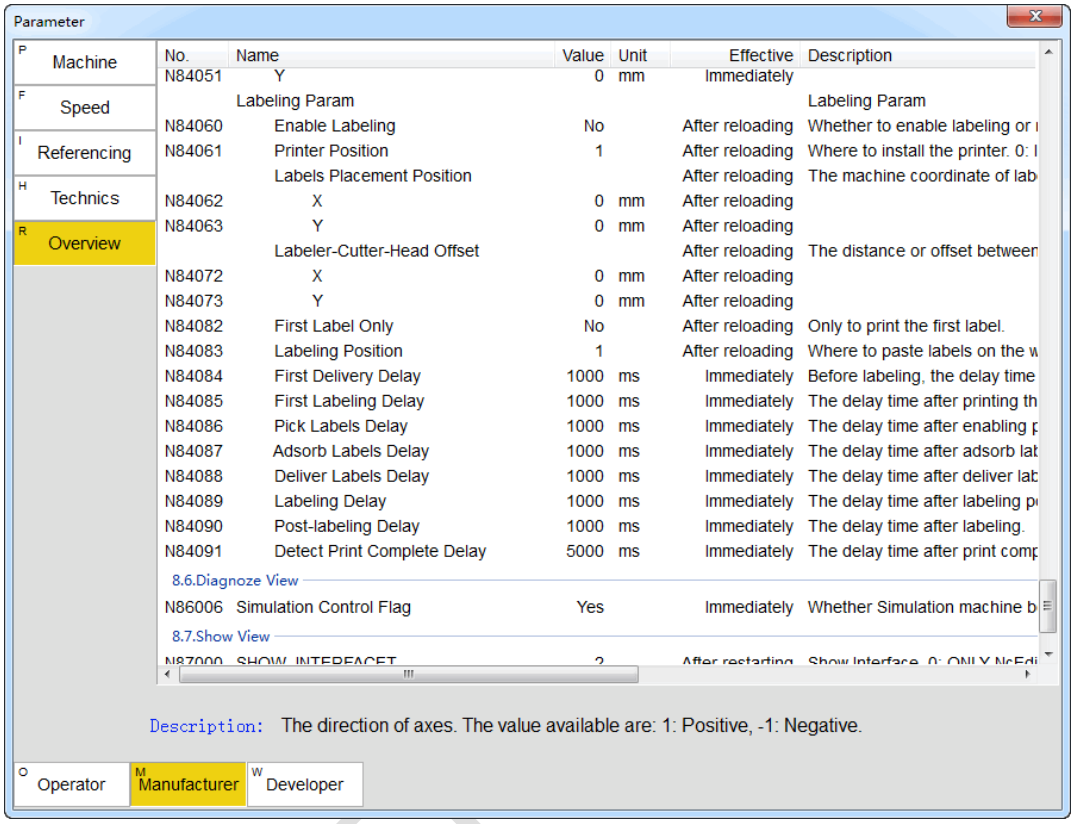
- If the printer has been installed, confirm the following:
  - The machine has been connected with printer (with ethernet port).
  - The printer works well.
  - The printer can be controlled through third party BarTender.exe and PrintLabel.exe.
- If the printer has not been installed, please refer to *Commissioning of NcStudio V10 Glass Cutting Label*.

To use label function, do the following:

1. Click **Parameter**. **Parameter** dialog box pops up.
2. In the dialog box, click **Technics** to switch to **Technics** interface.
3. Click **Manufacturer** in the dialog box.
4. Set the following parameters according to actual situation:
   - N84061 Printer Position
   - N84062~N84063 Labels Placement Position
   - N84072~N84073 Labeler-Cutter-Head Offset
   - N84083 Labeling Position
5. Set the value of **N84060 Enable Labeling** as **Yes**.

6. **Optional:** Set other parameters according to actual situation.

   The following figure shows label parameters:



7. Click **Load** to load a file with label information.

8. Click 

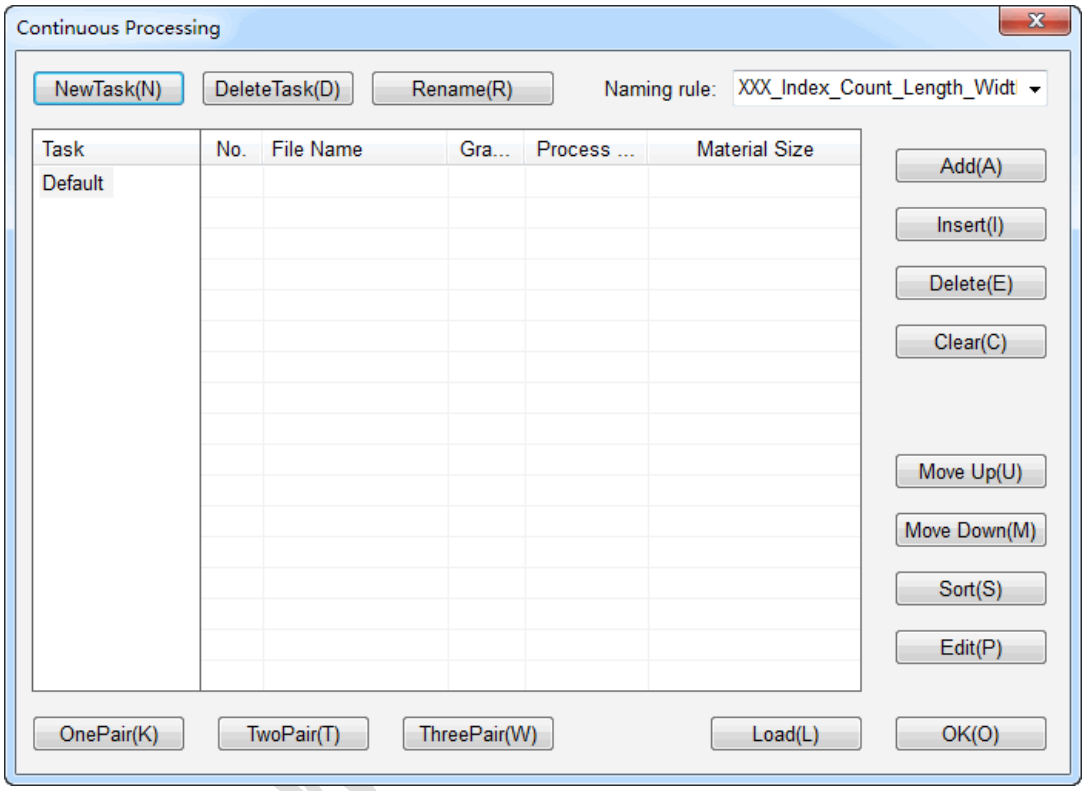
. Labeling starts.

5.7 Continuously Machine

For batch operation, continuous machining is a good choice.

To machine continuously, do the following:

1. Click **Continuous** in tool bar. A dialog box named **Continuous Processing** pops up:



2. Select **Naming rule** in the dialog box.

   The system provides four naming rules.

   - If the file name is one of them, the system records and displays its related information.
   - If the file name is not one of them, click **Edit** to set its machining parameters information.

3. Click **Add**. Select the file.
4. Double click **Process Times** value. A dialog box pops up. Fill in the information according to actual situation.
5. Click **Load**.
6. Select **Streamline/EdgeFindOnly/DirectCut**.

7. Click **Parameter**. A dialog box pops up.
8. Click **Overview** and **Manufacturer**. Set the following parameters:
   - **EdgeFindOnly/DirectCut**:
     - N84012 Edge Finder Position Box Delay
     - N84045 Enable Position Box
   - **Streamline**:
     - N84000 Up Cylinder Delay
     - N84012 Edge Finder Position Box Delay
     - N84045 Enable Position Box
     - N84050 X
     - N84051 Y
9. Click 

to start machining.
10. Click input port **Ejection**.

    In online status, the system will transfer and load the glass that has been cut, and automatically machine the next toolpath.

5.8 Set Absolute Value

The servo system of absolute encoder supports recording machine origin position. Through this function, set machine origin once during commissioning. When it is used in practice, returning to machine origin is not required.

To use absolute function, a servo motor with absolute encoder is required.

To set absolute value, do the following:

1. Click **Machine Tool**. A dialog box pops up.
2. Set datum.
   - If the datum has been set before, do the following:
     1. Click **OneCorrect**.
     2. Click **DoubleJIGGLE**. A dialog box named **DoubleJIGGLE** pops up.
     3. Click **Y BK.REF.PT**.
   - If the datum has not been set before, do the following:
     1. Click **OneCorrect**.
     2. Click **DoubleJIGGLE**. A dialog box named **DoubleJIGGLE** pops up.
     3. Adjust axis according to actual position of Y1Y2 axes.
     4. Click **OK**.

        Compare the verticality between X-axis and Y-axis and the parallelism between Y1-axis and Y2-axis. If it does not meet the requirement, set datum again.
3. Move the target axis to machine origin.
4. Click **RefSet**.
   Password is required.
5. Click the datum button that corresponds to the axis.
6. Click **OneCorrect**.
7. Close the dialog box. Restart the software.

After the software is updated, the datum information will be lost. Restore the software via **Import Datum** and **Export Datum**.

