User Manual of Ncstudio V15 Laser Cutting (Tube) CNC Control System-R2

Version: 2022.12.25 2nd version
Author: Product Testing Department

Contents

Preface

Thank you for using the NcStudio-V15 Laser Tube Cutting Control System!

This manual gives a detailed introduction to the use of **NcStudio-V15 Laser Tube Cutting Control System** (NcStudio-V15 for short), including:

- Product introduction
- Tube machining flow
- Loading and editing of tool path files (including graphic operation, setting technic, nesting, arrange the path, and simulation machining)
- Machining related operation
- Set parameter
- System maintenance
- System management

Please read this manual carefully before installing and using this product. Any improvements or technical changes will not be notified by other means. For more information, you can visit the official website: https://www.weihong.com.cn/en/.

Symbols

Special symbols are used to indicate content that require your extra attention.

Used to highlight suggestions that need your extra attention and supplementary information and notes to the text body.

Hazards or unsafe practices which may result in equipment damage, data loss, lowered equipment performance or other unpredicted consequences.

Product Introduction

1.1. Function Overview

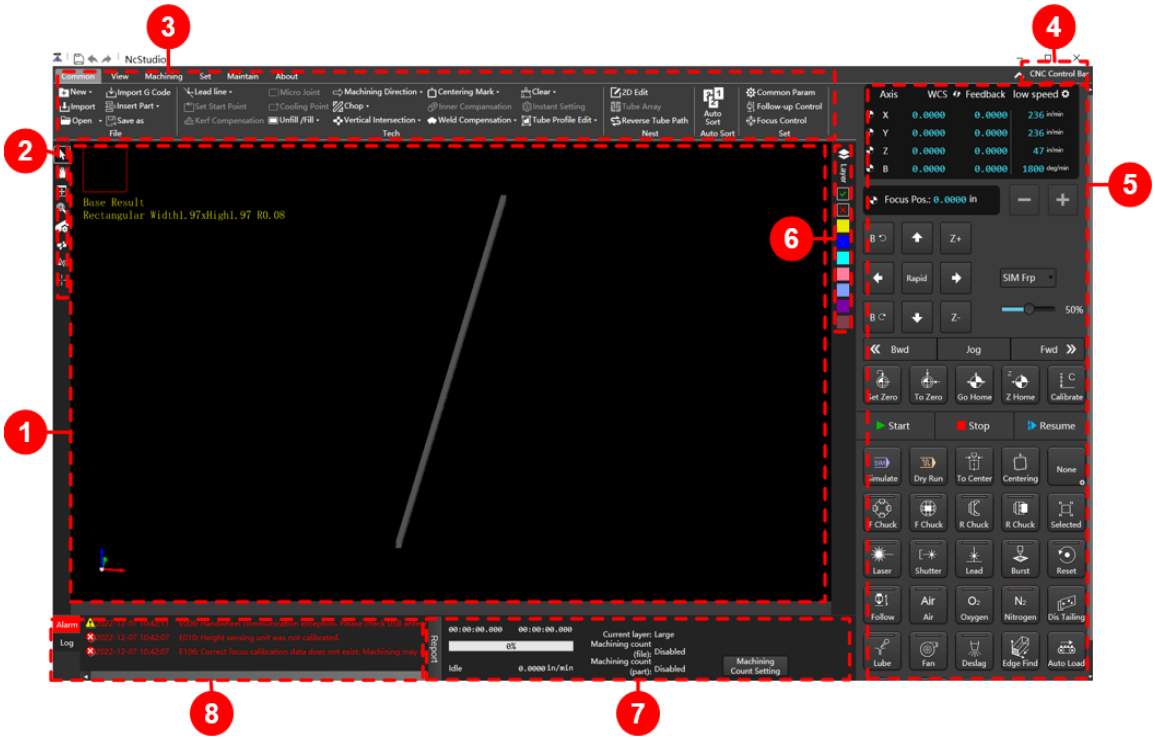
Weihong's **NcStudio-V15 Laser Tube Cutting Control System** includes laser cutting technic processing, common layout function and laser machining control. The main functions include tool path file edit, system parameter set, customized machining process, system maintenance set and cutting control.

The tube type supports circular tube, rectangular tube, oval tube, waist tube, angle steel, channel steel and other special shaped tubes.

1.2. Introduce the Main Interface of the Software

This chapter mainly introduces the main interface of **NcStudio-V15**.

The graphic is as follows:



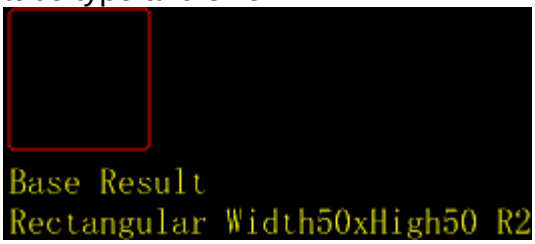
1.Drawing Area 2.Common Toolbar 3.Menu Bar 4.Machine Control Bar Display Button 5.Machine Control Bar 6.Layer Toolbar 7. Machining Information Statistics Bar 8. Alarm/Log Information Bar

1.2.1. Drawing Area

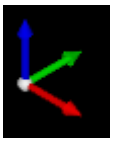
View tubes from different perspectives, preview cutting effects, and select graphics and parts to add technics.

Include the following areas:

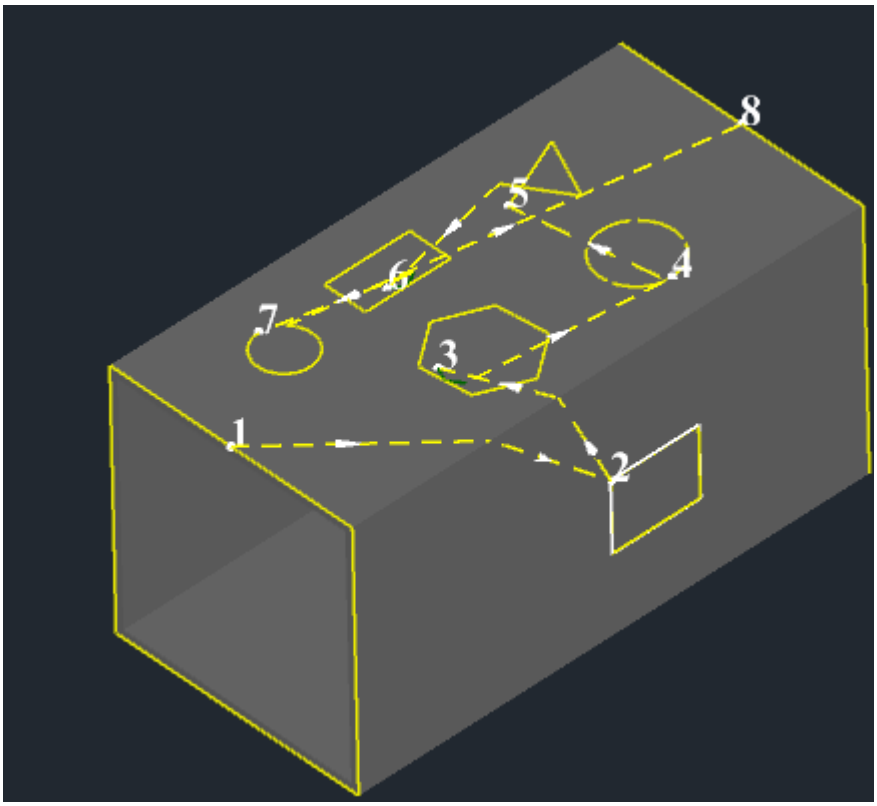
- Top left corner area: Display the tube projection section outer contour, view name, tube type and size.



- Bottom left corner area: Display the 3D coordinate system.



- Middle area: Display the base layer or nest result and the added technic.



1.2.2. Common Toolbar

Provide common tool buttons:

Button	Description
Select Graph	Choose any graphic you want.
Pan	Move the graphic in a straight line direction to change the coordinate position of the graphic without changing the shape and size of the graphic.
Best View	The adaptive size of the graphic is displayed in the window.
Zoom by Rect	Enlarge the part of the graphic to the size of the view window.
Tube Cutting Setting	Check the tube thickness and total tube length, which can be set.
Tube Intersection	The tube surface perforation can be made by intersecting branch tubes on the tube, and the array can be set.
Tube Cutting	Cutting graphic is generated on the tube to cut off the tube parts.

Button	Description
 Mark Point	It is used to set the mechanical coordinate of the target position as the mechanical coordinate of the marked point, and move the cutting head back to the marked point when necessary.

1.2.3. Menu Bar

The following function tabs are included:

Tab	Function description
Common	It includes file operation, set technic, nest operation, automatic sorting and set common parameters of the machine tool. For related operations, see Loading or Drawing Tool Path, Edit Path File, Set Common Parameter, Follow-up Control, and Focus Control.
View	Select graphics, display technics, and adjust views. For related operations, see View Operation.
Machining	Machining related settings, including zeroing, centering, technic library management, machining mode selection, viewing production reports, operation reports, and logs. For related operations, see Machining Related Operation.
Set	It is mainly used to set system parameters, ports, drivers and lasers. Lead screw error, feed cutting compensation and special material pulling parameter setting. For related operations, see Set Parameter.
Maintain	It include automatic settings, external equipment maintenance management and common maintenance management tools.br>For related operations, see System Maintenance.
About	It includes software language, unit, theme, password and other settings, common shortcut key description, parameter backup, installation package production, etc. For related operations, see System Management.

1.2.4. Machine Control Bar Display Button

Show or hide the Machine Control Bar.

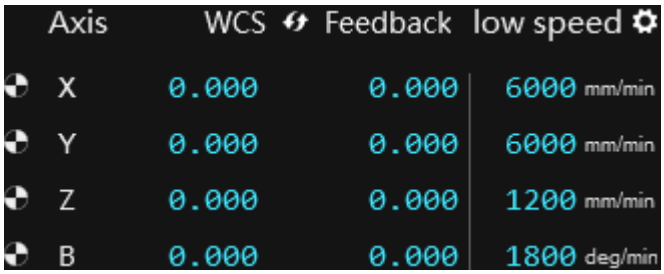
1.2.5. Machine Control Bar

The CNC control bar includes:

- Coordinate Display Area
- Manual Control Area
- Common Function Buttons
- Machine Tool Control Button

1.2.5.1. Coordinate Display Area

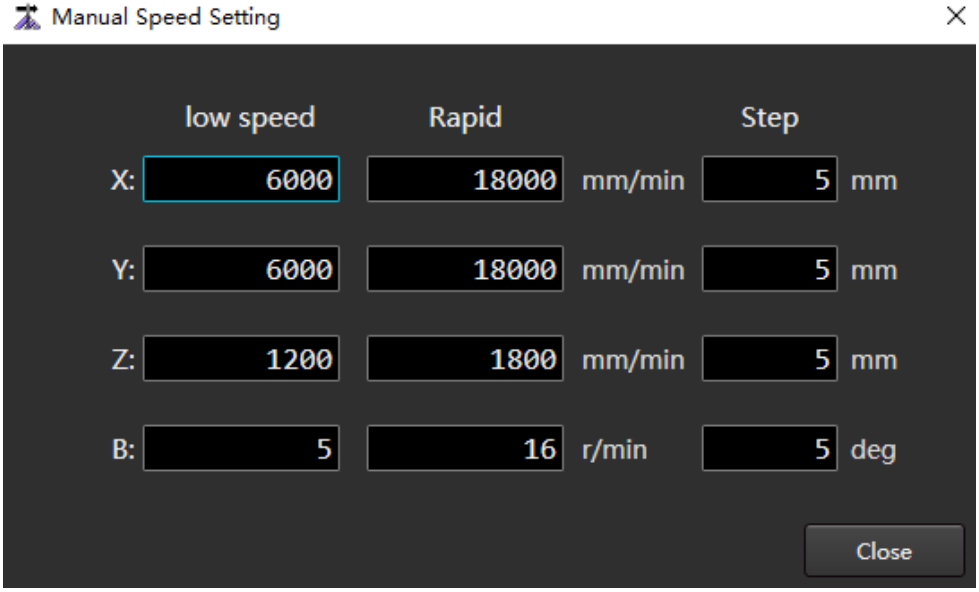
Display WCS, MCS, feedback coordinates, low speed, rapid and step distance of each axis:



After returning to the mechanical origin, sign will appear in front of each axis.

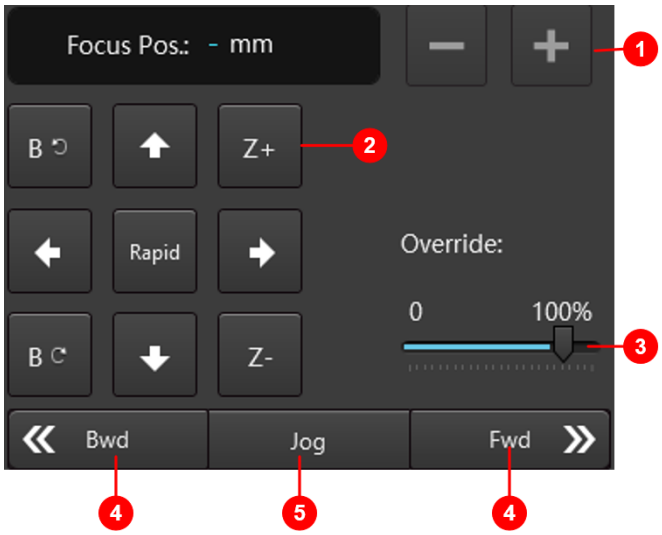
In this area, the following operations can be performed:

- Click WCS to switch WCS and MCS.
- Click to set the low speed, rapid and step of X, Y, Z and B axes.



1.2.5.2. Manual Control Area

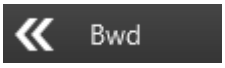
The manual control area is as follows:



1. Focus position

   Click / button to adjust the focus position.
2. Axis direction button

- Click the direction button  (Y+),  (Y-),  (X+),  (X-),   to control the positive or negative movement of each axis of the machine tool.
- Click the direction button   to control the rotation direction of axis B.

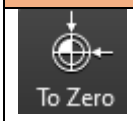
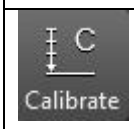
3. Feed override
   Before or during machining, select the following methods to adjust the feed speed.
   - Drag the override speed bar with the mouse to adjust the feed speed.
   - Click the target position of the override speed bar.
   - After clicking the override speed bar, press the **PgUp**, **PgDown** or ↑, ↓, ←, → keys on the keyboard.
4. Machine tool motion control button
   Click  /  button, the machine tool will continuously move in reverse/forward direction along the tool path.
   - In **Jog** mode, release the button to stop the machine.
   - In **Step** mode, the movement stop after setting the step size value.
5. Mode selection button
   According to the actual situation, click  /  to switch mode.

Mode	Description
Jog mode (default mode)	▪ Click the single axis direction button, the axis moves at a continuous low speed, and stop after the button is released. ▪ At the same time, click multiple axis direction buttons. The selected axis move at a continuous low speed at the same time and stop at the same time after the button is released.▪ At the same time, click  the rapid button and the single axis direction button. The axis moves in a continuous rapid mode and stop when the button is released.▪ At the same time, click  the rapid button and multiple axis direction buttons. The axis moves in a continuous rapid mode and stop at the same time after the button is released.
Step mode	Click the axis direction button, and the machine tool will stop after moving the set step size (default value is 5mm).To customize the step value, click  in the upper right area of **Coordinate Display Area** . **Note：** Do not set the step size too large or click the axis direction button frequently to avoid damage to the machine tool due to incorrect operation or too frequent operation.

1.2.5.3. Common Function Buttons

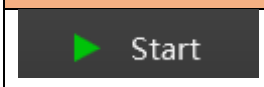
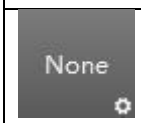
Common function buttons:

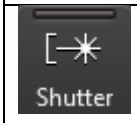
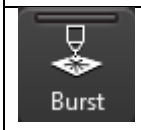
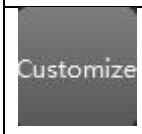
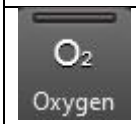
Button	Description
	Return to the workpiece origin.

Button	Description
	Control the feeding axis, laser head horizontal moving axis, laser head follow up axis, rotation axis or focus axis to return to the workpiece origin.
	Execute all axes back to mechanical origin.
	Execute Z-axis return to mechanical origin.
	Execute the specified action of cutting head.