6 Miscellaneous Operations

This chapter mainly introduces the following miscellaneous operations of NcStudio V10:

- Diagnosis
- Screw error compensation
- Register
- NcCloud

6.1 View Diagnosis

Diagnosis information includes the following:

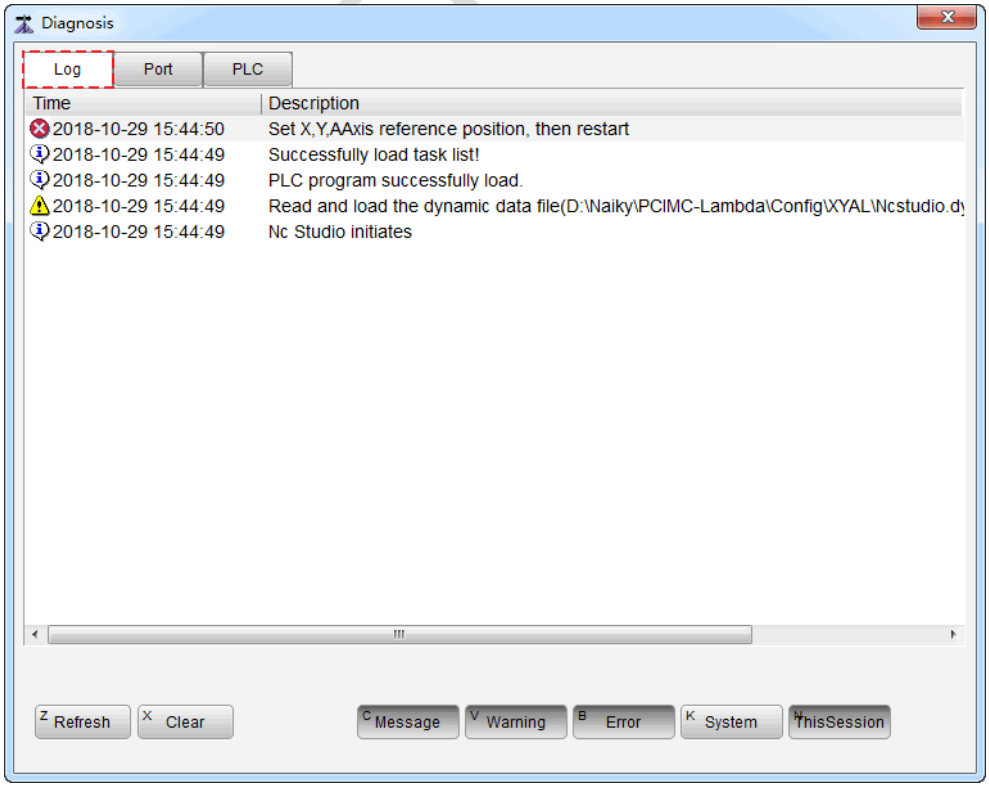
- Log
- Port
- PLC

6.1.1 View Log

Log information includes operating information, workpiece coordinate information, alarm information, etc.

To view log information, do the following:

1. Click **Machine Tool** → **Log**. A dialog box named **Diagnosis** pops up:



2. Click the button at the bottom of the dialog box to view corresponding information.

Please clean system log regularly. Excessive logs slow down the performance and responsiveness of the system.

6.1.2 View Port

Please refer to Detect I/O Port for details.

6.1.3 View PLC

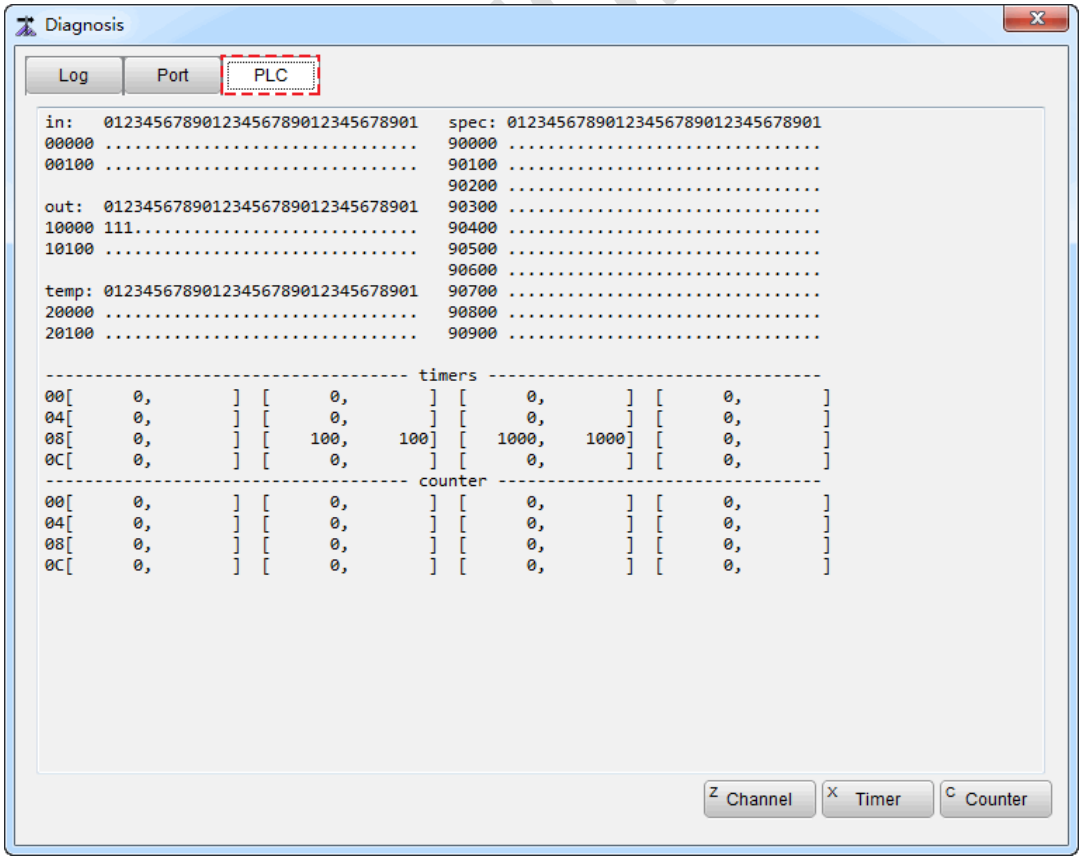
With built-in PLC module of NcStudio software, output port can be controlled by PLC.

PLC information is used in PLC commissioning. Generally, this function is used by debugging personnel. For details, please refer to *NcStudio Programming Manual*.

To view PLC information, do the following:

1. Click **Machine Tool** → **Log**. A dialog box named **Diagnosis** pops up.

2. Click **PLC** to switch to **PLC** interface:



3. Double click the buttons at the bottom of the dialog box, and enter **Channel**, **Timer** and **Counter**.

4. Click **OK**.

6.2 Compensate Screw Error

Screw error compensation includes pitch error compensation and backlash compensation.

6.2.1 Compensate Backlash

The spindle is fixed on the screw. There exists difference between screw external thread and internal thread. When spindle moves in one direction, it must go through the error between threads. Such error compensation is called backlash compensation.

To compensate backlash, do the following:

1. Measure screw error with dedicated gauge.

2. Fix the gauge on spindle. And rotate the meter pointer to origin.

3. Manually move the meter pointer by **a** millimeters, and move the pointer in the reverse direction by **a** millimeters. If the meter pointer moves by **b** millimeters actually, then the backlash is **(a-b)** millimeters.

4. Click **Parameter**. A dialog box pops up.

5. Click **Manufacturer**. Set value of parameter **N12010~12013 Backlash** that corresponds to the axis to (a-b).

6. Restart the software.

6.2.2 Compensate Pitch Error

Calculation formula:

Error = Actual machine coordinate - Nominal machine coordinate

To compensate pitch error, do the following:

1. List the modification value of each point on error curve in a table. And save the table in CNC system memory.

2. Click **Parameter**. A dialog box pops up.

3. Click **Manufacturer**.

4. Set the following parameters:
   - Set the value of parameter **N12000 Enable Leadscrew CMPN** as **Yes**.
   - Set the value of parameter **N12001 Only Backlash CMPN** as **No**.
5. Restart the software.