1.2.5.4. Machine Tool Control Button

Start and stop machining and related operations of laser cutting:

Button	Description
	Start this machining task from the beginning.
	Stop this machining task.
	After the machining is stopped, the machining shall be continued from the position where the last machining was stopped under the condition of ensuring the accuracy of the mechanical coordinates.
	Enter simulate mode. The system does not drive the machine tool to perform corresponding mechanical and electrical actions, but only simulates the machining path at high speed in the drawing area.
	Enter dry run mode. Run the machine tool without turn on the laser and machining related ports , check whether the machining action is correct.
	Control the rotation axis of the machine tool and the transverse movement axis of the cutting head to return to the fixed position: the rotation axis return to the workpiece origin, and the transverse movement axis of the cutting head returns to the tube centerline.
	Adjust the tube to horizontal and find the center of the tube. The centering method can be selected. Please refer to the buttons below for details.
	Set the centering method. The text on the button indicate the currently selected centering method. You can only select the part of the centering method through this button, and the optional centering method is the centering action available for automatic matching of the current tube type. For more centering methods, see Auto Leveling and Centering.

Button	Description
	The jaws that control the front chuck release. After the chuck control type is selected in the system parameters, the chuck will be controlled by IO/torque/position through the port.
	The jaws that control the front chuck clamp. After the chuck control type is selected in the system parameters, the chuck will be controlled by IO/torque/position through the port.
	The jaws that control the rear chuck release.
	The jaws that control the rear chuck clamp.
	Select graphics during machining, including **Machining**, **Dry Run**, **Simulate** and **Resume**.
	Press and hold the laser valve until it is released and closed. When machining start, the system automatically open the laser valve.
	Click open shutter, and then click close. The shutter must be manually clicked to open. First open the shutter, then open the laser valve, and finally the laser will come out.
	Click open lead, and then click close, it must be opened manually. The lead is used to indicate the position of the laser on the sheet.
	Open the laser valve regularly. The system open the laser valve at the same time, and automatically close after continuously setting the burst parameters.
	Reserved user-defined button.
	Click to open the follow up valve. After opening, the distance between the nozzle and the workpiece surface can be adjusted in real time to keep it at a fixed value. At the beginning of machining, the system automatically open the follow up valve.
	Click to manually open the blowing valve, and then click to close it, the blowing gas is air. At the beginning of machining, the button will be highlighted, and the system will automatically open the blowing valve to blow air.
	Click to manually open the blowing valve, and then click to close it, the blowing gas is oxygen. At the beginning of machining, the button will be highlighted, and the system will automatically open the blowing valve to blow oxygen.

Button	Description
	Click to manually open the blowing valve, and then click to close it, the blowing gas is nitrogen. At the beginning of machining, the button will be highlighted, and the system will automatically open the blowing valve to blow nitrogen.
	Start dis tailing. When processing the tailings, it is necessary to ensure that the system is in an idle state, and that the feeding axis, the horizontal moving axis of the laser head, the following axis of the laser head, the clamping axis of the blanking shelf, and the axis that makes the blanking shelf move forward or backward have all returned to the original point.
	Turn on lube.
	Click to manually open the fan power supply, and then click to close. Set the system parameters of **External Equipment Control-Exhaust** category to control the fan to start or stop automatically when machining start or end.
	Set the system parameter **Enable Back Blow** or **Enable Side Blow** to **Yes**, and click this button to enable slag removal.
	Click this button, and the cutting head will perform a tube edge find.
	Click this button to automatically load.
	Turn on the lighting.
	Enable or shield auxiliary support tube.

1.2.6. Layer Toolbar

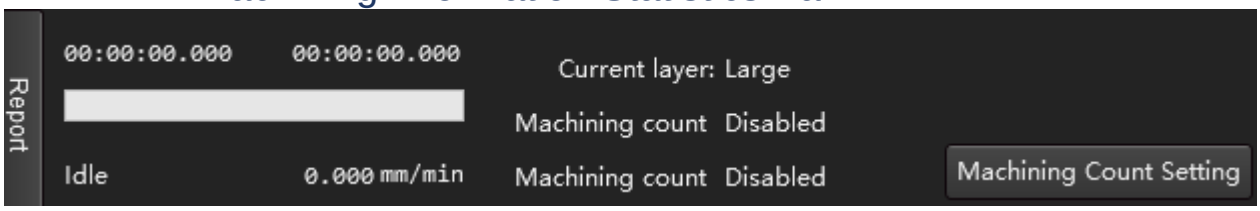
Perform layer related operations.

Including the following parts:

-  : Set layer parameters.
-  : Sets the selected graphic machining.
-  : Set the selected graphic not machining. At this time, the graphic is white.

-  : Set the color of the selected object to the color of the corresponding layer.

1.2.7. Machining Information Statistics Bar

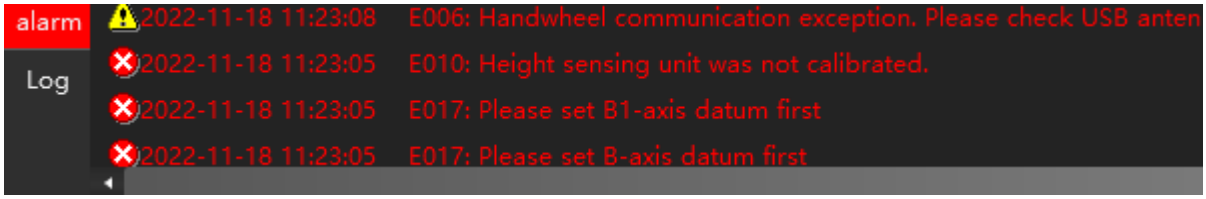


The following operations can be performed:

- Click **Report** to pop up the **Report** dialog box View Report.
- Click **Machining Count Setting** to pop up the **Work Count** dialog box to view and set the machining count.
- View the current total machining time, the remaining time of re machining (when cycle machining is enabled), the current layer, machining count, speed, system status and other information.

1.2.8. Alarm/Log Information Bar

Display machining information and operation error prompt information.



The following operations can be performed:

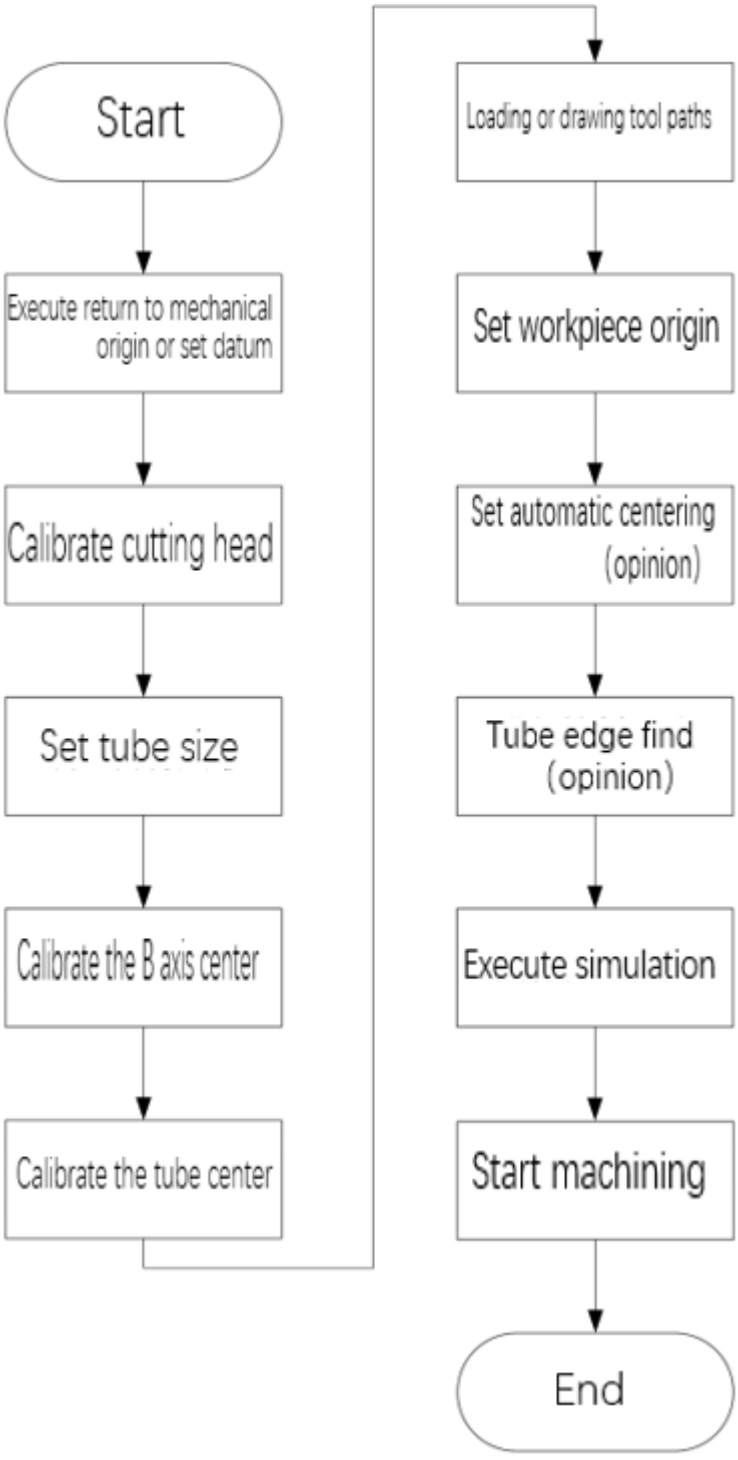
- Double click **Alarm** / **Log**, and the **Log** dialog box pop up. View Log information.
- Double click the corresponding alarm item or log information item to view the time, reason, solution or log details of the alarm in the pop up dialog box.

2. Quick Start

2.1. Tube Machining Flow

Through this part, you can quickly get familiar with the machining flow of laser cutting tubes by the control machine of the **NcStudio CAM Laser Tube Stand-alone Control System**, and control the machine to cut tubes by laser.

The flow chart is as follows:



Process description:

Number	Step	Description
1	Execute return to mechanical origin or set datum	Return to the mechanical origin to synchronize the mechanical coordinate system of the system with the mechanical coordinate system of the machine tool, so must return to the mechanical origin before machining. For details, see Execute Return to Mechanical Origin or Set Datum.
2	Calibrate cutting head	Calibrate the cutting head to ensure that the relative distance between the cutting head and the plate remains unchanged under the follow up state. For details, see Calibrate Cutting Head.
3	Set tube size	Set according to the actual situation. For details, see Set Tube Size.
4	Calibrate the B axis center	Calculate the mechanical coordinate value of the rotary axis center. For details, see Calibrate B-axis Center.
5	Calibrate the tube center	For details, see Calibrate the Tube Center.
6	Loading or drawing tool paths	Load the path file. For details, see Loading or Drawing Tool Paths.
7	Set workpiece origin	Mark sure the actual position of the workpiece origin on the workpiece. For details, see Set Workpiece Origin.
8	Set automatic centering	Optional step: when the tube is long, the tube will be automatically leveled and divided after cutting a certain length, and the breakpoint resume will automatically after completion. For details, see Auto Leveling and Centering.
9	Tube edge find	Optional step: move the feeding axis, so that the cutting head is precisely positioned above the tube cutoff surface. For details, see Tube Edge Find.
10	Execute simulation	Check whether the machining range and tool path travel range are reasonable through simulation, and check the machining path. For details, see Execute Simulation.
11	Start machining	The formal machining phase control the start of machining. For details, see Normal Machining.

2.2. Common Operation

2.2.1. Set Workpiece Origin

The zero point of each axis in the tool path is the workpiece origin. Before machining, it is necessary to determine the actual position of the workpiece origin on the workpiece.

Operating Steps:

1. On the machine tool control bar, click **X+** / **X-** / **Y+** / **Y-** / **B+** / **B-** button to move the cutting head to the target position.

2. In the machine tool control bar, click 

or press the **F5** key to set the current position as the workpiece origin.

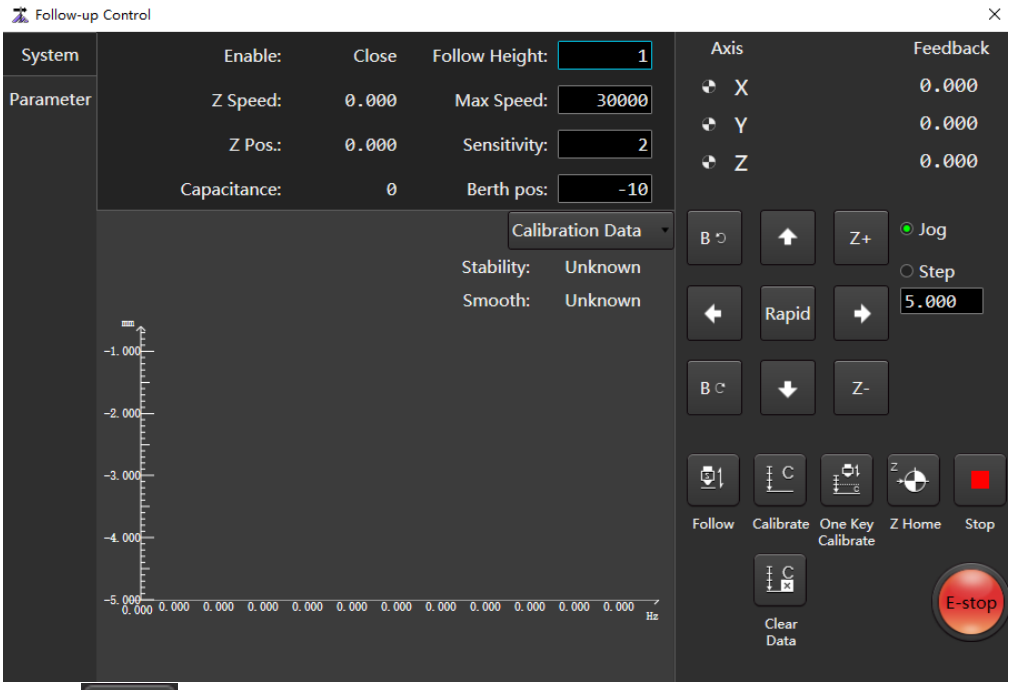
2.2.2. Calibrate Cutting Head

The purpose of calibrating the cutting head is to measure the capacitance and position relationship between the cutting head and the plate, so as to control the Z axis up and down floating in real time. To ensure that the relative distance between the cutting head and the plate remains unchanged under the follow up state.

Operating Steps：

1. In the menu bar, click **Common** → 

to open the **Follow-up Control** dialog box:



2. Click  **Follow** to open the follow-up valve.
3. Click  **Calibrate** to calibrate the servo, solve the problem of servo motor zero drift caused by speed loop control, and the system automatically generate the value of parameter **Servo Compensation**.
4. According to the actual situation, select the following methods, move the cutting head to about 5mm near the plate surface, and keep the plate surface still all the time:
   - If the cutting head has not been calibrated, click the Z-axis direction button, move the cutting head to about 5mm on the board surface, and keep the board still all the time. In the machine tool control bar, click  to calibrate the cutting head.

- If the cutting head has been calibrated, click  **One Key Calibrate** on the **Follow-up Control** page.

The system automatically perform calibration, which take about 20s to complete.

After calibrating the cutting head, view it in the **Calibration Data** area of the **Follow-up Control** dialog box:

- If **Stability** and **Smooth** are higher than **Good**, continue normal machining.
- If **Stability** and **Smooth** are lower than **Good**, need to recalibrate.

2.2.3. Tube Edge Find

By moving the feed axis, the cutting head is positioned exactly above the tube section.

The process of edge finding is as follows:

1. The laser head follow-up axis follows to the following height once.
   Calibrate before use.
2. The cutting head move along the negative direction of the feed axis and stop when it reaches the edge of the tube.

   Caution

   After the automatic edge finding is completed, retract will be executed and the workpiece origin will be set automatically.

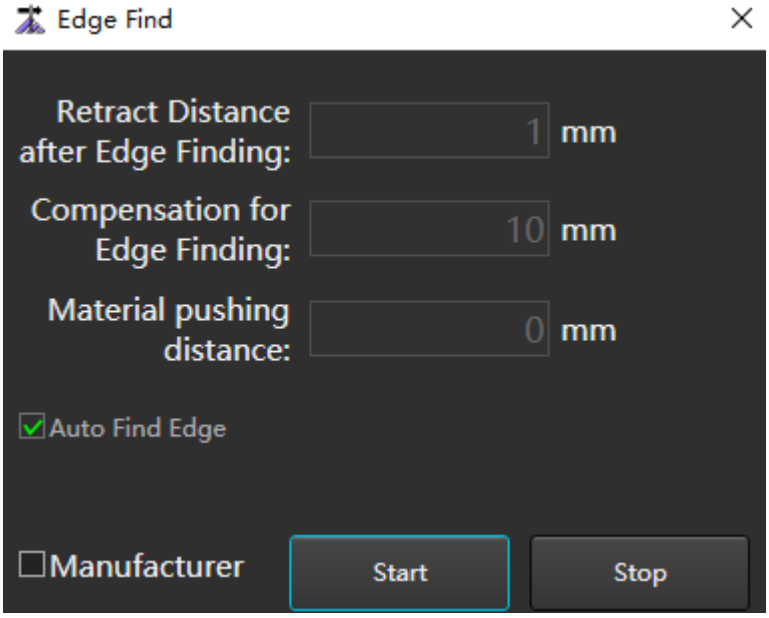
Operating Premise：

Make sure that the cutting head is completely above the tube.

Operating Steps：

1. In the machine control bar, click 

to open the **Edge Find** dialog box:



2. Check **Manufacturer** and set the following parameters：

Parameter	Description
Retract distance after edge finding	Retract distance after automatic edge finding. Range：-10~1000.
Compensation for edge finding	Compensation for edge finding range：-10~100.
Material pushing distance	To avoid no tubes when edge finding. Set the retract distance, that is, before edge finding, the distance of material pushing in the positive direction of the Y axis. After the movement is completed, turn on the follow-up, and the Y axis moves backward to start edge finding.Range：0~10000.
Height of edge finding	Range：0.5~10.

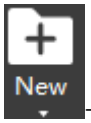
3. Click **Start**, if need stop to click **Stop**.

2.2.4. Set Tube Size

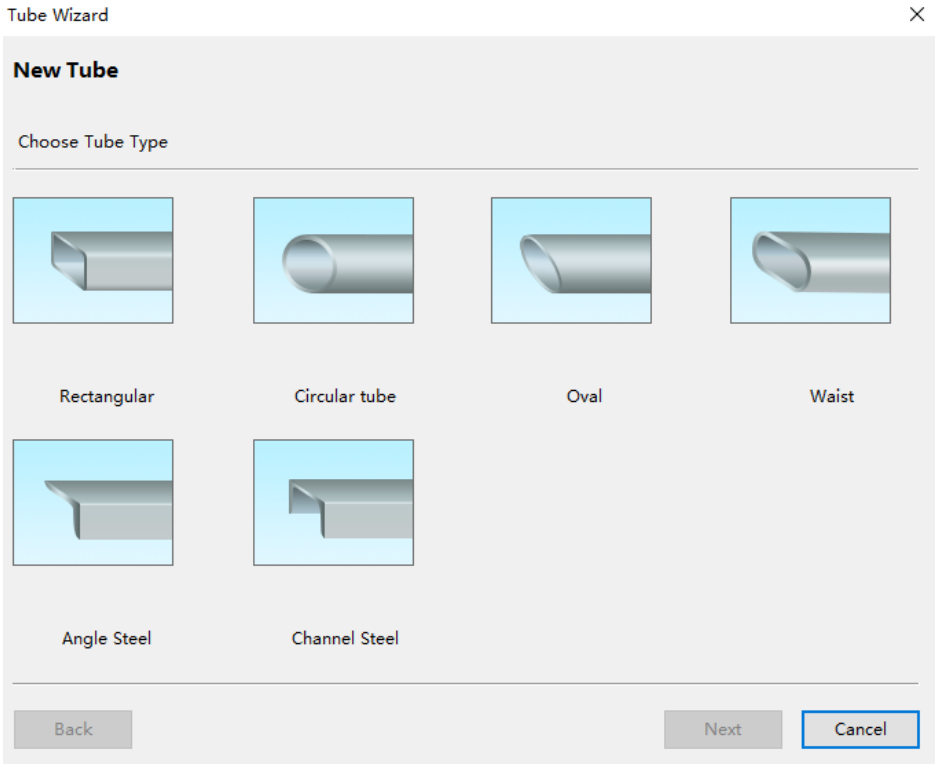
Set the type and size of the machining tube in the software according to the actual situation.

This chapter takes rectangular tube as an example.

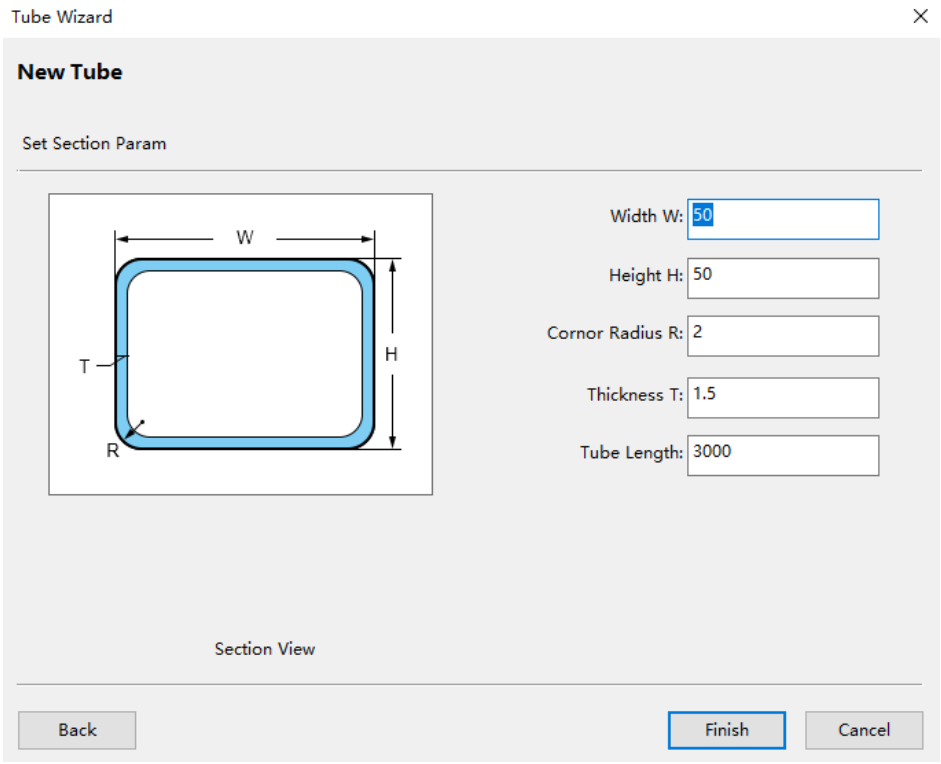
Operating Steps：

1. In the menu bar, click **Common** → 

→ **Tubes** to open the **Tube Wizard** dialog box：



2. Select **Rectangular Tube** to set the size of the new tube - rectangular tube：



When setting the R corner radius of rectangular tube, it can be about 0.5mm larger than the original radius. Prevent the offset of the tube size from causing poor cutting effect of some edges of the tube.

3. Click **Finish**.

Related Task：

To modify the tube size. For details, see Tube Cutting Set.

2.2.5. Mark Point

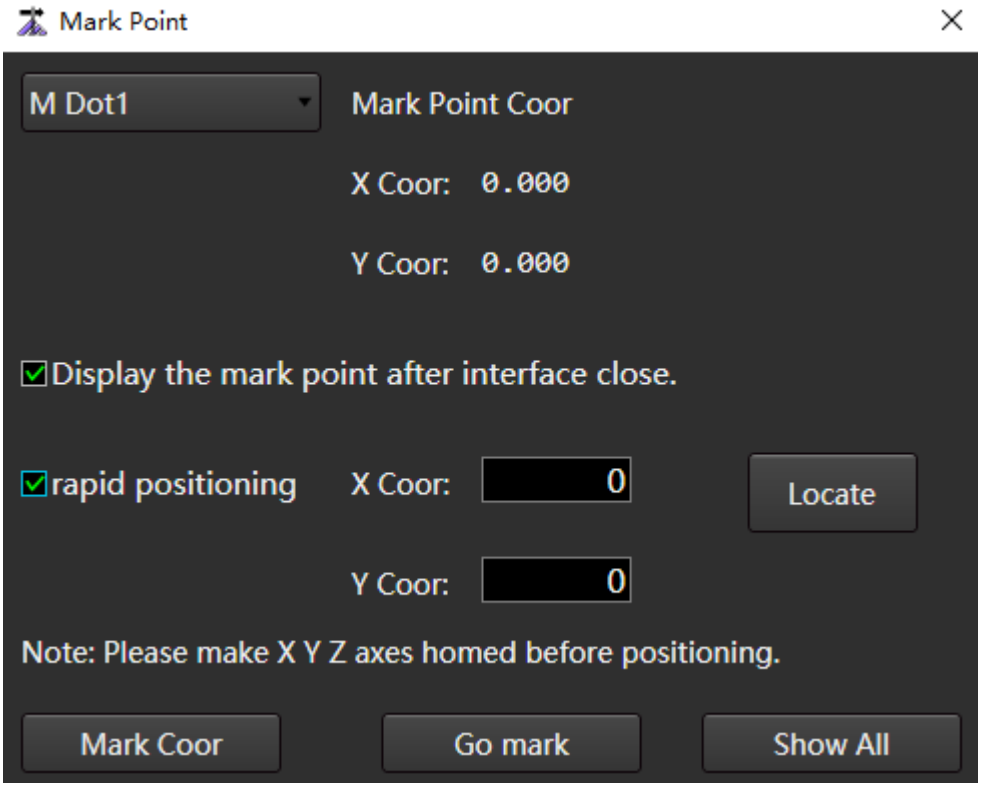
It is used to set the mechanical coordinate of the target position as the mechanical coordinate of the marked point, and move the cutting head back to the marked point when necessary.

Operating premise：

Make sure that each axis has returned to the machine origin.

Operating Steps：

1. In the menu bar, click **Common** → **Mark Point** to open the **Mark Point** dialog box：



2. In the machine tool control bar, click the X axis and Y axis direction buttons to move the cutting head to the target position.
3. Select **M Dot n**. The value range of n is 1~8.
4. Click **Mark Coor**.
5. After selecting the target position, click **Go Mark**, and the cutting head will automatically return to the marked point position.
6. **（Optional：）** If the target position is not a mark point and it is necessary to quickly locate to the specified mechanical coordinate position, check **Rapid Positioning**, set the coordinate value and click **Locate**.

Related Tasks：

In the **Mark Point** dialog box, you can also perform the following operations：

- If you need when to close the mark point dialog box, the drawing area still displays the mark points, check **Display the Mark Point after Interface Close**.
- To display all marked points in the drawing area, click **Show All**.

2.2.6. Set Common Parameter

Common parameters include machine param, user habit, gas param, burst param, follow-up control, clean by gas, and unit switch.

Operating Steps：

1. In the menu bar, click **Common** → 

to open the **Common Param** dialog box：

2. Check **Manufacturer** to activate **Machine Param**.
3. Set **Machine Param**, and the parameters are described as follows:

Parameter	Description
G00 speed	G00 speed of each axis during machine tool machining.Range: 1~100000.
G00 acc change time	G00 acceleration change time of each axis during machine tool machining.Range: 1~10000.
Max acceleration at corners	The maximum acceleration of feeding movement on adjacent axes, and the recommended value is 1~2 times of the acceleration.Range: 0.1~50000.
G00 acc	The maximum acceleration of each axis during machine tool machining.Range: 0.001~50000.
Feedrate acc	The total acceleration of the acceleration phase during machine tool machining.Range: 0.1~50000.
Feedrate acc change time	Change time of uniaxial acceleration during machining.

Parameter	Description
Rcircle max s	The maximum allowable speed corresponding to a reference circle with a diameter of 10 mm.Range：1~1000000.

4. Set **User Habit**, and the parameters are described as follows：

Parameter	Description
Do piercing at resume	Whether to enable do piercing at resume.
Additional actions after machining	It includes not move, return to marked point, return to fixed point, return to workpiece origin and other additional behaviors of X and Y axes after machining.
Outter fixed height cutting	When checked, the outer length is cut using the fixed height action.
Additional actions begin machining	Includes no operation and automatic clearing of workpiece coordinates.
Heavy tube speed	After checking, configure the tube heavy speed limit conditions： Make sure to add and set materials in Manage Material before configuration. 1. Click **Speed Limit Setting** to open the **Heavy Tube Speed Limit** dialog box. 2. Enter a value in the **Data Array** input box and click **Confirm**. 3. In the **Data** area. Configure speed limit strategy for different tube weights, acceleration and speed of Y axis and B axis. **Note：** Data area, the weight is required to be input in an incremental manner. 4. Click **Update**.
Set tube length	When configuring different tube length, it shall be in the chuck clamping position. Click **Set Tube Length** to open the **Set Tube Length** dialog box. For the setting method, see Set Tube Length.
Check ckstatus before machining	If checked, it will be checked whether the status of the front and rear chucks is consistent with that set in **Ckstatus Setting** before machining.
Ckstatus setting	Configure the chuck or unclamp action of the front and rear chucks for detection before machining. 1. Click **Ckstatus Setting** to open the **Ckstatus Setting** dialog box. 2. Check the chuck to be set, and Check **Chucking** or **Unclamp** according to the actual situation. 3. Click **Confirm**.

5. Set **Follow-up Control**, and the parameters are described as follows：

Parameter	Description
Leapfrog	Whether to enable leapfrog function.
Leaping Min. dis	When the distance is less than this value, leapfrog will not be carried out and the cutting head will not be lifted. Directly traverse to the next graphic starting point.

Parameter	Description
Follow Max. H	When the cutting height/perforation height is less than this value, it directly follows the set height; When the cutting height/piercing height is greater than this value, follow it to 1mm and then lift it to the set height.
Z up Pos.	After return to the mechanical origin, close the following or the mechanical coordinate position where the Z axis stop at the end of machining.
Fixed height	After the fixed height cutting is enabled, the follow-up is not opened during the cutting process, and the Z axis is fixed at a **Fixed Height Position**. You can move to the actual height and click **Gst Fixed Pos** or enter it manually.

6. Set **Gas Param**, and the parameters are described as follows:

Parameter	Description
Default gas type	Open the default gas used in the blowing port. When the user select oxygen blowing, the proportional valve can open the port.
Gas change delay	It is mainly used for progressive pierce and staged pierce. If the cutting gas is different from the pierce gas, the laser is not turned off during the gas switching process after the completion of piercing.
1st point blow delay	Blowing delay after machining start/breakpoint resume.
Blow delay	The blowing port is switched from closed state to open state, and the blowing delay will be executed.
P. in idle	The air pressure value for manual blowing in idle state.
Gas on distance	The maximum linear distance between two entities switch.

7. Set **Clean by Gas**, and the parameters are described as follows:

Parameter	Description
Clean by gas	Whether to enable clean by gas function. Clean by gas function is used to blow air before cutting, which is used to clean the nozzle and make the gas more sufficient in the pipeline to ensure the actual cutting quality.
Gas pressure	Percentage of air pressure used for gas cleaning.
Duration	The duration of gas cleaning.
Interval	The time interval between cleaning when the number of clean is greater than 1.
Times before start	Number of times the gas was cleaned when the machining was started.
Times in resume	The number of times the gas is cleaned when the breakpoint resumes.

1. Set **Burst Param**, and the parameters are described as follows:

Parameter	Description
Power	Set the laser intensity at burst.
Frequency	The frequency of light emitted by the pulse during burst.
Duty cycles	The duty cycle corresponding to burst.
Delay	Duration of laser on during burst.

2. Set **Unit Switch**, and the parameters are described as follows:

Parameter	Description
Speed	Optional: mm/s, mm/min, m/min
Acc	Optional: mm/s^2, m/min^2, $G(10m/s^2)$
Time	Optional: ms, s
Pressure	Optional: bar, psi, Mpa

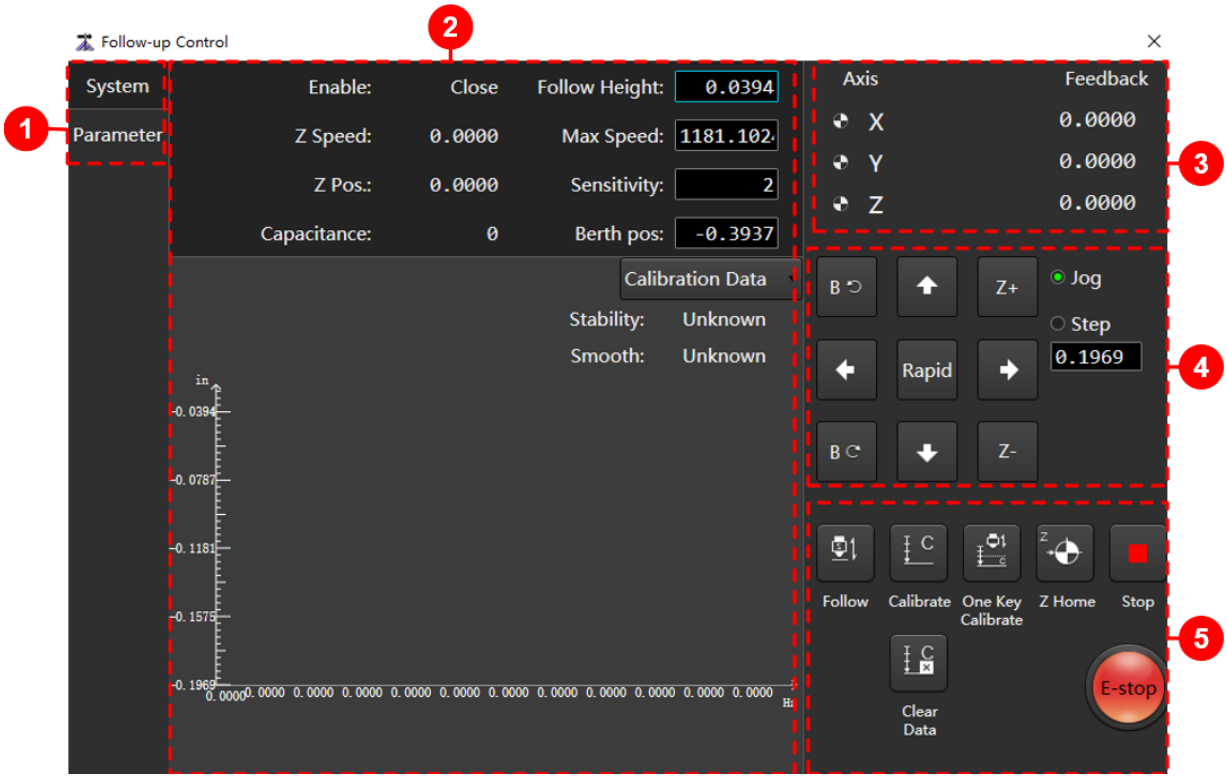
3. **（Optional：）** If you need to switch to system parameters, click **System Param** to open the **System Parameter** dialog box. For details, see System Parameter.
4. Click **Confirm**.