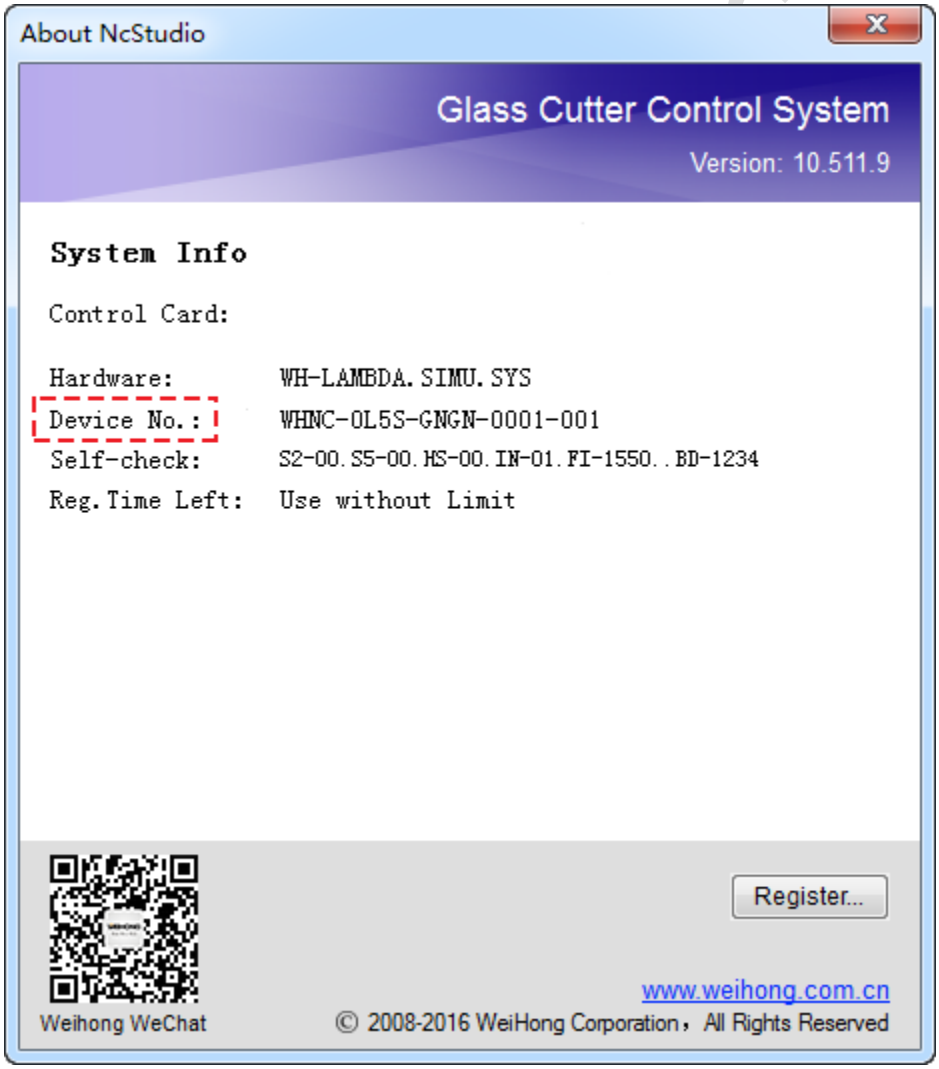
6.3 Register

Registration function is used to register use time of card and drive.

6.3.1 Get a Registration Code

You can get a registration code either in WeChat or via the manufacturer.

Before getting a registration code, click **Help** → **About NcStudio** to get the **Device No.**:



To get a registration code, do one of the following

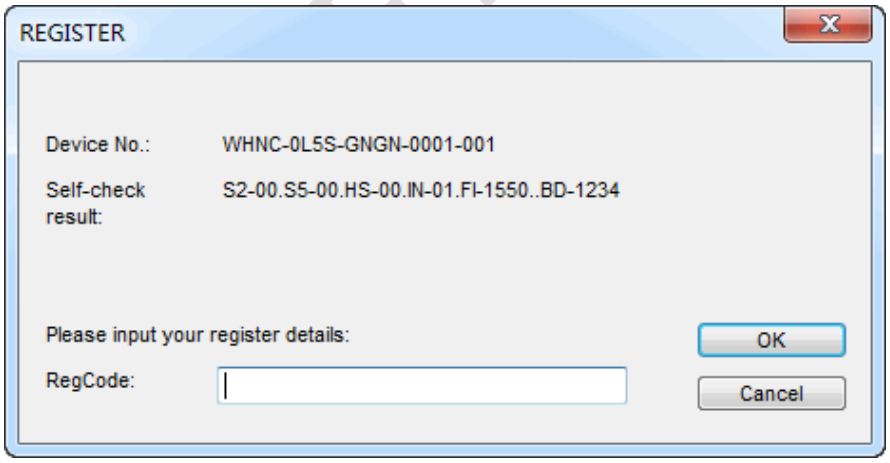
- Do the following to get a registration code in WeChat:
    1. Enter **WEIHONG** official account in WeChat.
    2. Click **Service** → **Register** → **Activate Account** to register.
    3. Return to **Log In** page. Fill in the blanks, and click **Log in** to jump to **Time** page.
    4. On **Time** page, enter related information.
    5. Click **Generate**.
- Do the following to get a registration code via manufacturer:
    1. Send device No. to manufacturer.
    2. Ask the manufacturer to use APP "NcStudio Generator" to generate a registration code.

6.3.2 Register in NcStudio

To register in NcStudio, do the following:

1. Click **Help** → **About NcStudio** → **Register**.

   A dialog box named **REGISTER** pops up:



2. Fill in **Registration code**.
3. Click **OK**.

Registration completes.

If drive registration enabled, card and drive will be registered at the same time.

If exception prompt occurs during registering, please refer to Exception prompt occurs during registering.

6.4 Use NcCloud

NcCloud, independently developed by WEIHONG, is a system of cloud-based monitoring and control.

To use NcCloud, click **Help** → **NcCloud**. **NcCloud Assitant** pops up.

Please refer to *NcCloud Assistant User Manual* for details.

7 Auxiliary Drawing Operations in NcEditor

This chapter mainly focuses on auxiliary drawing operations, including:

- Setting lead line
- Catch option
- Combining objects
- Group
- Array
- Chamfer
- Setting tool path
- Expanding bidirectionally

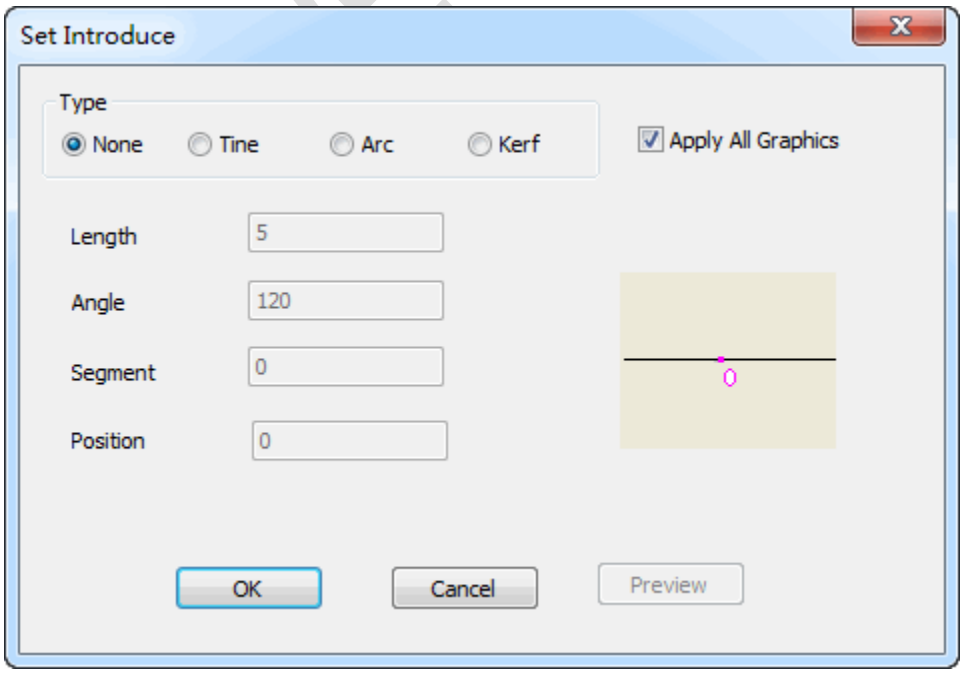
7.1 Set Lead Line

If the object is closed, set lead line in workpiece scrap area of toolpath to avoid machining error and workpiece damage.

To set lead line, do the following:

1. Select the target closed graphic.
2. Do one of the following to call **Set Introduce** dialog box:
   - Right click in edit window and select **Set Introduce**.
   - Click **Object** → **Set Introduce** in menu bar.

   Set Introduce dialog box is as follows:



3. Select lead line type and enter values of related parameters in the dialog box.

4. Click **OK**.

7.2 Catch Option

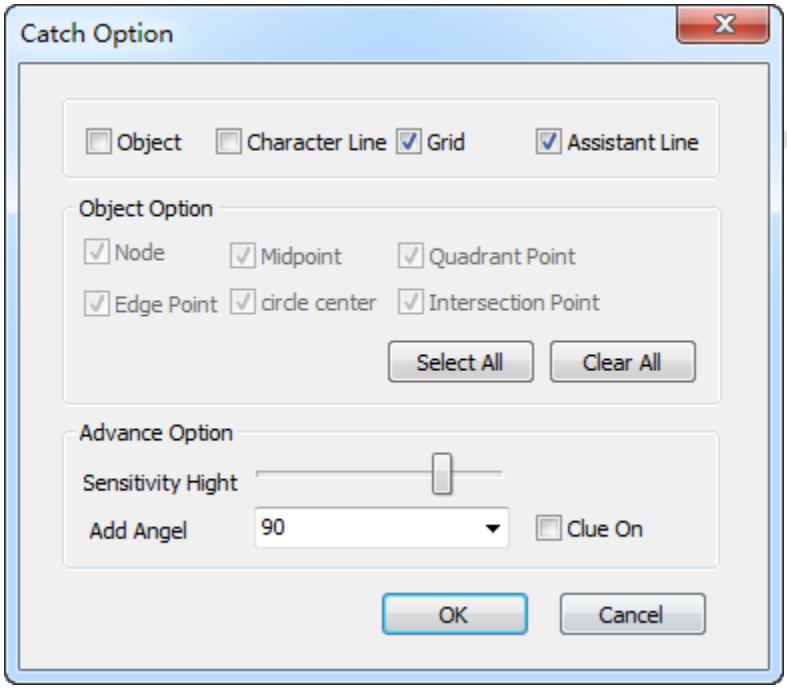
Catch option is used to catch feature points of object in drawing precisely.