2.2.7. Follow-up Control

2.2.7.1. Overview

The corresponding relationship between capacitance value and distance is used to control the Z axis floating up and down in real time to ensure that the relative distance between the cutting head and the plate is always constant.

In the menu bar, click **Common** → Follow-up Control to open the **Follow-up Control** page:



1. Page Switch Area
2. Follow-up Control Area / Follow-up Parameter Setting Area
3. Coordinate Display Area
4. Manual Control Area
5. Follow-up Control Button

2.2.7.1.1. Page Switch Area

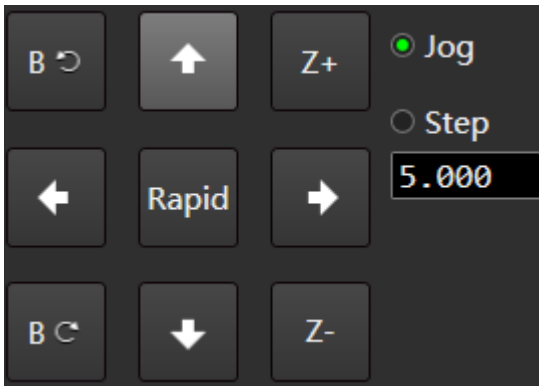
Page	Description
System page	Enter the follow-up control area.
Parameter page	Enter the follow-up parameter setting area.

2.2.7.1.2. Coordinate Display Area

The mechanical coordinates and workpiece coordinates of each axis are displayed.

2.2.7.1.3. Manual Control Area

Manually control the machine tool movement.

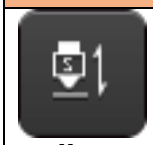


The manual control area includes:

Control button	Description
Axis direction button	Click the direction button corresponding to each axis to control the positive or negative movement of each axis of the machine tool.
Continuous rapid mode	▪ In the continuous low speed mode, click the **Rapid** button to highlight and switch to the continuous rapid mode. ▪ Press an axis direction button, the machine tool moves at low speed/high speed, and stop after releasing the button. ▪ Press several direction buttons at the same time, the selected axis moves at low speed/high speed at the same time, and it stop at the same time after releasing the button.
Step mode	Click the axis direction button, and the machine tool will stop after moving the set step.

2.2.7.1.4. Follow-up Control Button

Control the machine tool to perform follow-up related operations.

Control button	Description
 **Follow**	The system automatically runs the following function when there is calibration data. Click this button again, and the following will stop and the Z axis will stop automatically following when it returns to the berth position, and the Z axis will stop at the current position.
 **Z Home**	The Z axis will return to the mechanical origin. Click this button again to stop returning to the mechanical origin. After the execution of returning to the mechanical origin, a ▪ sign will appear in front of the Z axis.
 **Stop**	The system will stop the current movement and enter the idle state, which is a method for the system to interrupt the task normally in the follow-up control process.
 **E-stop**	System emergency stop.
 **Servo Calibration**	It is a special button under the speed loop control mode. The system automatically performs compensation and clear servo zero drift.
 **Calibration**	Perform the calibration of the cutting head. Click this button again to stop automatic calibration.
 **One Key Calibration**	The system will automatically calibrate the cutting head with one key.
 **Clear Data**	Perform data clearing, and the software will clear the calibration data and reset the touch part capacitance.

2.2.7.1.5. Follow-up Control Area

Include the following areas:

- Main Parameter Area
- Oscillographic Area

2.2.7.1.5.1. Main Parameter Area

The parameters displayed in this area are divided into:

- Real time monitoring parameters (values cannot be modified):

Parameter	Description
Enable	After click  **Follow**, the follow enable changes from closed state to open state, and this parameter changes from displaying **Close** to **Open**.
Z speed	Displays the current Z axis running speed.
Z Pos.	Displays the current Z axis mechanical coordinate.
Capacitance	Displays the current capacitance value. When the cutting head is closer to the plate, the parameter is smaller; When the cutting head touches the plate, this parameter is 0 (metal plate).

- Part of common follow-up parameters (values can be modified) :

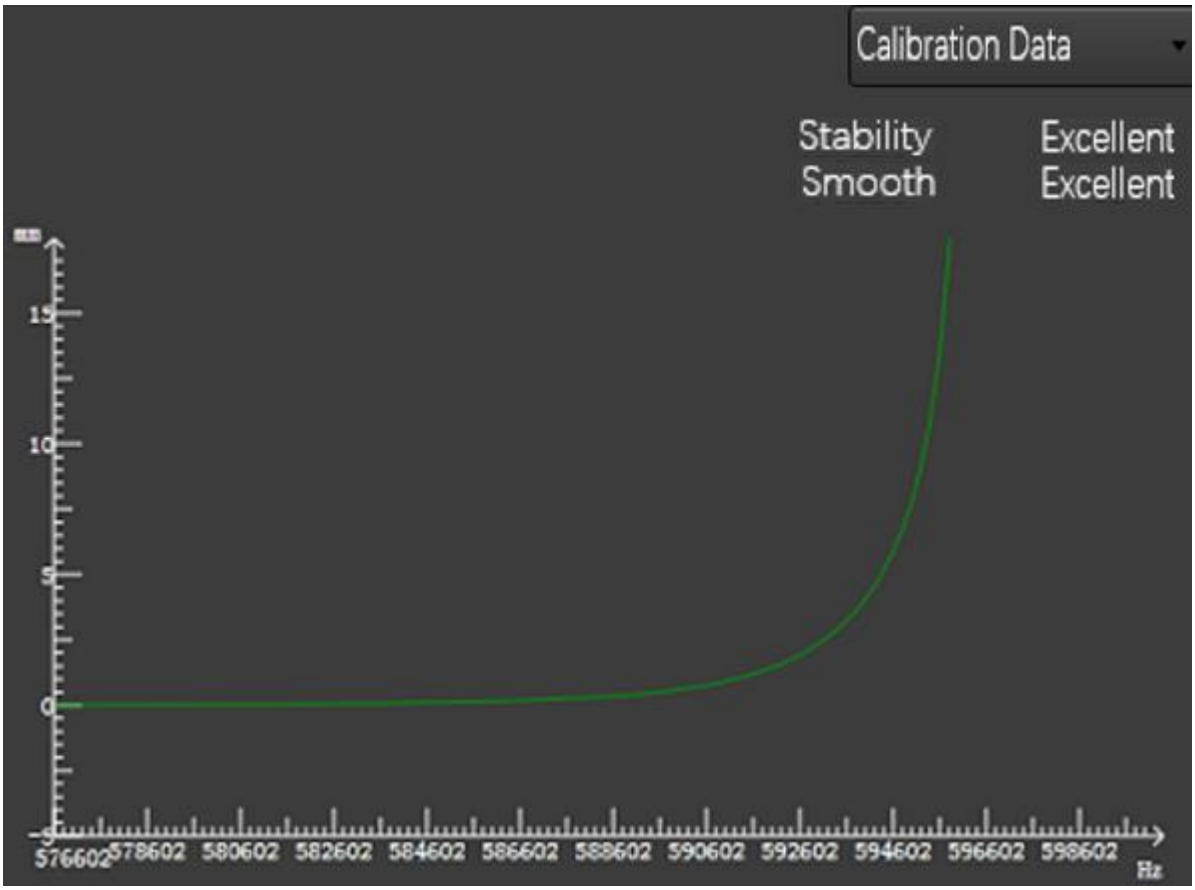
Parameter	Description
Follow height	The height of the cutting head when following.
Max speed	The maximum speed of the cutting head when following.
Sensitivity	Control the sensitivity of follow-up.
Berth pos	After returning to the mechanical origin, turn off the following or when the machining is finished, the berth position of the cutting head's mechanical coordinate

Click the current value of the parameter and enter the parameter value to be modified in the dialog box. For details, see Parameter.

2.2.7.1.5.2. Oscillographic Area

Click the **Calibration Data** button in the upper right corner to switch to the following curve page:

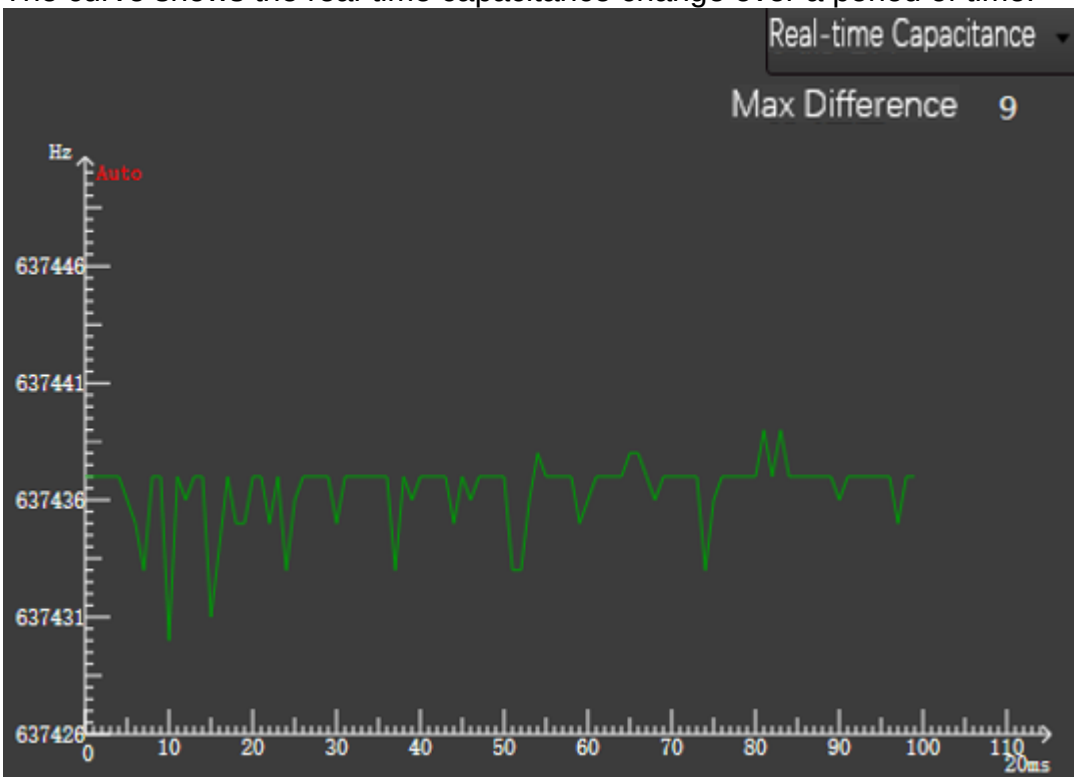
- **Calibration Data**
  The curve shows the corresponding relationship for the capacitance and position between the cutting head and the plate when the cutting head is automatically calibrated.



- X axis: Capacitance value.
- Y axis: The distance between the cutting head and the plate.

- **Real-time Capacitance**

The curve shows the real-time capacitance change over a period of time.



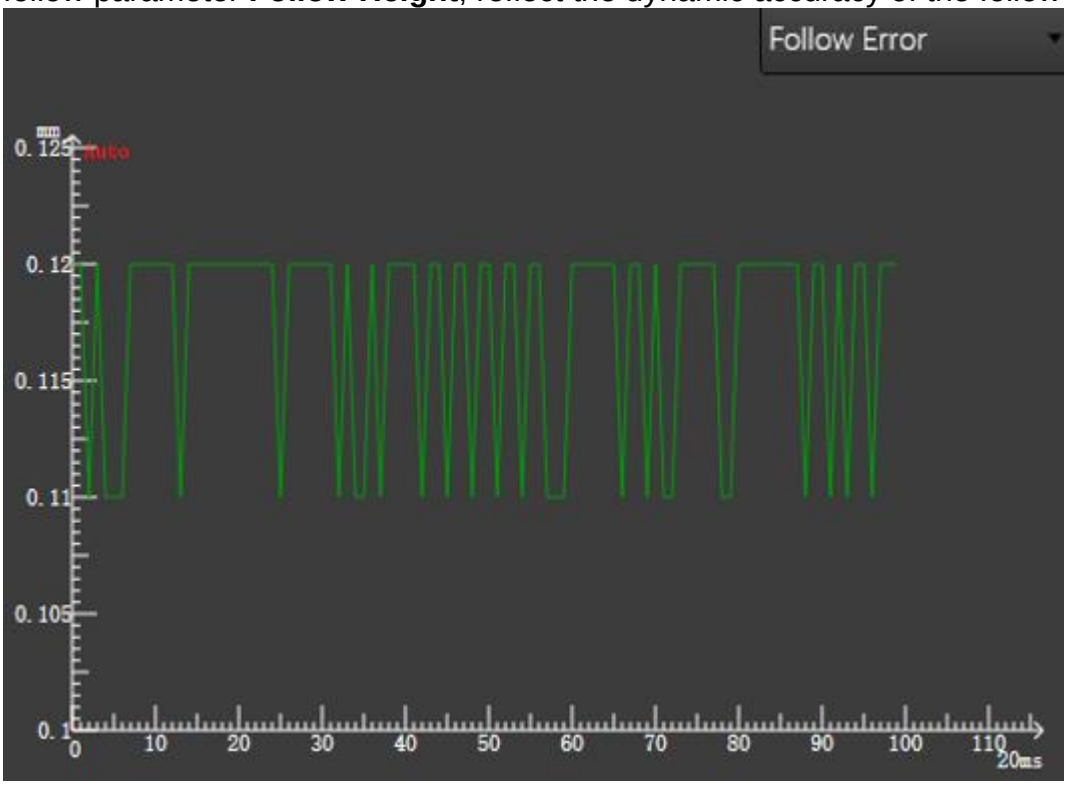
- X axis: Time.

- Y axis: Capacitance value.

While keeping the cutting head and plate stationary, observe the **Max Difference** in the upper left corner, which reflect the difference between the maximum and minimum capacitance during this period. Because the greater the value is, the greater the interference is, and the more unstable the capacitance measurement is, so **Max Difference** is not greater than 30 as the ideal value.

- **Follow Error**

The curve shows the difference between the current **Follow Height** and the set follow parameter **Follow Height**, reflect the dynamic accuracy of the follow effect.



On the **Follow Error** page, pause the waveform: Double click any point in the page to pause the waveform, and the Auto sign at the top of Y axis will become AutoII.

2.2.7.1.5.3. Follow-up Parameter Setting Area

Display all parameters related to servo control. For details, see Parameter.

Based on user permissions and identity, the follow-up parameters are divided into operator parameters and manufacturer parameters, and the system displays operator parameters by default.

Select the following method to open the **Parameter Setting** dialog box and enter the parameter value to be modified:

- After moving the cursor to the current value of the parameter, double click the left mouse button.
- Press the direction keys ↑, ↓, ←, → on the keyboard, and then press the **Enter** key.

To view or modify the manufacturer's parameters, check the manufacturer and enter the password.

2.2.7.2. Operation

After understanding the layout and application of the follow-up debugging operation interface, start follow-up debugging.

Before the follow-up debugging, do the following operations:

1. Execute Preparatory Project.
2. Detection Capacitance.

Follow up debugging according to the following steps:

1. Execute Servo Calibration.
2. Execute Automatic Calibration.

After debugging, Check Follow-up is successfully debugged or not.