To use catch option function, do the following:

1. Do one of the following to open **Catch Option** dialog box:

   - In menu bar, click **View** → **Catch Option**.
   - In tool bar, click .

   Catch Option dialog box is as follows:



2. Check the feature points that you need to catch according to the shape of the graphic.

3. Set **Sensitivity Hight** and **Add Angel**.

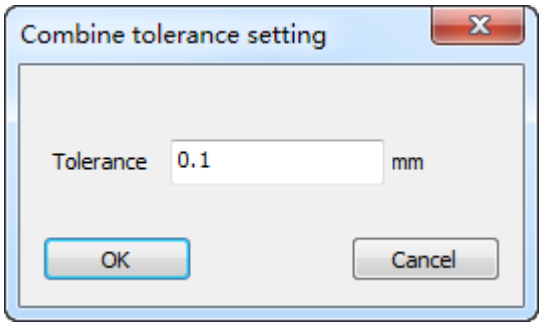
4. Click **OK**.

7.3 Combine Objects

This function is used to combine straight line, arc and open polyline mutually, and improve machining efficiency.

To combine objects, do the following:

1. Click **Object** → **Combine Object Tolerance Setting**. A dialog box named **Combine tolerance setting** pops up:



2. Enter **Tolerance** value according to actual situation.
3. Click **OK**.
4. Select at least two objects whose endpoints are near to each other.
5. Do one of the following to combine the selected objects:
   - Click **Object** → **Combine Object** in menu bar.
   - Right click in edit window and select **Combine Object**.

7.4 Use Group

Group function includes the following:

- Group
- Disbanding group

7.4.1 Group

Group function is used to edit objects integrally.

To use group function, do the following:

1. Do one of the following to select target objects:
   - Check the objects in object list area.
   - Press **Ctrl** in keyboard and click the objects at the same time.
2. Do one of the following to compose the objects:
   - Click **Object** → **Group** in menu bar.
   - Right click in edit window and select **Group**.

7.4.2 Disband Group

Disbanding group function is used to edit part of objects.

To disband group, do the following:

1. Check the target group in object list area.
2. Do one of the following to disband the group:
   - Click **Object** → **Break Group** in menu bar.
   - Right click in edit window and select **Break Group**.

After disbanding, the objects in this group will return to last group.

7.5 Use Array

To improve machining efficiency, use array function to display objects in rectangular or circular array during batch machining.

To use array function, do the following:

1. Select an object or group in object list area.

   To array multiple objects or groups, group first.

2. Click **Object** in menu bar or right click, and select array type:

   - Rect array
   - Circle array

   A dialog box pops up.

3. Set parameters in the dialog box.

4. Click **OK**.

The selected objects will be displayed in rectangular or circular array.

7.6 Use Chamfer

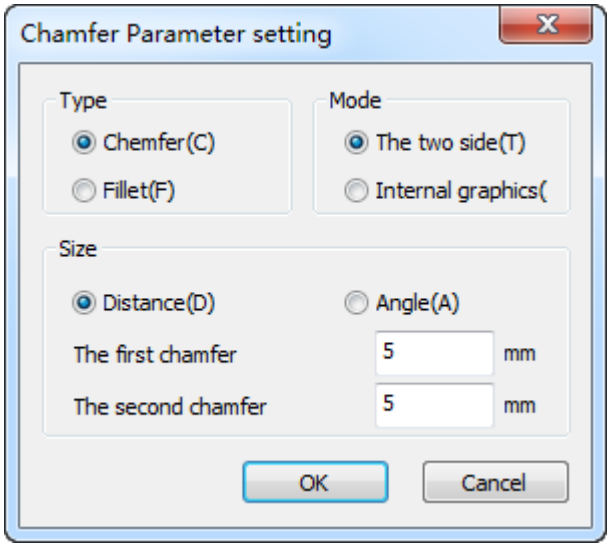
To deburr on parts, use chamfer function to cut the angle of workpiece to slope.

To use chamfer function, do the following:

1. To call **Chamfer Parameter Setting** dialog box, do one of the following:

   - Click **Object** → **Champfer** in menu bar.
   - Right click in edit window and select **Champfer**.

   Chamfer Parameter Setting dialog box is as follows:



2. Select chamfer type, mode and size setting mode, and enter size.
   - Select **Distance** mode: Enter chamfer distance between two edges.
   - Select **Angle** mode: Enter chamfer length and angle of the first edge.
3. Click **OK**.
4. Click the object.

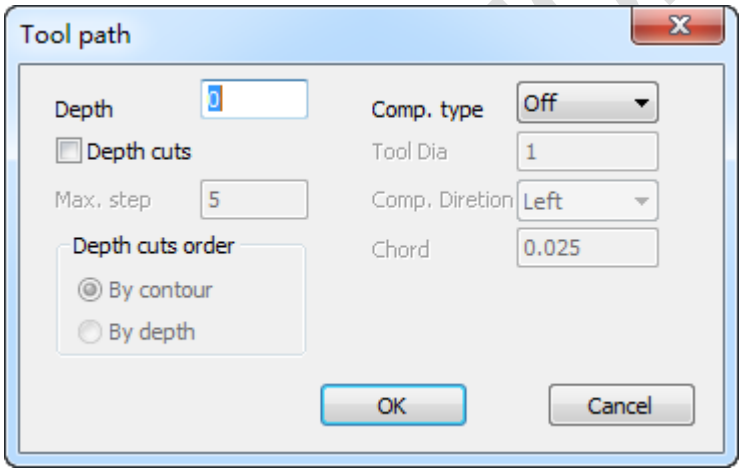
7.7 Set Tool Path

Tool path function is used to set tool compensation by setting tool diameter.

To set tool path, do the following:

1. Check the target object to select it in object list area.
2. To call **Tool path**, do one of the following:
   - Click **Object** → **Tool Path** in menu bar.
   - Right click in edit window and select **Tool Path**.

   Tool Path dialog box is as follows:



3. Enter the values in the dialog box.
4. Click **OK**.

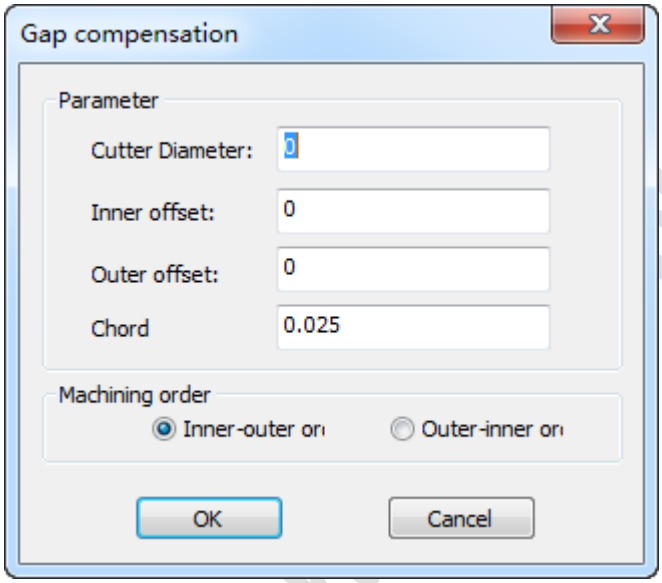
7.8 Expand Bidirectionally

Expanding bidirectionally function is used to set left and right tool compensation at the same time.

To expand bidirectionally, do the following:

1. Check the target object to select it in object list area.
2. Do one of the following to open **Gap compensation** dialog box:
   - Click **Object** → **Expand Bidirectionally** in menu bar.
   - Right click in edit window and select **Expand Bidirectionally**.

   Gap compensation dialog box is as follows:



3. Enter the values in the dialog box.
4. Click **OK**.

8 Set Bus Function

This chapter mainly introduces bus function in bus type configuration.

Setting bus function includes the following:

- Setting drive axis address
- Setting control system parameters
- Setting drive parameters

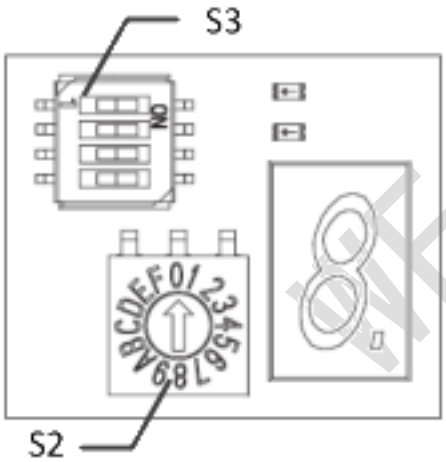
Before commissioning, please check the hardware configuration, and make sure all have been correctly connected.

8.1 Set Drive Axis address

Setting drive axis address is the key to data transmission among control system, Lambda controller and the drive.

8.1.1 Set Axis Address of YASKAWA Drive

Setting axis address of YASKAWA drive via rotary switch S2 and toggle switch S3.