2.2.7.2.1. Execute Preparatory Project

Operating Steps:

1. Check that the hardware is installed correctly.
2. Check that the drive parameters and follow-up Parameter have been set.
3. Move the Z axis in the positive or negative direction in the step mode, and observe whether the Z axis coordinates change the corresponding step length.
   Pay attention to the positive and negative changes. If they are inconsistent, repeat steps 1 to 2.
4. Check and ensure that there is no alarm in the software, and the **Current Capacitance** parameter on the interface has a value display.
5. Make sure that the Z axis direction is adjusted correctly.
6. Make sure that the **System Parameter** of the software, the basic movement and coordinate display are correct, and the Z axis can return to the mechanical origin correctly.

2.2.7.2.2. Detection Capacitance

Check the status of the capacitance sensor when the cutting head is stationary or running.

Operating Steps:

1. Contact the plate with the cutting nozzle and ensure that the current capacitance value is 0
2. Set the follow-up parameter **Z Axis Berth**.
3. Move the cutting head where the distance between the cutting head and the plate surface is more than 30mm, and keep the cutting head still to ensure that the current capacitance value is stable:
   If it is unstable, the current electrical interference is serious. For details, see Serious Electrical Interference.

2.2.7.2.3. Execute Servo Calibration

In non-bus control system, servo calibration solves the problem of servo motor zero drift caused by speed loop control.

Operating Steps:

1. In the manual control area, click the direction button to move the cutting head to the middle of the travel to prevent the cutting head from moving beyond the travel range.
2. Click  **Servo Calibration**, and the system will automatically generate the value of servo parameter **Servo Compensation Parameter**.
   At this time, the cutting head moves back and forth slightly to compensate.

2.2.7.2.4. Execute Automatic Calibration

Acquisition capacitance data matches the correspondence between capacitance and height.

Operating Premise：

- The follow-up parameter **Non-metal Calibration** has been set to the type of actual calibration material.
- Capacitance detection is 0.

Operating Steps：

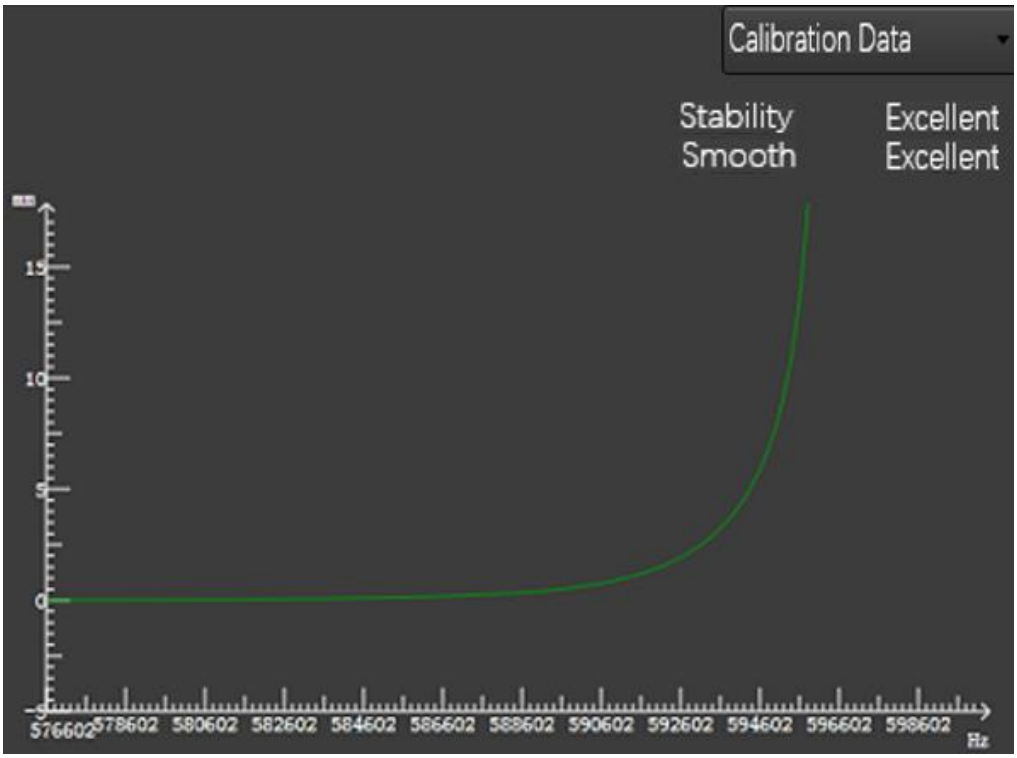
1. Move the cutting head to about 5mm near the plate surface and keep the plate surface still.
2. Click  **Calibration**, the system starts to calibrate, and takes about 20s to complete the calibration.
3. **（Optional：）** If the system needs to calibrate the cutting head automatically, click  **One Key Calibration**.

   Note

   If it is not calibrated, one key calibration cannot be used.

The automatic calibration process is as follows：

1. The cutting head slowly moves downward to detect and touch the plate.
2. After touching the plate, move upward for 5mm.
3. The cutting head moves down slowly for the second time to detect and touch the plate.
4. After touching the plate, slowly move up the set calibration distance and acquire the calibration data to obtain the calibration curve：



After calibration, the system automatically estimates the stability and smooth of the calibrated curve:

Index	Description
Stability	It refers to the data difference between falling 5mm to touch the plate and lifting 5mm to touch the plate. The greater the difference, the worse the stability. If the stability is **Poor**, it may be that the vibration is large or the external interference is strong, and it needs to be recalibrated.
Smooth	Refers to the smoothness of the curve. If the smooth is **Poor**, it means that the curve is not smooth, has undulations or burrs, and needs to be recalibrated.

Caution

During calibration, pay attention to using the E-stop button in time to prevent damage to the machine tool caused by continuous pressing of the cutting head when the touch part capacitance is incorrect.

2.2.7.2.5. Check Follow-up

Operating Premise：

Successfully calibrated.

Operating Steps：

1. During the **Follow-up Control** operation, ensure that the cutting head is not oscillated and the following distance is correct.
   After that, you can use a screwdriver or a small piece of iron to move back and forth under the cutting head, and observe whether the cutting head move up and down

according to the position of the screwdriver or the small piece of iron, and whether the cutting head is ossification.
If oscillation, reduce the positioning sensitivity and increase the inposition tolerance parameter to suppress oscillation.

2. In the menu bar, click **Set** → 

to find the system parameter **Out Margin Check** and set it to **Yes**.
Enable this feature can effectively improve security.
3. Draw the tool path, machine without turn on the laser, and observe whether the cutting head is oscillation during the following process.
If oscillation, reduce the positioning sensitivity and increase the inposition tolerance parameter to suppress oscillation.

2.2.7.3. Parameter

All follow-up parameters and their descriptions on the **Parameter** page of the **Follow-up Control** dialog box:

- Setting
- Follow Setting
- Follow-up
- Calibration Setting
- Speed Setting
- Real-time Statue Check
- Manual Speed

2.2.7.3.1. Setting

Parameter	Description	Range	Value
Axis direction	Specifies the direction in which the mechanical coordinate value of the Z axis increases. When manually operating the machine tool, if the axis movement direction is opposite to the direction determined by the **Right Hand Rule**, modify this parameter.	▪ 1：Positive direction. ▪ -1：Negative direction.	1
Pulse equivalent	The displacement or angle generated by each control pulse on the Z axis.	0.000001mm/p~999mm/p	0.001mm/p
Workbench stroke upper limit (Z)	Upper limit of soft limit value.	-1000mm~99999mm	0
Workbench stroke lower limit (Z)	Lower limit of soft limit value.	-99999mm~0mm	-1000

Parameter	Description	Range	Value
Screw pitch (Z)	The screw pitch in the Y axis direction.	0mm~360mm	10
Coarse positioning direction (Z)	In the process of returning to the mechanical origin, the motion direction of the Z axis coarse positioning stage.	▪ 1：Positive direction. ▪ -1：Negative direction.	1
REF switch positioning speed	In the process of returning to the mechanical origin, the feed speed of the Z axis coarse positioning stage.	0.1mm/min~10000mm/min	1800
Retract distance (Z)	The additional travel distance of Y-axis after the end of the fine positioning stage returning to the mechanical origin. A positive value means go back in, a negative value means go out, and a value of 0 means not move.	-100mm~1000mm	2

2.2.7.3.2. Follow Setting

Parameter	Description	Range	Value
Stand-off distance	The relative distance between the cutting head and the plate during follow-up control.	0mm~30mm	1
ZUP position	After return to the mechanical origin, turn off the following or the mechanical coordinate where the Z axis berth at the end of machining.	-100mm~100mm	-10
Safe lift height	The safe height for lift when the Z axis does not return to the mechanical origin.	0mm~100mm	40
Follow-up to the max height directly	Follow the maximum height directly.	0.01mm~16mm	5

2.2.7.3.3. Follow-up

Parameter	Description	Range	Value
Positioning sensitivity	Control the sensitivity of follow-up positioning movement.	1~20	8
Follow-up sensitivity	Control the sensitivity of follow-up.	1~5	1
Follow-up feed forward	In order to speed up the following, within a certain range, the larger the value, the faster the response speed. If the feed forward is too large, the follow up will follow the oscillation.	0~100	50

Parameter	Description	Range	Value
Inposition tolerance	When it is detected that the height is **Follow height ± Inposition Tolerance Value**, it is considered that the follow-up is in place.	0mm~655mm	0.3
Vibration suppressing level	The higher the oscillation suppression level is, the stronger the suppression effect on the occurrence of plate oscillation in the machining process is, and the single follow-up sensitivity will be correspondingly reduced.	0~5	1
Servo compensation parameter	Special parameters in speed loop control mode. The value generated after servo calibration.	-255~255	0
Part touching delay（positioning）	Touch part delay during positioning control.	0ms~20000ms	300
Part touching delay（follow-up）	Touch part delay during follow-up following status.	0ms~20000ms	500
Part touching delay（pierce）	Touch part delay during pierce.	0ms~20000ms	600
Enable anti-collision	When it is enabled, the cutting head is automatically lifted up when it is detected that the cutting head may collide during dry running.	▪ Yes： open ▪ No： close.	Yes
Speed gain	Special parameters in speed loop control mode. The rated power of the motor divided by 10V matches the speed command input gain in the motor.	10~2000	300
Capacitance threshold to trigger cutting head alarm	Threshold value of capacitance to trigger exception alarm in cutting head.	100Hz~100000Hz	500
Additional capacitance threshold to trigger cutting head alarm	When the capacitance compensation is turned on, the threshold value of capacity to trigger exception alarm in cutting head will be added with additional detection tolerance.	100Hz~100000Hz	1500
Capacitance compensation	Whether capacitance compensation is carried out.	▪ Yes： open ▪ No： close.	No

2.2.7.3.4. Calibration Setting

Parameter	Description	Range	Value
Enable non-metal calibration	Whether non-metal calibration is used.	▪ Yes: Automatic calibration of metal materials. ▪ No: Automatic calibration of non-metal materials, such as wood, plastic, etc.	No
Touch part capacitance	The capacitance value marked by frequency when touching part.	0Hz -1000000Hz	0
Calibration length	During calibration, the capacitance data within this range shall be recorded. When the Z axis travel is short, the parameter value can be appropriately reduced.	5mm~50mm	18
Part touching speed	When calibrating, the speed of touch part movement.	0mm/min~10000000mm/min	80
Calibrating speed	Calibration speed.	0mm/min~10000000mm/min	80
Capacitance fluctuation detection	When the capacitance fluctuation is less than the threshold value every 1mm during calibration, the calibration process is interrupted.	-	30

2.2.7.3.5. Speed Setting

Parameter	Description	Range	Value
Z G00 speed	The speed of downward and upward movement of the floating head. When the dry running speed is set to a large value, the calibration length needs to be increased so that there is enough deceleration area when following the downward direction to avoid collision with the plate.	Maximum speed of 0~axis	15000
Follow acceleration	Follow the acceleration.	1000mm/s^2~50000mm/s^2	12000
Axis maximum speed （Z）	The following speed and dry running speed cannot be greater than this speed.	1mm/min~100000mm/min	30000

Parameter	Description	Range	Value
Max speed of Z axis motor	Maximum speed of Z axis motor.	1000r/min~20000r/min	6000

2.2.7.3.6. Real-time Statue Check

Parameter	Description	Range	Value
Detect out-margin	Whether or not to enable detect out target at any time, and stop moving when it encounters out-margin.	▪ Yes: Detection ▪ No: No detection.	Yes
Empty leapfrog detection tolerance	Empty leapfrog detection tolerance.	0mm~225mm	3

2.2.7.3.7. Manual Speed

Parameter	Description	Range	Value
Manual high (Z)	Acceleration of Z axis in manual mode.	0mm/s^2~100000mm/s^2	5000
Manual low (Z)	The speed of Z axis during high speed operation in manual mode.	1200mm/min~30000mm/min	1800
Manual continuous low (Z)	Default speed of Z axis in manual mode.	0.1mm/min~1800mm/min	1200

2.2.7.4. Common Problem

Through this section, you can view the problems encountered in the follow-up debugging process and their solutions.

2.2.7.4.1. Serious Electrical Interference

Reason

- The position of the servo driver affects the electrical interference.
- The shielding layer is damaged or wound to the external iron frame.
- There is not conductive between pin 4 of M16 three core aviation plug drag chain cable and amplifier.
- There is a gap between the amplifier of the follow-up instrument and the machine tool.
- Radio frequency cable is damaged.
- The machine tool has poor contact with the ground.

Solution

- **Methods for Physically Eliminating Interference**
  - Make sure that the servo driver, lambda 5E controller and EX33A expansion terminal board are in good contact with the ground.
    If the contact is poor, drive the ground pile again.
  - Ensure that the cable shield is intact.
    If not, replace the cable shield.
  - Ensure that pin 4 of the M16 three core aviation plug drag chain cable is conductive to the amplifier.

If not, replace the cable.
- Make sure that the amplifier of the follow-up instrument is in full contact with the machine tool.
  If not, polish the veneer with sandpaper to remove the oxide layer before installing the amplifier.
- Test whether the radio frequency cable is intact through the multi-meter.
  If not, replace the radio frequency cable.
- Make sure that the machine tool is in good contact with the ground.

2.2.7.4.2. There is a Deviation Between the Set Follow Height and the Actual Follow Height

Reason

The ceramic ring or nozzle was not calibrated when it was replaced, or it was not firmly installed. The capacitance fluctuates greatly during blowing, resulting in a certain deviation of capacitance curve.

Solution

Follow the steps below to troubleshoot the problem:

1. Ensure the ceramic ring or nozzle is securely installed.
2. Ensure that the capacitance fluctuation is within the set compensation range when blowing.

If the above is normal, recalibrate. For details, see Execute Servo Calibration and Execute Automatic Calibration.

2.2.7.4.3. Capacitance Feedback is Normal, Calibration Result is Good, but Cutting Head Often Stops Working

Reason

It may be that the external force generated by the gas of the cutting head causes the poor contact between the internal contact of the ceramic ring and the signal port of the cutting head, triggering the touch part alarm, so that the cutting head will stop working when the nozzle has no direct contact with the plate during the cutting process.

Solution

Replace the qualified ceramic ring.

2.2.7.4.4. When Cutting Thin Plate, the Cutting Head Oscillation Seriously, which Leads to Deformation of the Cutting Workpiece Contour

Reason

When cutting thin plates, due to the large cutting air pressure, the plate oscillation greatly during the cutting process. Therefore, it is necessary to reduce the parameter **Positioning Sensitivity** and increase the parameter **Inposition Tolerance** to suppress oscillation.

Solution

Adjust the parameter **Oscillation Suppression**.

2.2.7.4.5. When Moving Z Axis or Directly Turning On Follow-up, the System Gives an Alarm of "Follow-up Error Status"

Reason

- The superposition of the reverse running direction of the Z-axis motor and the **Offset Phenomenon** caused by external interference.

- It is only caused by **Offset Phenomenon**.
  The simple determination method of **Offset Phenomenon** is as follows:
  a. Switch on the servo.
  b. Open the cutting software to enable the servo, and observe the servo driver display interface.
     If the value changes back and forth and the amplitude is relatively large, it indicates that the external electrical interference is relatively large.
  c. Observe that the coupling at the connection between the Z axis motor and the lead screw rotates back and forth slightly.

Solution

Recalibrate. For details, see Execute Servo Calibration and Execute Automatic Calibration.

2.2.7.4.6. Error in Encoder Direction or Axis Direction

Reason

Error in encoder direction or axis direction parameter setting.