For a YASKAWA drive, to set axis address, do the following:

1. Set S3: set switch 3 according to actual situation, and the other use the factory setting.

   Please refer to the table below for setting:

Switch No.	Function	Description	Factory
1	Setting communication speed	OFF: 4Mbps(M1) ON: 10Mbps(M2)	ON
2	Setting byte number	OFF: 17 bytes ON: 32 bytes	ON
3	Setting station address	OFF: Actual address=40H+S2 ON: Actual address=50H+S2	OFF
4	System reservation (Unchangeable)	OFF	OFF

2. Set S2: rotate S2 to the target value.

   Please refer to the table below for actual station address:

No.3 of S3	S2	Station address	No.3 of S3	S2	Station address
OFF	0	Invalid	ON	0	50H
OFF	1	41H	ON	1	51H
OFF	2	42H	ON	2	52H
OFF	3	43H	ON	3	53H
OFF	4	44H	ON	4	54H
OFF	5	45H	ON	5	55H
OFF	6	46H	ON	6	56H
OFF	7	47H	ON	7	57H
OFF	8	48H	ON	8	58H
OFF	9	49H	ON	9	59H
OFF	A	4AH	ON	A	5AH
OFF	B	4BH	ON	B	5BH
OFF	C	4CH	ON	C	5CH
OFF	D	4DH	ON	D	5DH
OFF	E	4EH	ON	E	5EH
OFF	F	4FH	ON	F	5FH

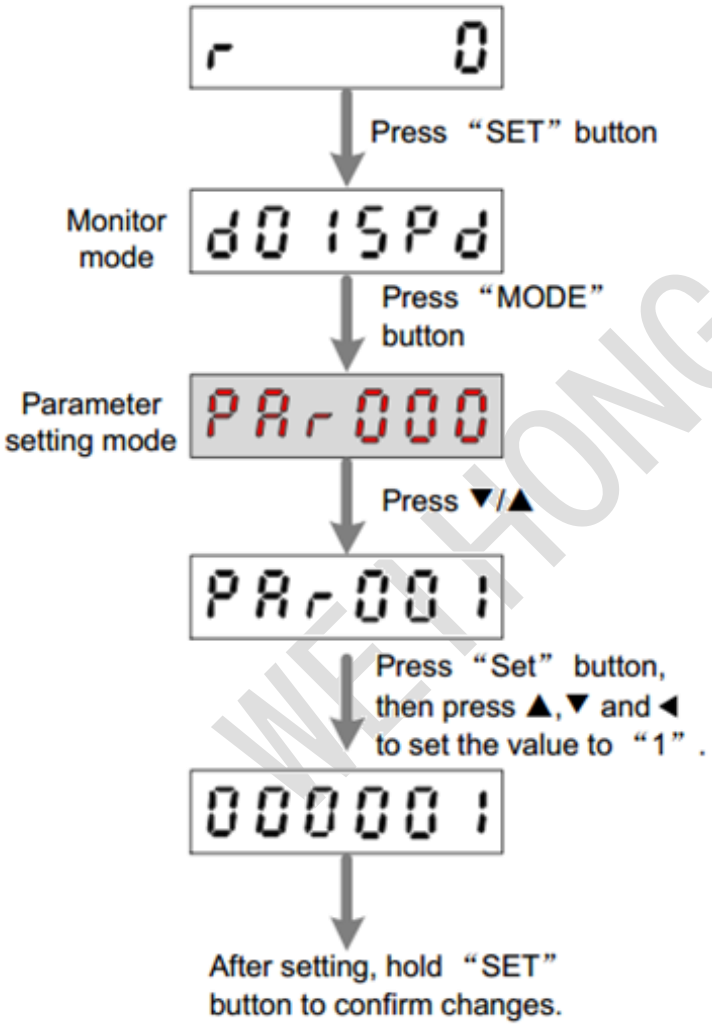
3. Restart the drive.

8.1.2 Set Axis Address of WISE Drive

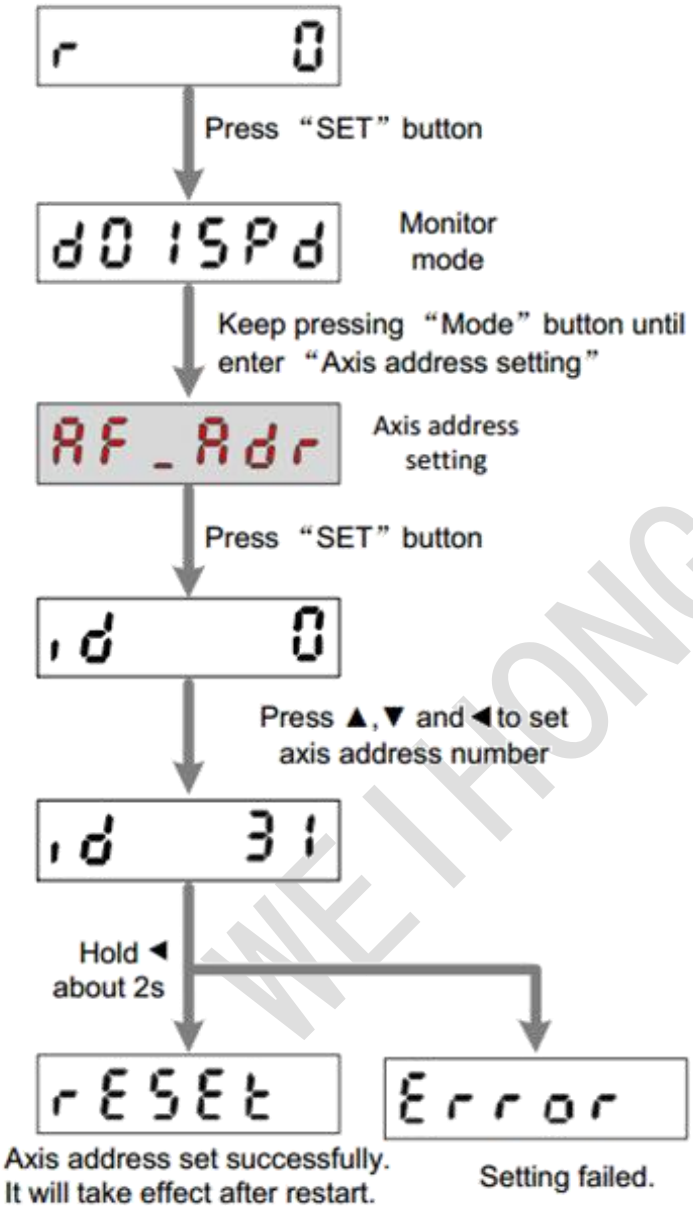
Setting axis address of WISE drive via the front panel.

For a WISE drive, to set axis address, do the following:

1. Set parameter **Pr001 Control mode setup** as **1 Position control mode** as following:



2. Set axis address No. as following:



3. Restart the drive.

8.2 Set Control System Parameters

To use bus function, setting control system parameters in NcStudio software is required.

To set the following parameters, please refer to Set Parameters:

N11001 Encoder type
Choose encoder type according to actual situation.

0: Normal

1: Incremental

2: Absolute

N15000 Control system type
Choose control system type.

0: Non-bus control system

1: Bus control system

N16050~16053 Servo type X/Y/A
Choose servo type.

0: YASKAWA ∑5 servo drive

1: YASKAWA ∑7 servo drive

2: WISE servo drive

N16000~16003 Driver station address X/Y/A
It should be the same as rotary switch setting of drive station address. 0 is invalid address.

Setting range: 0~15

N16010~16013 Driver station address setting switch X/Y/A
It should be the same as toggle switch S3 setting of drive station address.

Yes: PN

No: OFF

N16020~16023 Encoder digit X/Y/A
The digit number of encoder of the servo motor.

Setting range: 10~30

N16990 Second axis driver station address
The drive station address of the second axis.

Setting range: 0~15

N16991 Second axis driver station address setting switch
The drive station address setting switch of the second axis.

Yes: ON

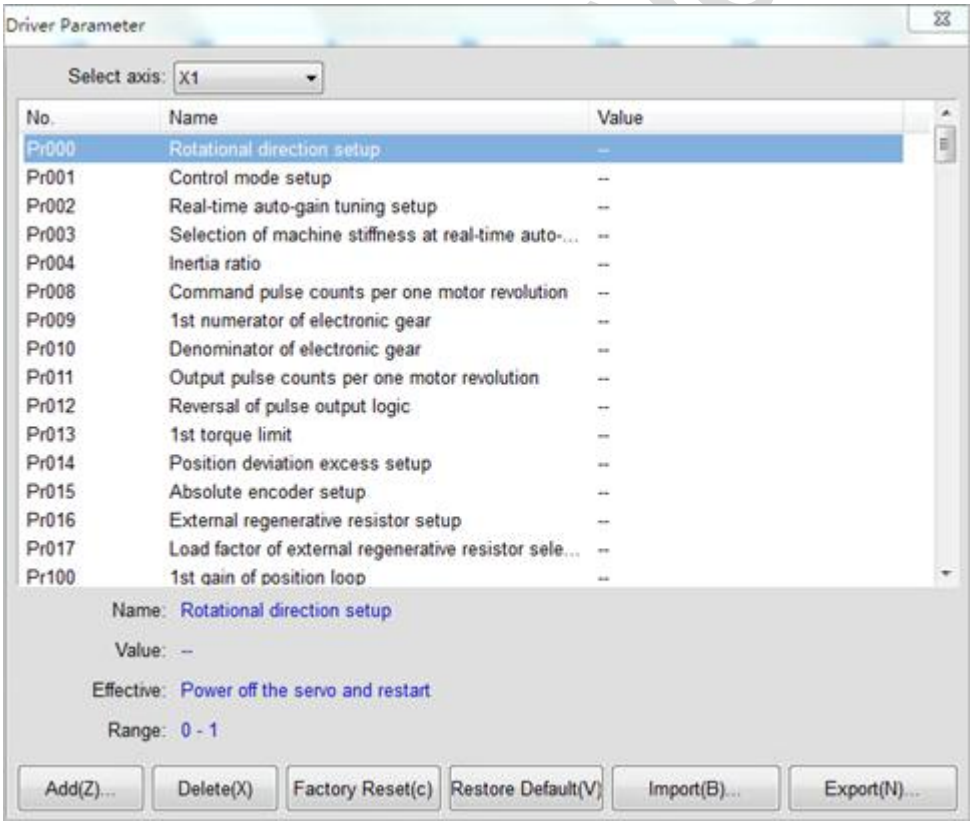
No: OFF

8.3 Set Drive Parameters

To use bus function, setting drive parameters in NcStudio software is required.

To set drive parameters, do the following:

1. Click **Machine Tool** → **Set Servo Param** in menu bar. A dialog box named **Driver Parameter** pops up.



For a WISE drive, all parameters will be shown in the dialog box. While for a YASKAWA drive, only part of parameters will be shown. If you need to add more, click **Add**.

2. Click **Select axis** to select an axis to be set.

   Different axis has different drive and parameters.

3. Scroll the mouse or press arrow keys "↑" and "↓" on keyboard to select parameter.

4. Press "Enter" or double click to modify the selected parameter.

5. Click **OK**.

6. Click **Machine Tool** → **Clear Servo Alarm**.

9 Frequently Asked Questions(FAQ)

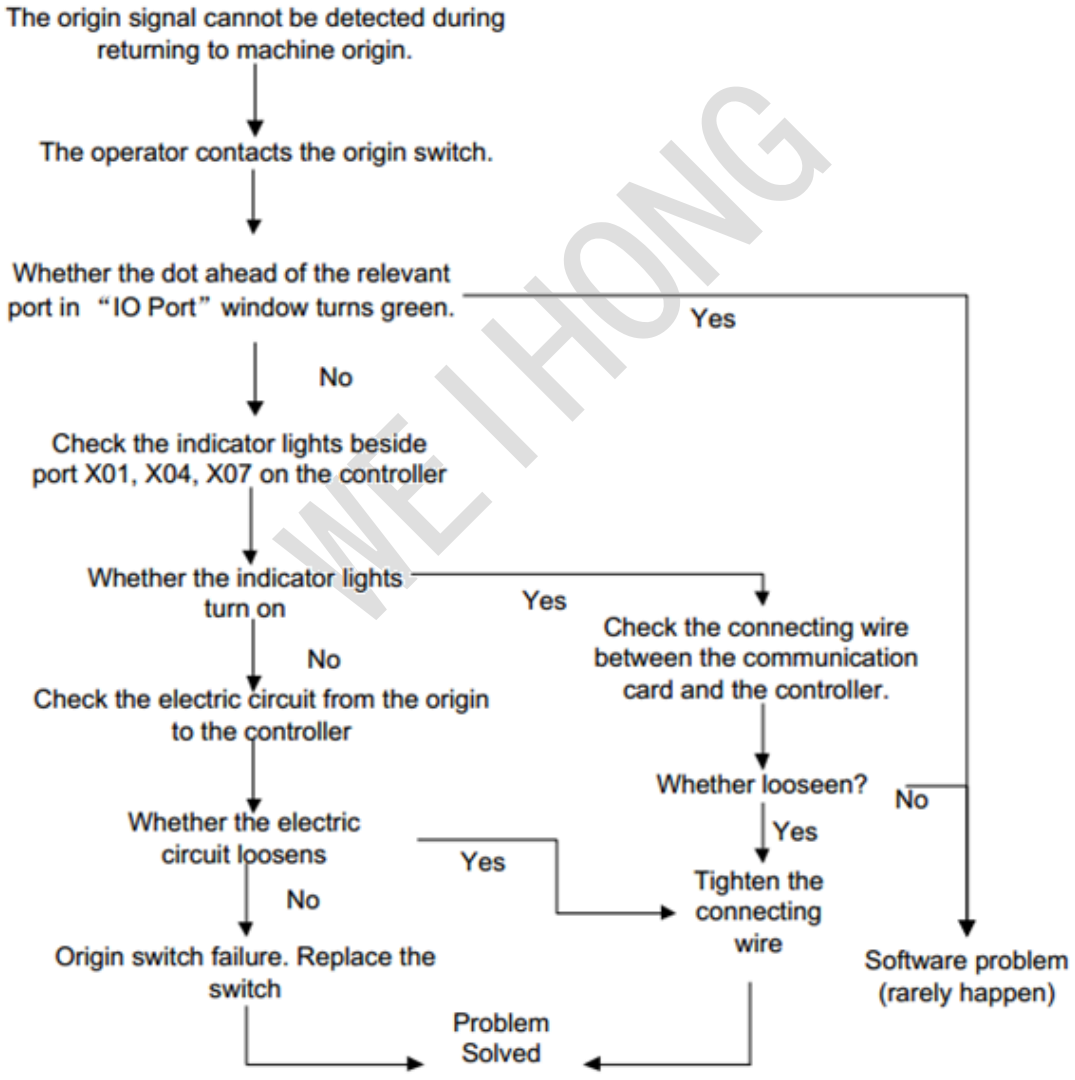
This section contains answers to common questions in **NcStudio V10 Glass Cutting CNC System**.

9.1 The origin signal cannot be detected during returning to machine origin.

Cause

Errors may exist in origin switch.

Troubleshooting



9.2 The direction of the machine during returning to machine origin is wrong.

Cause

1. The polarity of the origin signal is wrong.
2. Set related parameters incorrectly.

Troubleshooting

1. The polarity of the origin signal is N when the origin switch is normally open; otherwise, it is P.
2. Check parameter **N74020 Coarse Locating Direction**, and change the value of the parameter accordingly.

9.3 Coarse positioning speed is low during returning to machine origin.

Cause

1. The set value of parameter **N74030 Coarse Locating Velocity** is too small.
2. The polarity of the origin signal in the software does not match the type of the origin switch. If the origin switch is normally close, and the polarity of the origin signal is N, the machine will move away from the origin at fine positioning speed.

Troubleshooting

1. Modify the value of parameter **N74030 Coarse Locating Velocity**.
2. Modify the polarity of the origin signal according to the type of the origin switch.

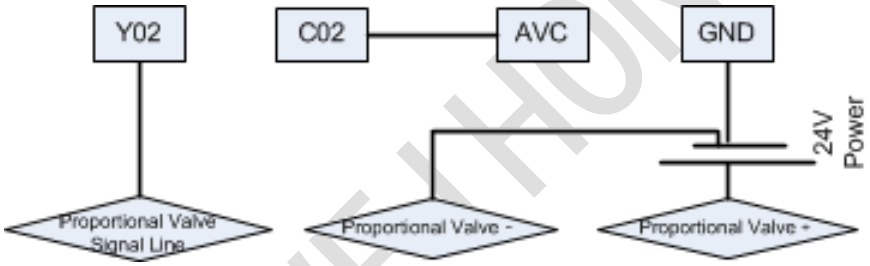
9.4 The machine air cylinder does not plunge cut after Plunge is clicked in machine control bar on the software interface.

Troubleshooting

1. In auto mode, check whether Plunge turns green after being clicked.
2. Click **Port**. Check the red dot ahead of output port **Analog Enable** turns green and outputs.
3. Check whether the output port Y02 on the terminal board outputs.

   If the port outputs, then the software is proved connected with the hardware.
4. Check whether the proportional electro-magnetic valve has been connected with the output port Y02 on the terminal board correctly.

   See following figure for wiring:



5. If the problem remains, use the multimeter to detect ends of ports **Y02** and **GND**, or the analog voltage between **Proportional Valve Signal Line** and **Proportional Valve**.
6. Check whether the wiring between the proportional valve and the terminal board loosens.
   If the wiring loosens, reconnect it.

9.5 A-axis rotates incorrectly during machining toolpath.

Troubleshooting

1. Modify the value of parameter **N83013 Angle for Tool Turning**. The suggested value is 60°.

2. If the angle of the adjacent two lines is about 60° by visual measurement, and A-axis still rotates incorrectly, please modify the value of the parameter to no more than 59°.

   The boundary angle value of the adjacent lines is prone to error. For example, the actual angle of the adjacent lines is 59.99°, while it is shown as 60° in the software. Therefore, it is necessary to continually modify the value until the problem is solved.

9.6 The machine shakes during machining ellipse.

Cause

If the value of parameter **N64100 Max Linear Machining Acceleration** is too great, such as 160000mm/min, the machine will shake during machining of ellipse (about 200*150mm).

Troubleshooting

1. Set the value of parameter **N64120 Max Control Acceleration** as twice to five times that of parameter **N64100 Max Linear Machining Acceleration**.

2. Increase the value of parameter **N64208 Reference Circle Max Velocity** to a greater value, such as 5000mm/min or 10000mm/min.

3. Set the value of parameter **N64207 Limit Arc Velocity** as **No**.

9.7 Software expired.

Cause

1. Attachment destroyed.

2. Drive expired.

Troubleshooting

1. Change the attachment.

2. Unlock the drive with iMotion software.

9.8 Exception prompt occurs during registering.

See the table below for details:

Prompt	Cause	Solution
Registration failure! Invalid registration code!	The registration code is incorrect.	Ask the manufacturer to provide a correct registration code, and register again.
Drive alarms. The alarm No. is 513.	Drive length of time registration expired.	Register length of time again. Unlock the drive with iMotion software.
Check wiring of drive N.	Set software parameters incorrectly. Set wiring/version/parameter of the drive incorrectly.	Check parameters. Check wiring of the drive.
Registration failure. The N drive fails to register.	Problem of drive wiring. Problem of drive.	Wiring problem: Check wiring of the drive. Drive problem: Change the drive. Other problems: Contact with WEIHONG.