Solution

Do the following:

- Modify the encoder direction and observe whether the alarm is released.
  If it is not released, modify the encoder direction to the value before setting, and change the rotation direction of the driver parameter axis.
- If the axis direction and encoder direction are both reverse, set the driver parameter axis rotation direction machine encoder to the reverse value.

2.2.7.4.7. Follow-up in Place Waiting Timeout

Reason

- The parameter **Inposition Tolerance** is set too small.
- Calibrate the data difference.
- During machining, it is affected by external slag spraying.
- Follow-up overshoot.

Solution

Do the following:

- Check whether the inposition tolerance is set too small.
  The recommended value is 0.1.
- Recalibrate. For details, see Execute Servo Calibration and Execute Automatic Calibration.
- Adjust the cutting technic.
- Ensure that the follow-up parameters and servo driver parameters are set correctly.
  If not, reset the servo driver parameters.

2.2.7.4.8. The Following Error is Too Large

Reason

The following error is greater than the set out margin tolerance value within a time.

Solution

Do the following:

- If the error is reported on a flat surface, it may be caused by the follow-up overshoot. Check whether the servo driver gain is too small.
  If it is too small, increase the servo driver gain.
- If an alarm occurs during climbing, the follow-up sensitivity may be set too small.

2.2.7.4.9. When the System is Idle or in the Process of Machining, Turn On the Following Touch Part Alarm

Reason

The capacitance shall not be greater than that of the touch part plate.

Solution

Follow the steps below to troubleshoot the problem:

1. Ensure that the set value of the bumper plate capacitance is appropriate.
   The default value of 0 is recommended.
2. If follow is on, touch part alarms:
   a. Ensure that the parameters **Pulse Equivalent**, **Feedback Pulse Number** and **Speed Gain** are correct.
   b. Make sure that the drive gain is correct.
3. Touch part alarm during machining:
   a. Make sure the manual follow is normal.
   b. Ensure that the capacitance fluctuation range is within 50 when blowing.
4. If the above is normal, the servo driver gain may be small, and increase the servo driver gain appropriately.

2.2.7.4.10. Touch Part Alarm when the System is Static

Reason

The capacitance value when the capacitance is less than or equal to the touch part.

Solution

Follow the steps below to troubleshoot the problem:

1. Ensure that the body capacitance value and touch part capacitance are correct.
   If not, replace the body capacitance value and touch part capacitance.
   For SE001 above V1.4, the normal value of the body capacitance is about 650000.
   The capacitance of SE0001 in V1.4 and below is about 1.3 million.
2. Use a multi-meter to measure whether the copper core from the nozzle to the cutting head sensor is conductive.
   If it is not conductive, it indicates that there is a problem with the cutting head.
3. Use a multi-meter to measure whether the nozzle is conductive to the RF cable copper core.
   If it is not conductive, it indicates that there is a problem with the RF cable. Replace the RF cable.
4. Measure whether the resistance between terminals 1-2 of SE001 is 4.8~5.3K Ω (the error range is allowed to be ≤ 5%), and whether the resistance between terminals 2-4 is 0 Ω~1 Ω.

If the resistance value is abnormal, SE001 is damaged. Replace SE001.

1. Measure whether the corresponding pins of M16 three core aviation line are conductive.
   If not, replace the cable.
2. If all the above are normal, replace the EX33A expansion terminal board.

2.2.7.4.11. Follow Overshoot

Reason

The servo response cannot follow the command speed.

Solution

Follow the steps below to troubleshoot the problem:

1. Ensure that the parameters **Pulse Equivalent**, **Speed Gain**, **Pulses Per Revolution** are set correctly.
2. Increase the servo driver gain.
3. Ensure that the maximum speed supported by Z axis matches the dry running speed, and the dry running speed of Z axis can be appropriately reduced.

2.2.8. Focus Control

Cutting different tubes requires different focus; In order to ensure the quality of machining, different focal points should be used during piercing and cutting. **The Focus Control** function is used to automatically adjust the focus during machining.

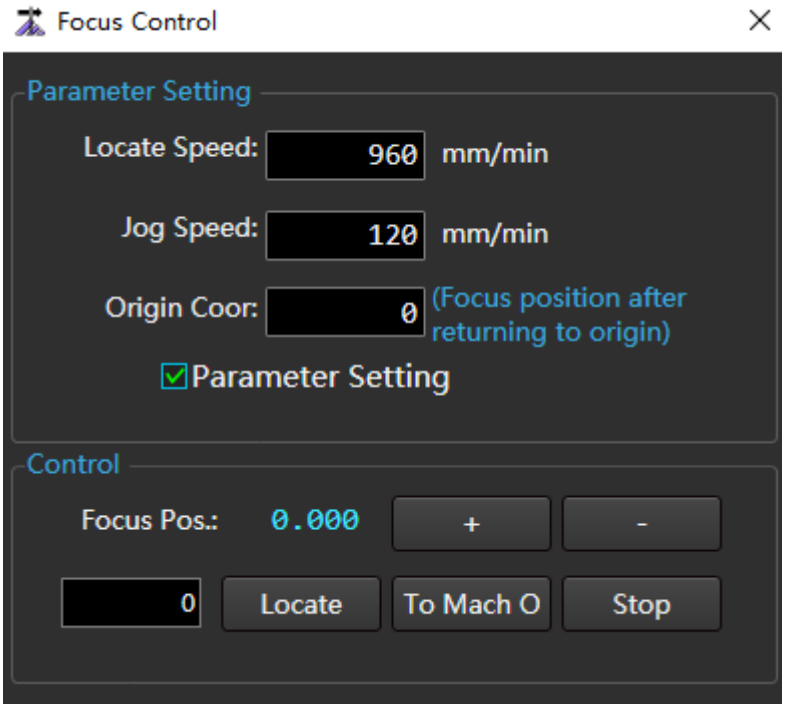
In the actual machining process, you can click **Layer Setting** and set the cutting parameter **Cutting Focus** or piercing parameter **Focus** in the **Layer Setting** dialog box to use this function.

Operating Premise:

Make sure that the system parameter **Enable Focus Control** is set to **Yes**, and restart the software to make the setting take effect.

Operating Steps:

1. In the menu bar, click **Common** → Focus Control to open the **Focus Control** dialog box:



2. Check **Parameter Setting** and set corresponding parameters.
3. Click the following buttons to control the action of the machine tool:

- **+** / **-**: Adjust the focus position with **Jog Speed**.
- **Locate**: Locate to the focus position set in the left input box at **Positioning Speed**.
- **To Mach O**: The W axis returns to the mechanical origin.
- **Stop**: The W axis stops moving.

4. Click **Confirm**.

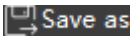
3. Loading or Drawing Tool Paths

3.1. Overview

Before machining, the tool path needs to be loaded or drawn.

After loading or drawing a graphic, you can perform the Edit Path File operation.

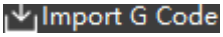
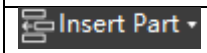
After operating the graph, you can select the following saving operation in the menu bar to save it as a tool path file in the format of .ncexa:

- Save the tool path file, click  .
- To save the new tool path file, click  .

Loading Tool Path

Drag the tool path file or nesting result file to be loaded to the software drawing area for loading, or use the following buttons to load:

Note: If the nesting result file (ncexa format) is opened, the nesting result list will be expanded and the nesting results will be added to the nesting list. Open other files without nesting results, they will be added to the base layer of the nesting list, and the nesting results list will not be automatically expanded.

Button	Description
	New tool path file, support: ▪ New Tube ▪ 3D Wrapping▪ Standard Part ▪ Import 3D Part
	Import tool path files in the format of .igs, .iges, .step, .stp.
	Open the tool path file in the format of .ncex, .ncexa.
	Import the G code format tool path file in the format of .nc, .g.
	Insert a part on the spare tube in the tool path file without covering the tool path. support: ▪ Insert File▪ 3D Wrapping▪ Standard Part
	Save tool file as .ncexa format file.

Draw Part

Select the following methods to draw the part:

- Import 3D Part.
- Draw graphic manually. For details, see Draw Graphic.
- Generate tool path graphic through tube intersection function. For details, see Tube Intersection.
- Generate tool path graphic through tube cutting function. For details, see Tube Cutting.

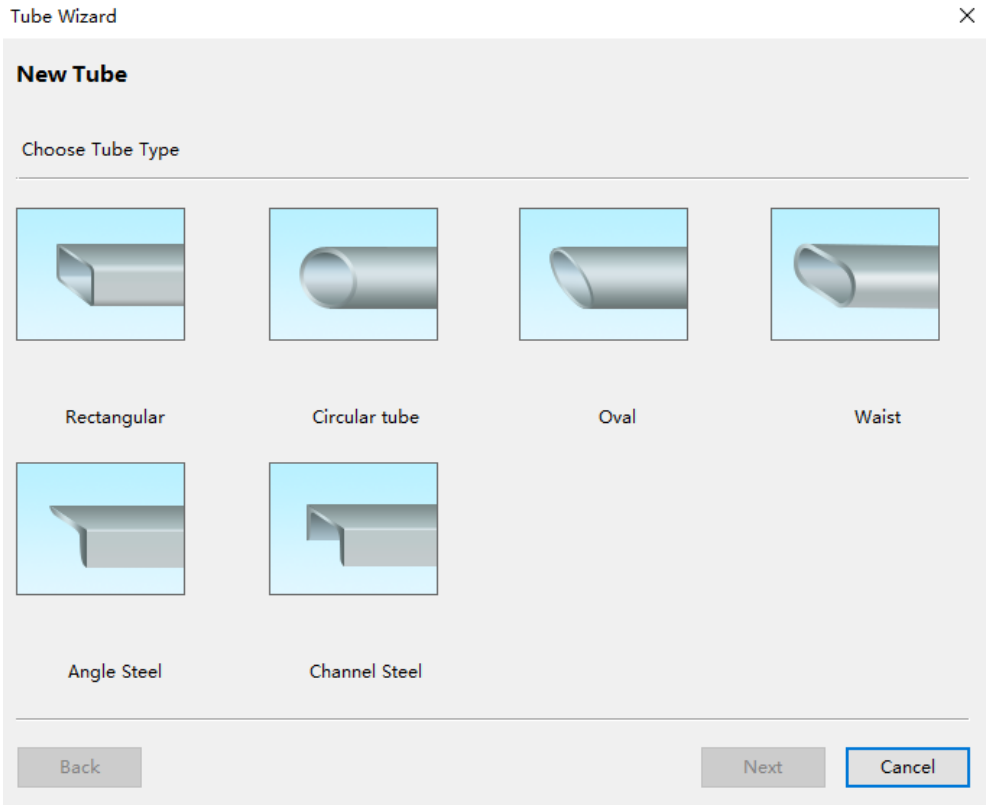
3.2. New Tube

Create a new standard tube.

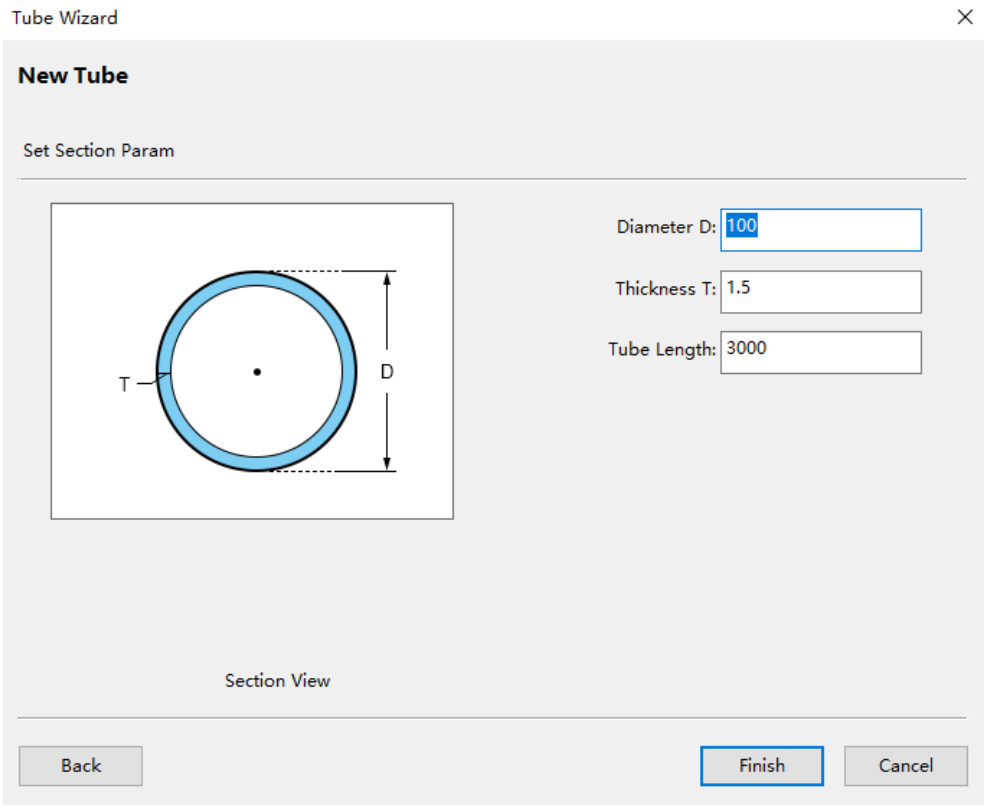
Operating Steps：

1. In the menu bar, click **Common** → 

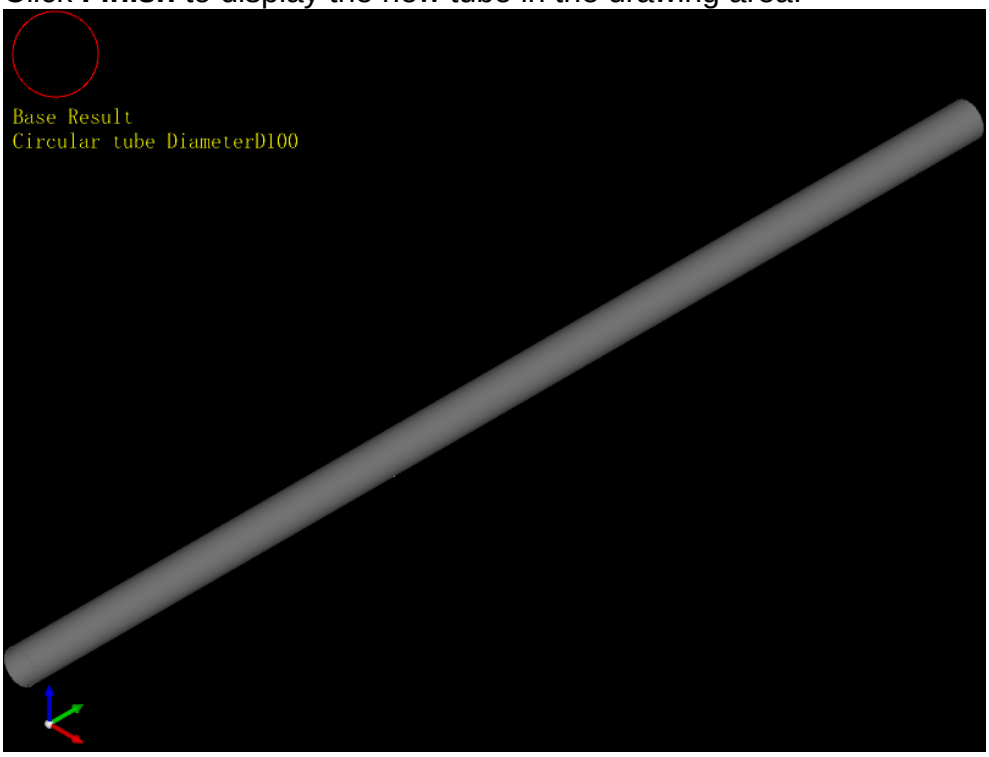
→ **Tubes** to open the **Tube Wizard** dialog box：



2. Click the corresponding tube type to open the **Set Section Param** dialog box：



3. Set the section parameters. The left diagram shows the meaning of the parameters.
4. Click **Finish** to display the new tube in the drawing area.