9.9 Alarm Information

NcStudio system stops when alarm information occurs. After the alarm is removed, it continues working.

Alarm information is in the table below:

Type	Content	Cause	Solution
 Warning message	Simulation results show that program range exceeds the machine travel limit.	Tool path of the program file exceeds the upper/lower limit of workbench travel, which are decided by settings of parameter N10020 and N10030 separately.	Check if the WCS zero is reasonable. Check the tool path program file. Modify parameter settings of N10020 and N10030 to enlarge the workbench travel limit.
	The system has not returned to the machine origin, failed to execute the operation!	The system has not returned to machine origin. Whether the system has to return to the machine origin is decided by parameter N74001.	Use this function after returning to the machine origin.
	The system is busy. This operation can't be executed.	Some illegal operations are performed under machining state.	Stop machining, and execute some operations under idle state.
 Limit alarm	Positive (negative) limit of X (Y/Z/A) axis	The polarity of X axis positive limit port is wrong.	Enter "IO Port" window in "DIAG" interface, and modify the port polarity.
		X axis runs into limit switch directly during motion.	Manually move X-axis away from limit switch.
		There is an error in limit switch itself.	Check if limit switch works normally.

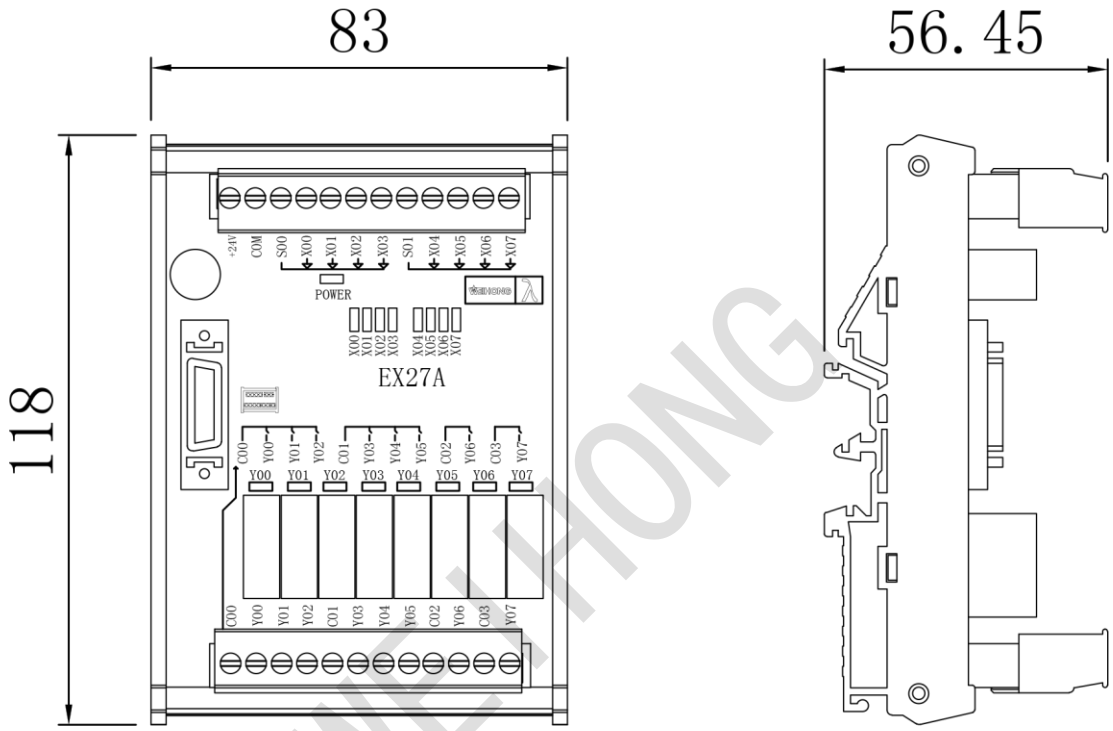
Type	Content	Cause	Solution
Servo alarm	Servo alarm of X (Y/Z/A) axis	The polarity of X-axis servo alarm port is wrong.	Enter “IO Port” window in “DIAG” interface, and modify the port polarity.
		There is an error in X-axis servo drive itself.	Check if X axis servo drive works normally.
E-stop alarm	E-stop button is pressed.	The polarity of E-stop port is wrong.	Enter “IO Port” window in “DIAG” interface, and modify the port polarity.
		The E-stop button is pressed.	Turn the E-stop button clockwise to make it bounced.
Machining alarm	Machining executive program failure	Software license expired	Register the software.
Terminal board not connected error alarm	Terminal board not connected.	Cable has not been firmly connected or there is error in Lambda controller.	Re-plug the cable and restart the software to observe the occurrence of the error; Polarity of port is wrong. Invert the port polarity and restart the software; Analyze possible causes according to the status of indicator “SYSTEM” on Lambda controller; Change another Lambda controller.

10 Related Dimensional Drawing

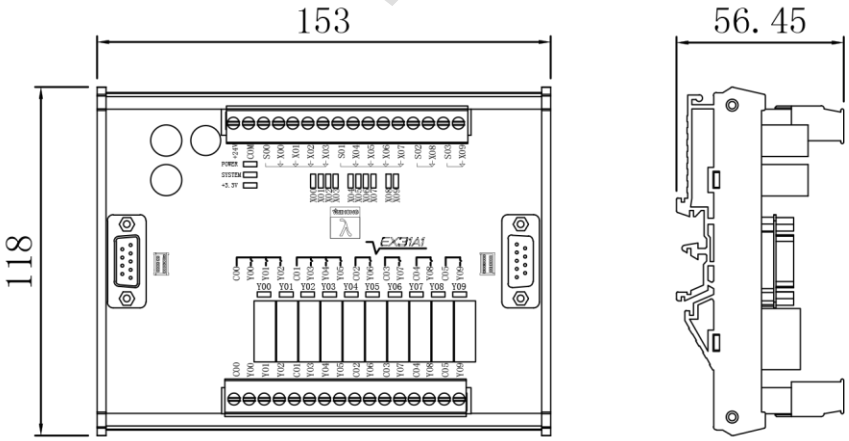
This chapter introduces product dimension. Before installing, please confirm that the space is sufficient.

Product dimensional drawing are as follows:

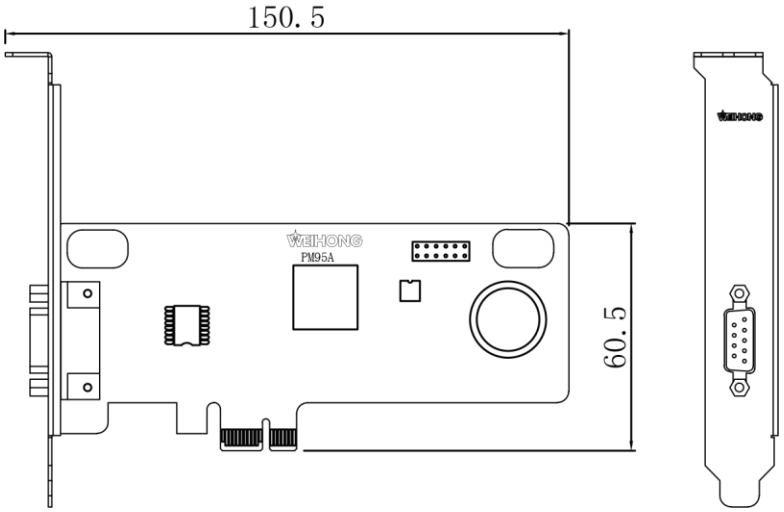
EX27A terminal board dimensional drawing:



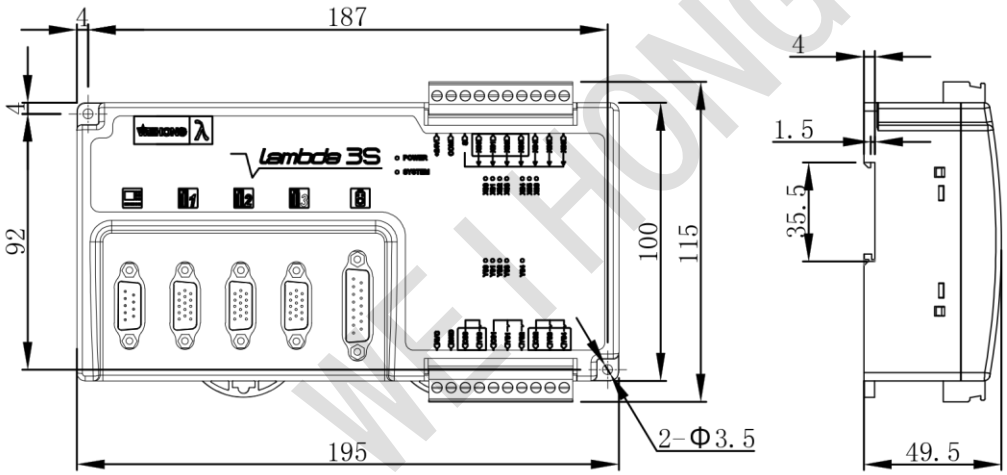
EX31A terminal board dimensional drawing:



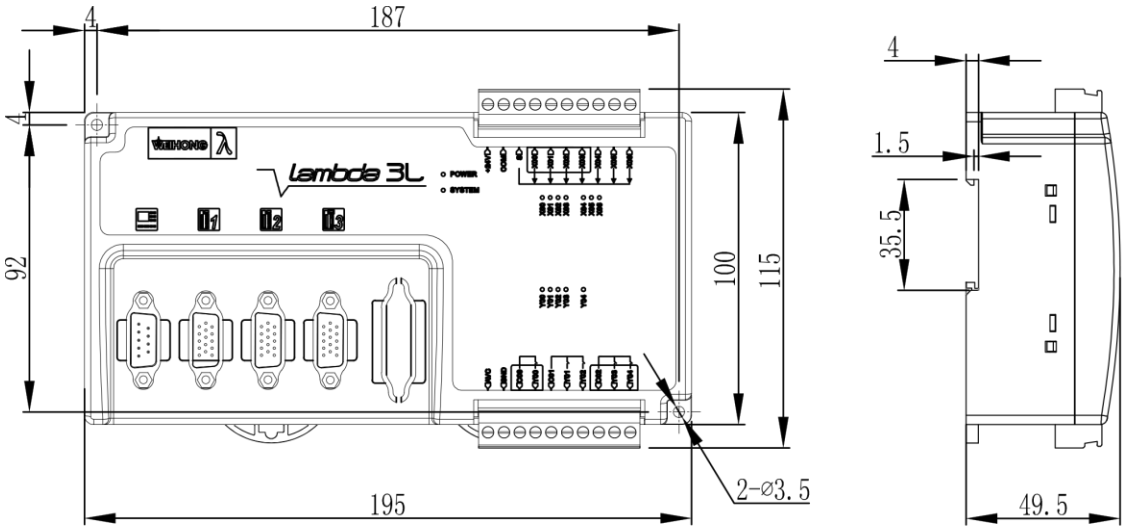
PM95A motion control card dimensional drawing:



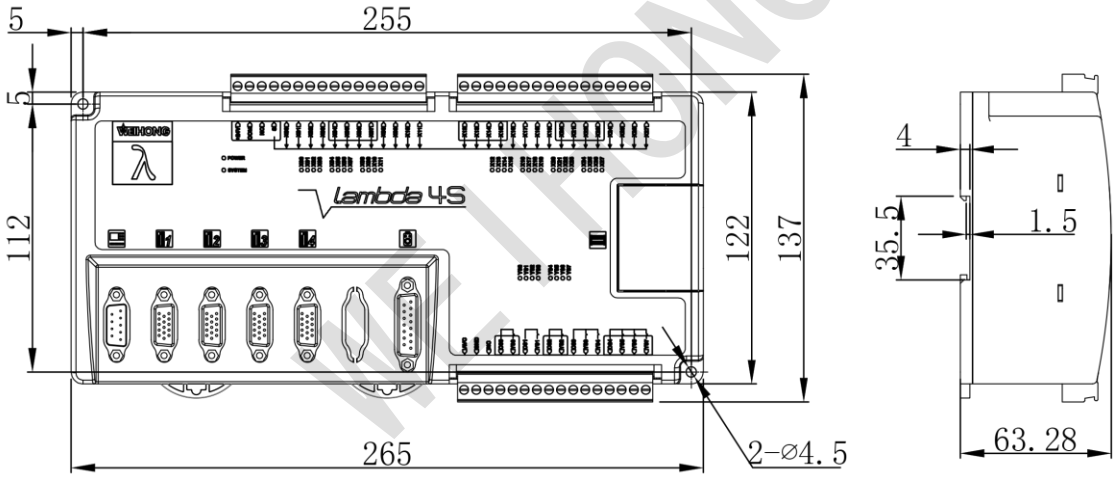
Lambda 3S controller dimensional drawing:



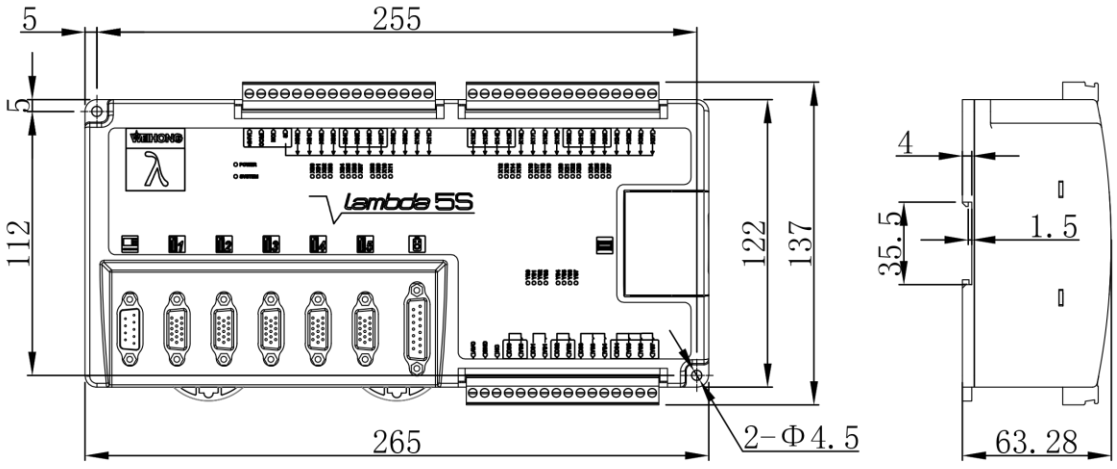
Lambda 3L controller dimensional drawing:



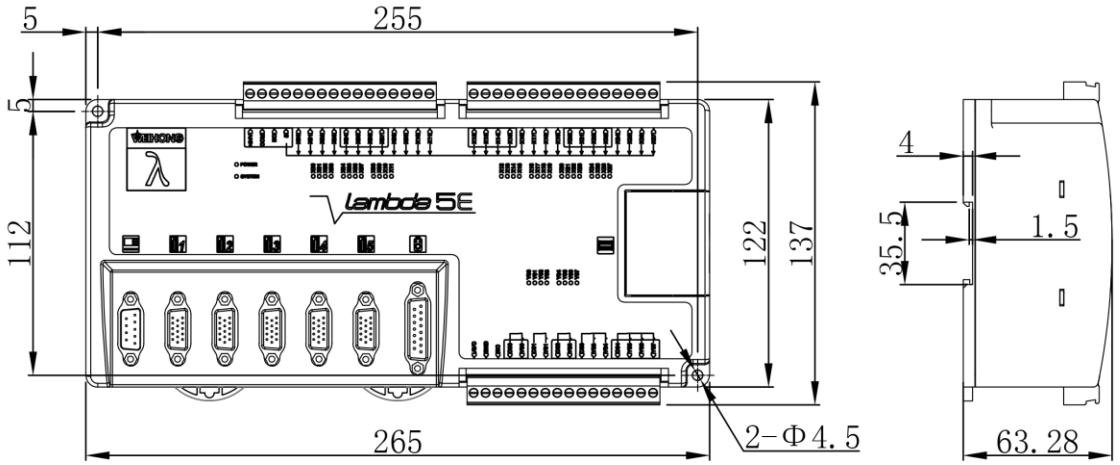
Lambda 4S controller dimensional drawing:



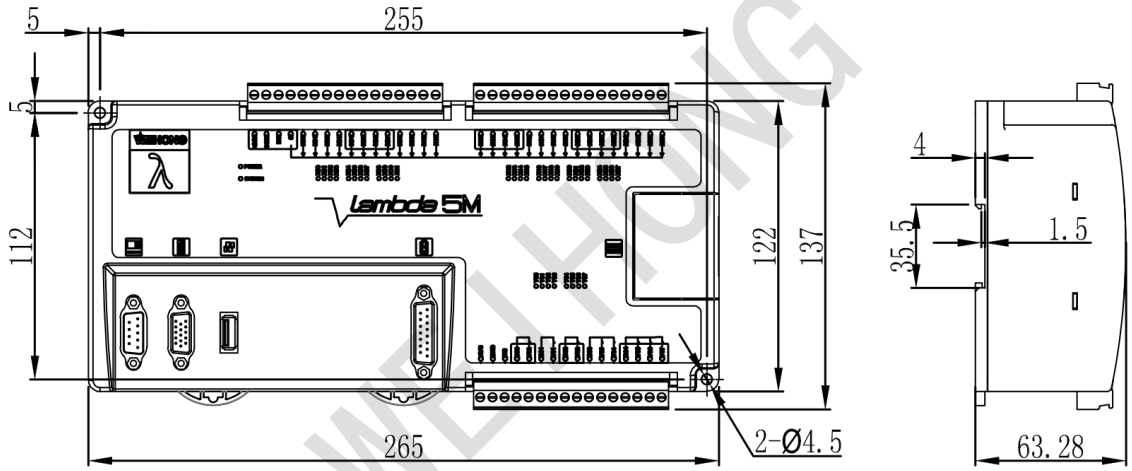
Lambda 5S controller dimensional drawing:



Lambda 5E controller dimensional drawing:



Lambda 5M controller dimensional drawing:



SPECIALIZED / CONCENTRATED / FOCUSED

Shanghai Weihong Electronic Technology Co., Ltd.

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

Hot-line: 400 882 9188

Website: www.weihong.com.cn/en