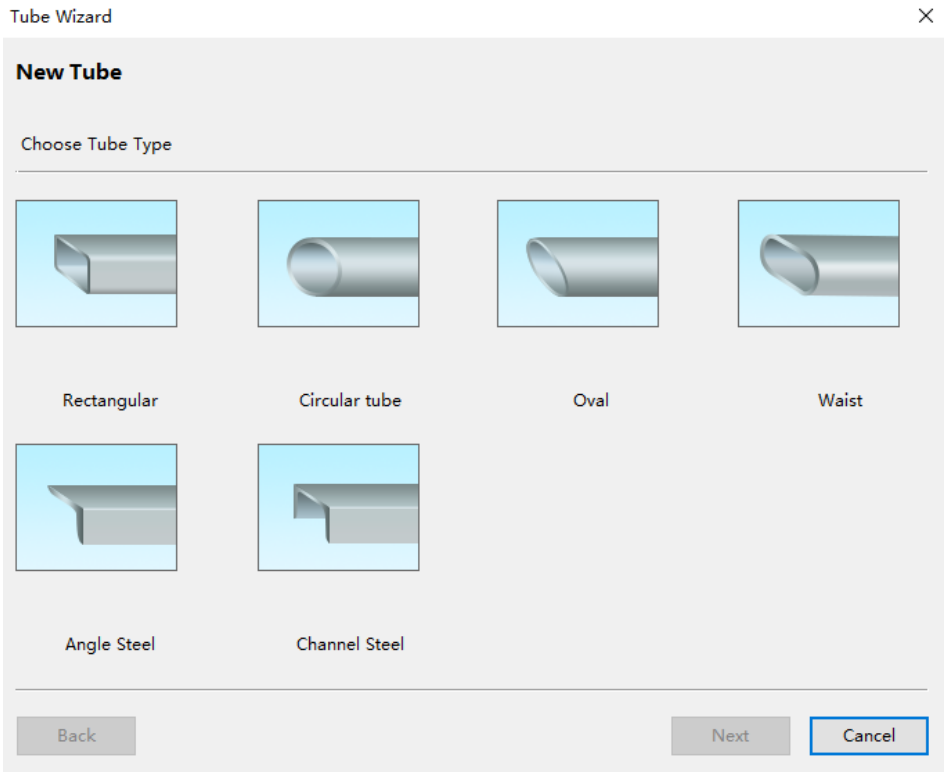
3.3. 3D Wrapping

The software support calling six standard tube types and configuring wrapping graphics. The wrapping graphics file support the following formats: .g, .nc, .dxf, .dwg, .plt.

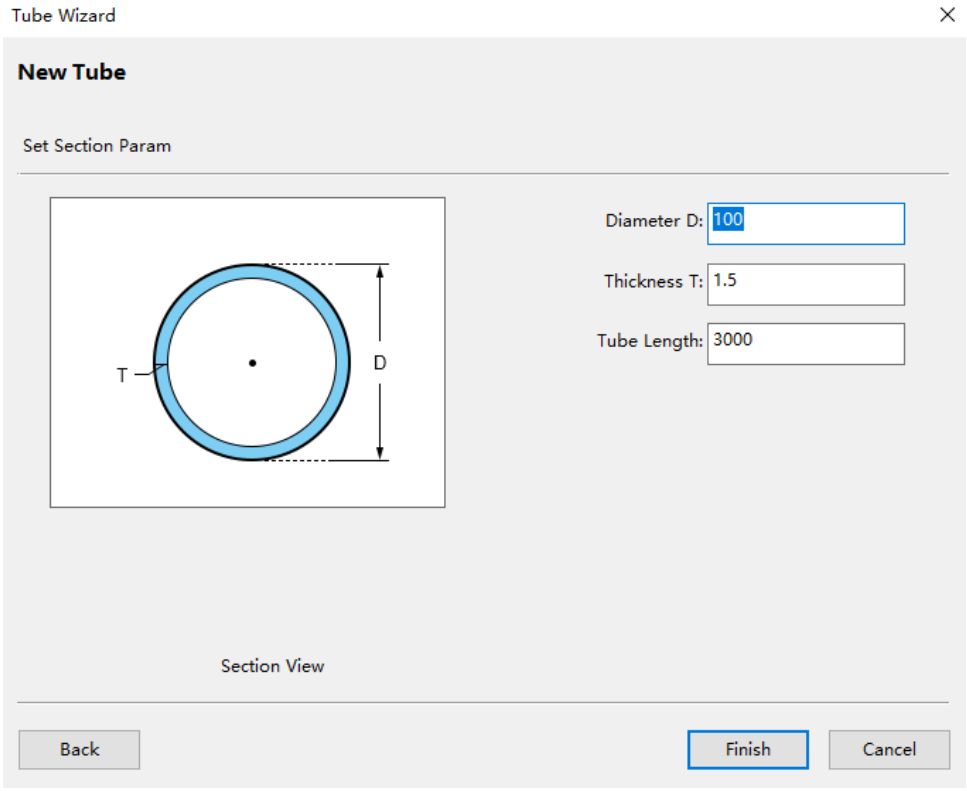
Operating Steps：

1. In the menu bar, click **Common** → 

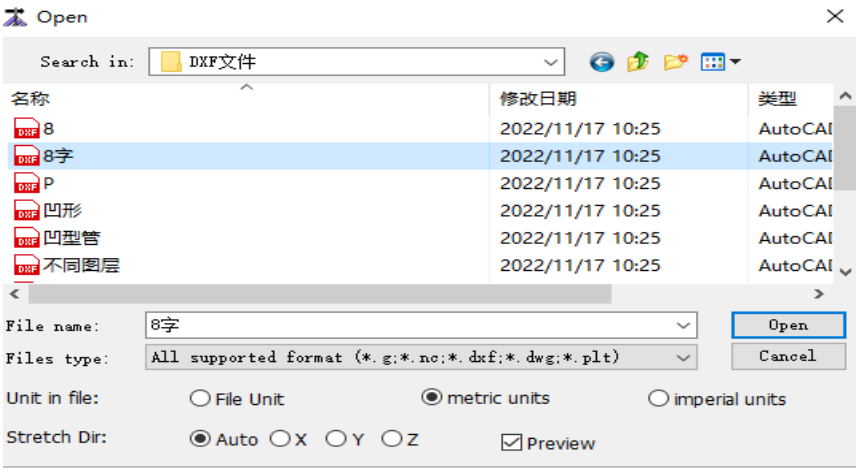
→ **3D Wrapping** to open the **Tube Wizard** dialog box:

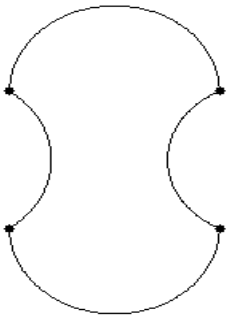


2. Click the corresponding tube type to open the **Set Section Param** dialog box:

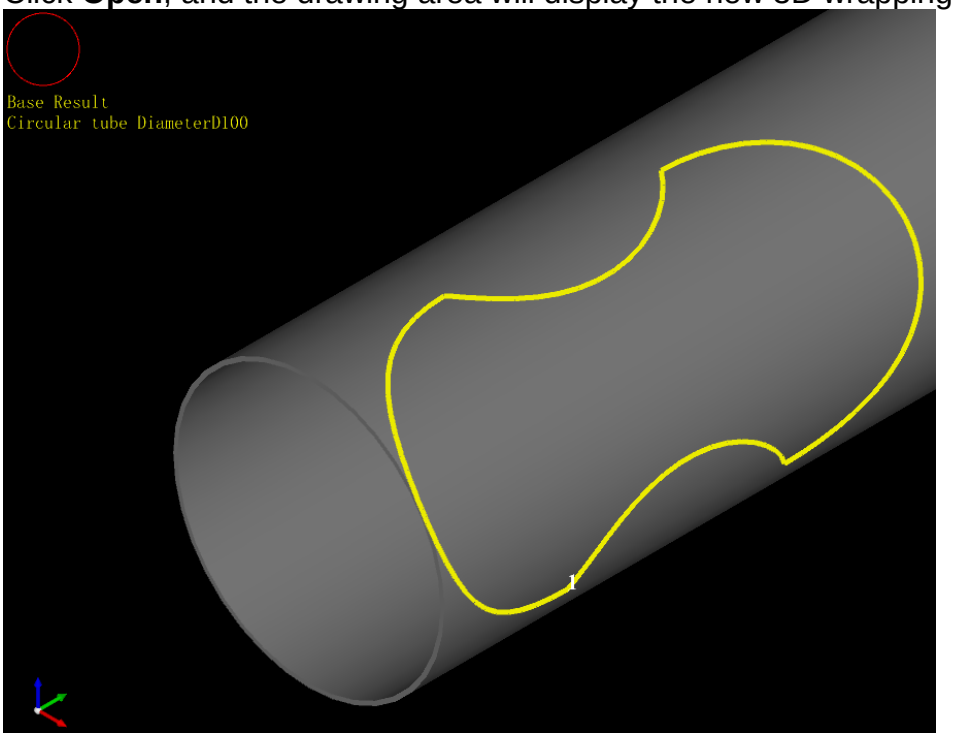


3. Set the section parameters. The left diagram shows the meaning of the parameters.
4. Click **Finish** to open the file selection dialog box.





5. Select the wrap file.
6. Set unit in file.
7. Sets the stretch direction.
8. Click **Open**, and the drawing area will display the new 3D wrapping tube.



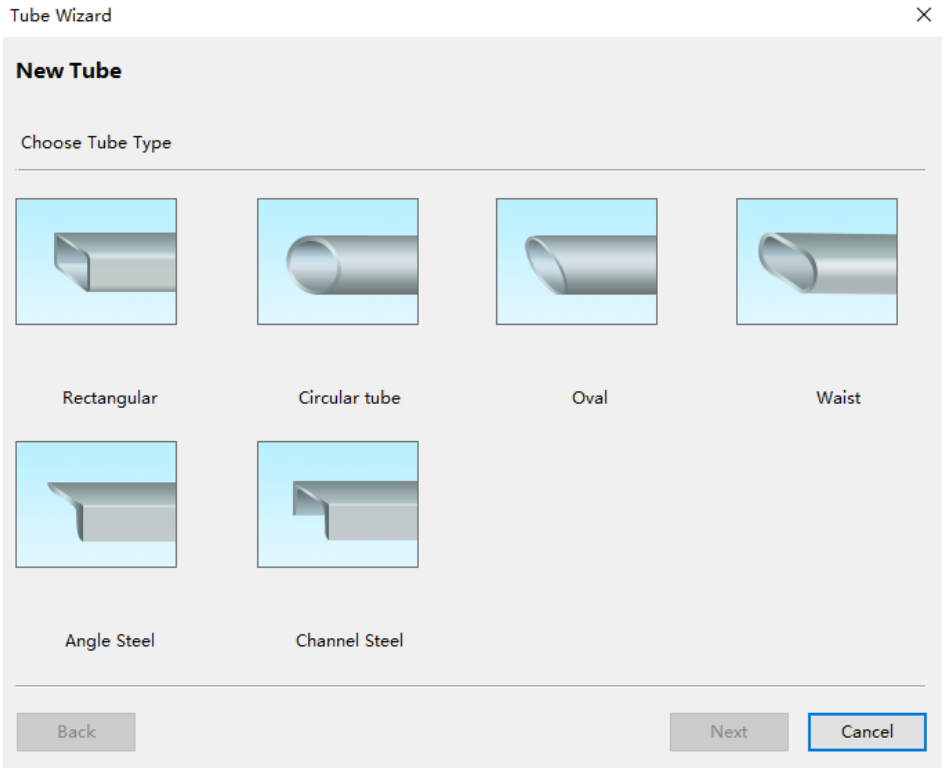
3.4. Standard Part

The software come with 6 commonly used standard tube type parts, which can be created by wizard to support users to use and modify part parameters.

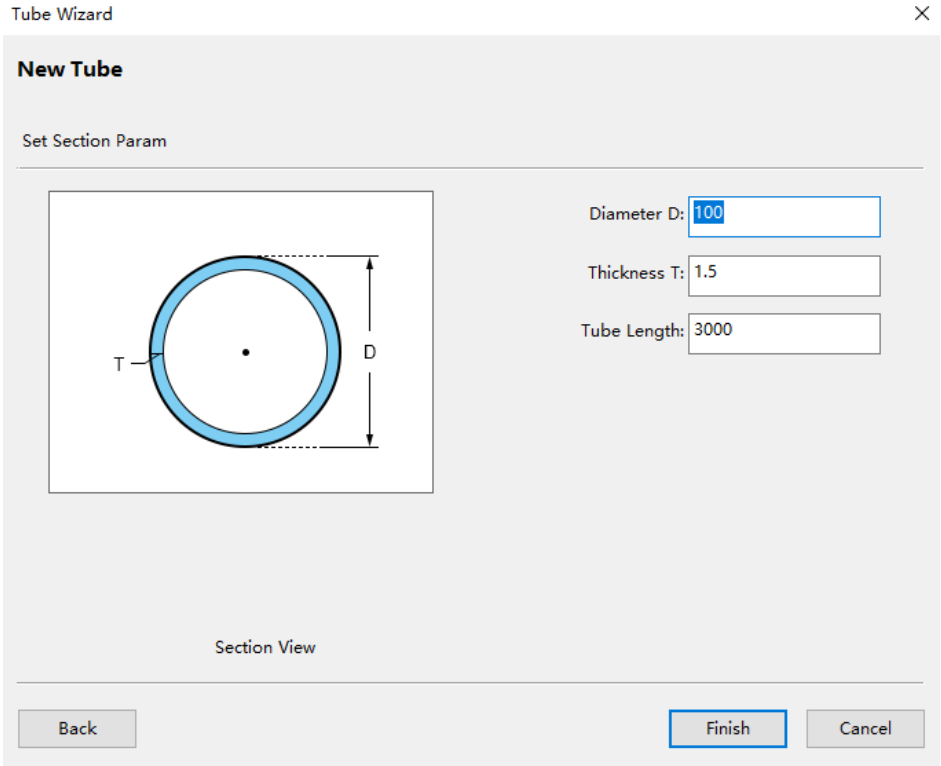
Operating Steps：

1. In the menu bar, click **Common** → 

→ **Standard Part** to open the **Tube Wizard** dialog box:

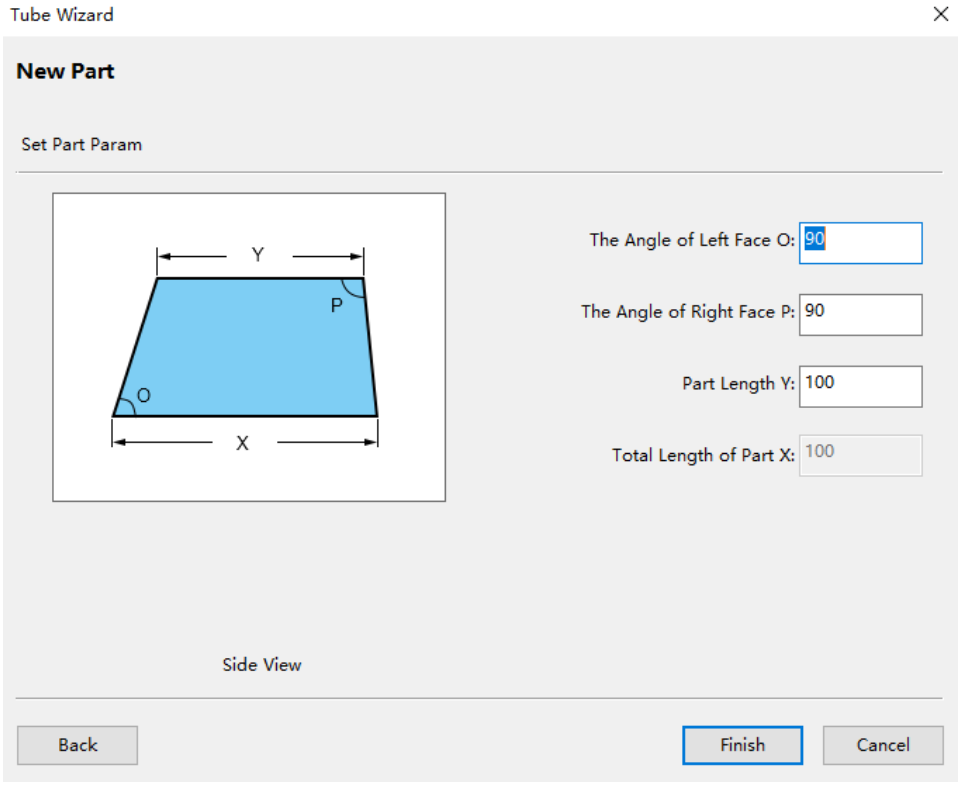


2. Click the corresponding tube type to open the **Set Section Param** dialog box:

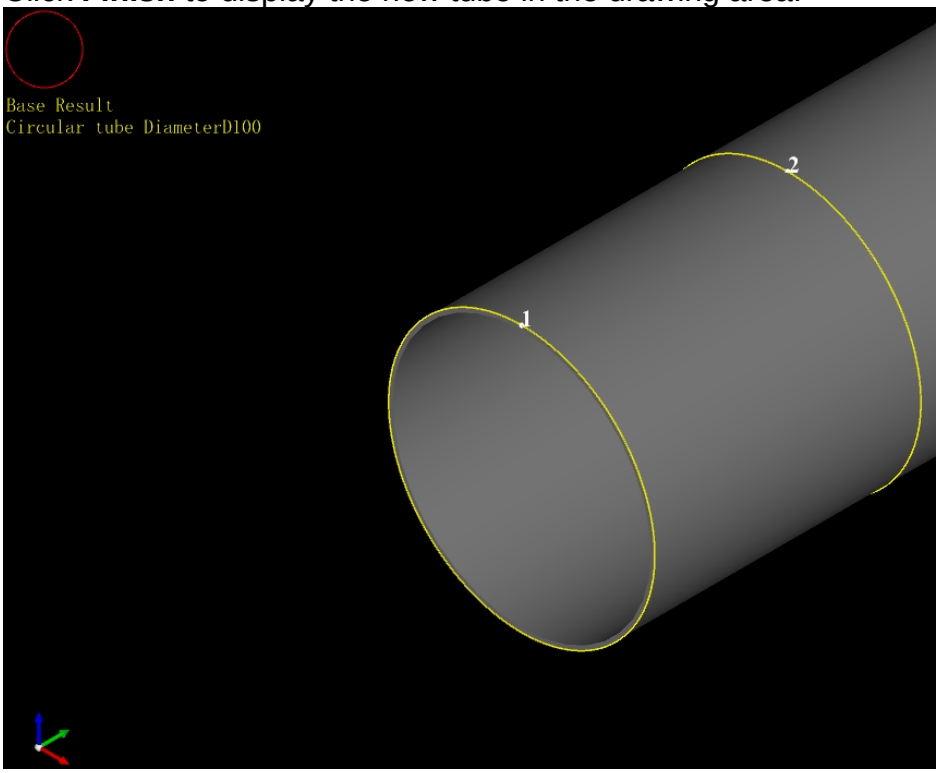


3. Set the section parameters. The left diagram shows the meaning of the parameters.

4. Click **Next** to open the **Set Part Param** dialog box:



5. Set the section parameters. The left diagram shows the meaning of the parameters.
6. Click **Finish** to display the new tube in the drawing area.



3.5. Import 3D Part

Import 3D parts are drawn by setting tube type, section and stretch information, intersecting and cutting tubes after creating new tubes.

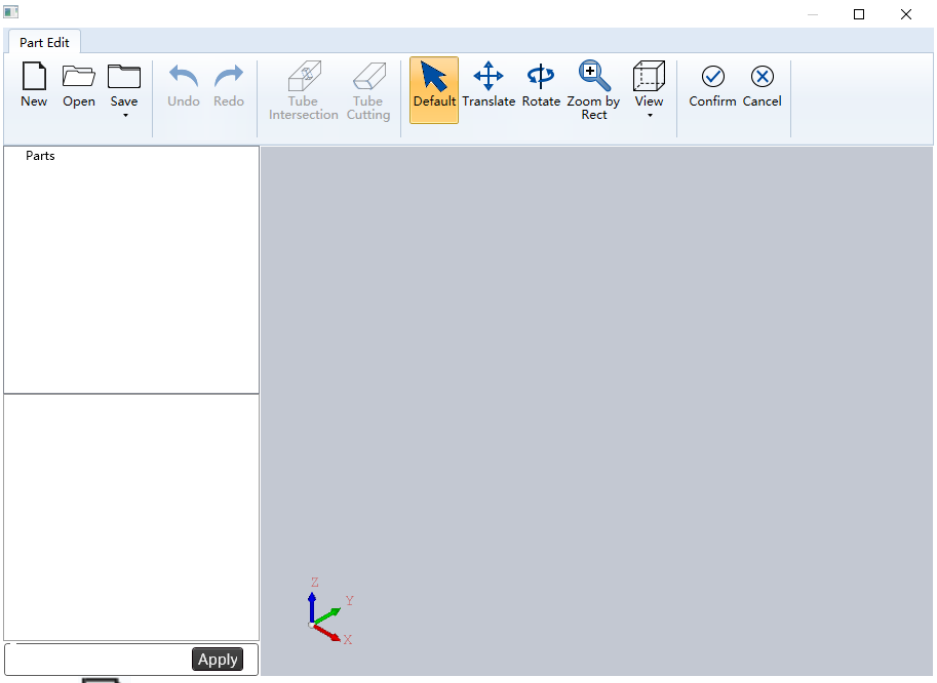
When creating a new tube type, specify the tube type and parameters. The tube type supports circular tube, rectangular tube, oval tube, waist tube, channel steel, and angle steel.

After using import 3D part function to draw parts, if you need to add other graphics, use 2D edit function to draw.

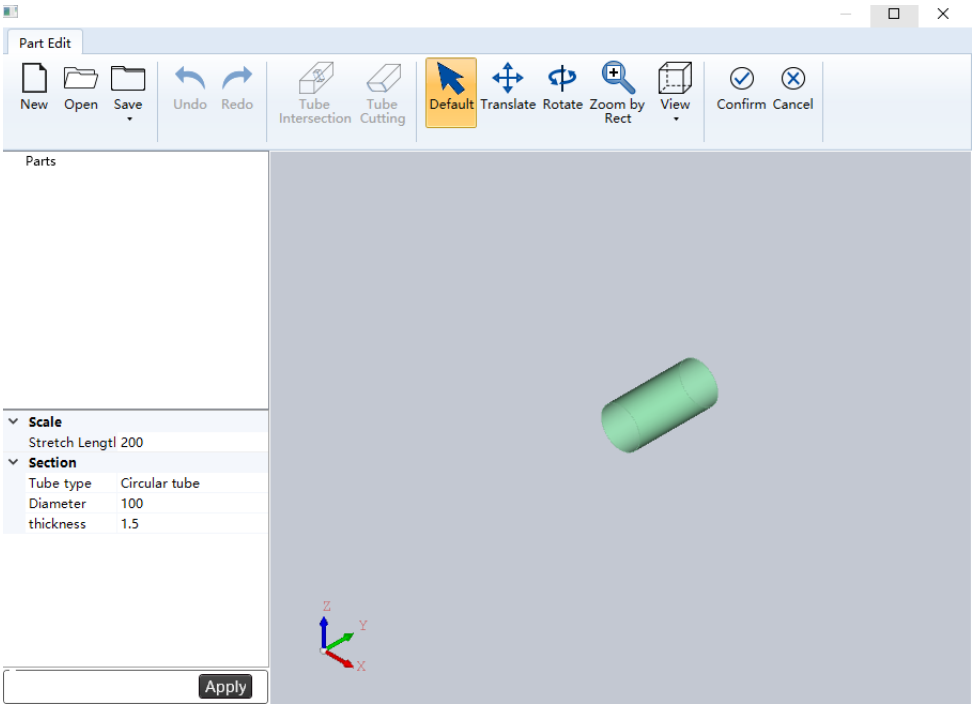
Operating Steps：

1. In the menu bar, click **Common** → 

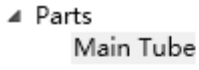
→ **Import 3D Part** to open the **Part Edit** page：

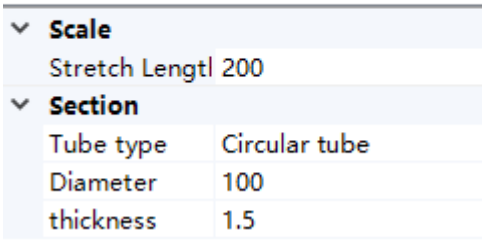


2. Click  **New**. The tube diagram pops up in the view, **Scale** and **Section** parameters pop up in the left list bar, as shown in the graphic：

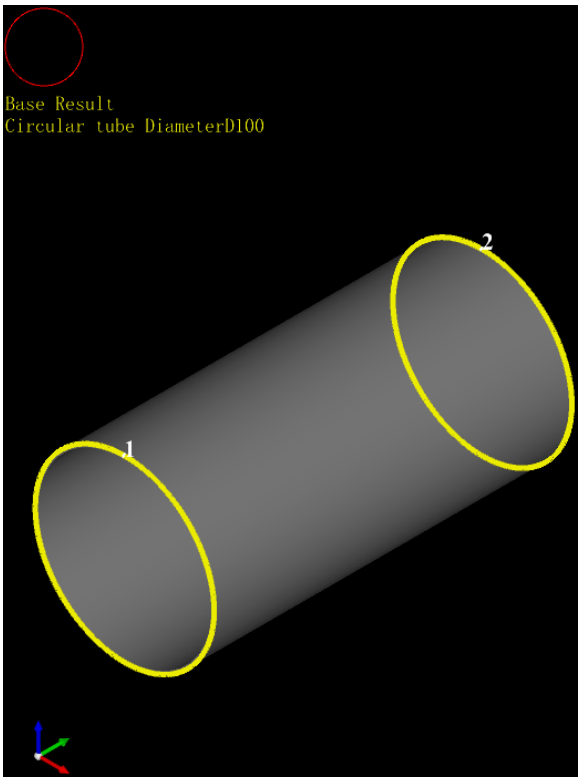


3. In the **Scale** and **Section** areas, select the tube type and set the tube size according to the actual needs.
4. After setting, click **Apply**, and **Main Tube** will be added to the **Parts** area.





5. Use the tube intersection and cutting functions to draw as needed. For details, see Set Tube Intersection and Set Tube Cutting.
6. After drawing, click  **Confirm** in the menu bar to return to the main interface, where the new part is displayed.

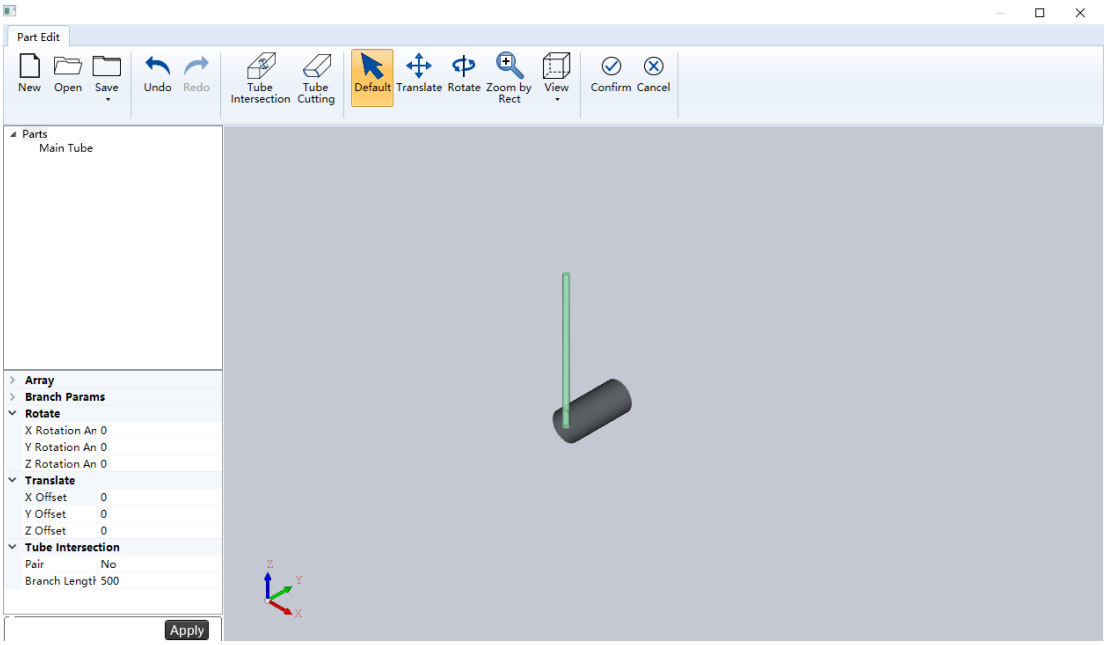


3.5.1. Set Tube Intersection

The intersecting function is to generate tube surface perforation through intersecting branch tube on the tube, and the array can be set. The types of intersecting holes include circular hole, rectangular hole and waist hole.

Operating Steps：

1. On the 3D drawing **Part Edit** page, click **Tube Intersection** in the menu bar, and the intersection parameter pops up at the lower left corner, as shown in the following graphic：



2. Set the parameters of translate, tube intersection, rotate, array and branch param. See the following table for detailed parameters：

Parameter	Description
X offset	The distance between the intersecting hole center and the starting point of the main tube in the X axis direction.
Y offset	The distance between the intersecting hole center and the starting point of the main tube in the Y axis direction.
Z offset	The distance between the intersecting hole center and the starting point of the main tube in the Z axis direction.
Pair	The branch tube through the whole tube and generate a pair on the main tube. Yes: generated; No; Do not generate.
Branch length	The length of the branch tube.
X rotation angle	The included angle between the straight line projected by the intersecting branch tube on the YZ plane and the positive direction of the Z axis.
Y rotation angle	The included angle between the straight line projected by the intersecting branch tube on the XZ plane and the positive direction of the Z axis.
Z rotation angle	The included angle between the straight line projected by the intersecting branch tube on the XY plane and the positive direction of the Y axis.
Array machining	Use intersecting arrays. Yes: enabled; No: not enabled.
Y array number	Number of array branches in Y direction.
Y array dis.	Array branch spacing in Y direction.
X array number	Number of array branches in X direction.
X array by angle	Yes: array by angle; No: arrays by spacing.
X array dis.	Array branch spacing in X direction. The parameter **X Array by Angle** takes effect when it is set to **No**.
X array angle	Array branch angle in X direction. The parameter **X Array by Angle** takes effect when it is set to **Yes**.
Branch type	Including **Circular Tube**, **Rectangular Tube** and **Waist Tube**.

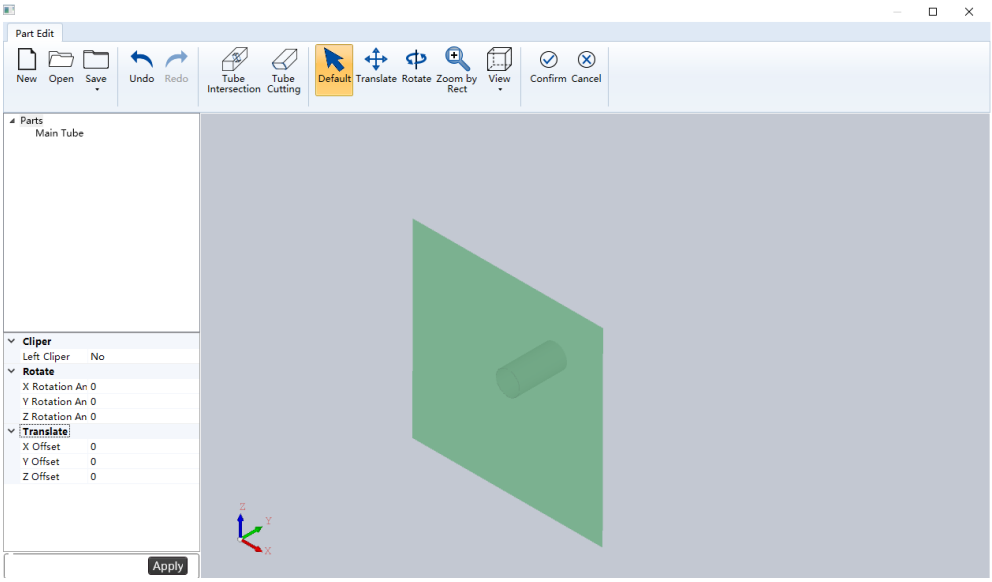
3. After setting, click **Apply** to add it to the **Parts** view area.

3.5.2. Set Tube Cutting

The tube cutting function is to generate cutting graphics on the tube for cutting tube parts.

Operating Steps：

1. On the 3D drawing **Part Edit** page, click **Tube Cutting** in the menu bar, and the cutting parameter pops up at the lower left corner, as shown in the following graphic：



2. Set translate, cliper and rotate parameters. See the following table for detailed parameter descriptions:

Parameter	Description
X offset	The distance between the center of the cutting plane and the starting point of the main tube in the X axis direction.
Y offset	The distance between the center of the cutting plane and the starting point of the main tube in the Y axis direction.
Z offset	The distance between the center of the cutting plane and the starting point of the main tube in the Z axis direction.
Left cliper	Yes: the tube with Y axis coordinate less than the current cutting surface is scrap; No: The tube with Y-axis coordinates greater than the current cutting plane is scrap.
X rotation angle	The angle between the cutting plane and the YZ plane.
Y rotation angle	The angle between the truncation plane and the XZ plane.
Z rotation angle	The angle between the truncation plane and the XY plane.

3. After setting, click **Apply** to add it to the **Parts** view area.

3.6. Insert Part

Insert a part on the spare tube in the tool path file without covering the tool path. support:

- Insert File
- Insert 3D Wrapping
- Insert Standard Part

3.6.1. Insert File

On the current tube, insert part tool path file and insert part position is:

- If the current tube has no part tool path, insert from the head of the tube.

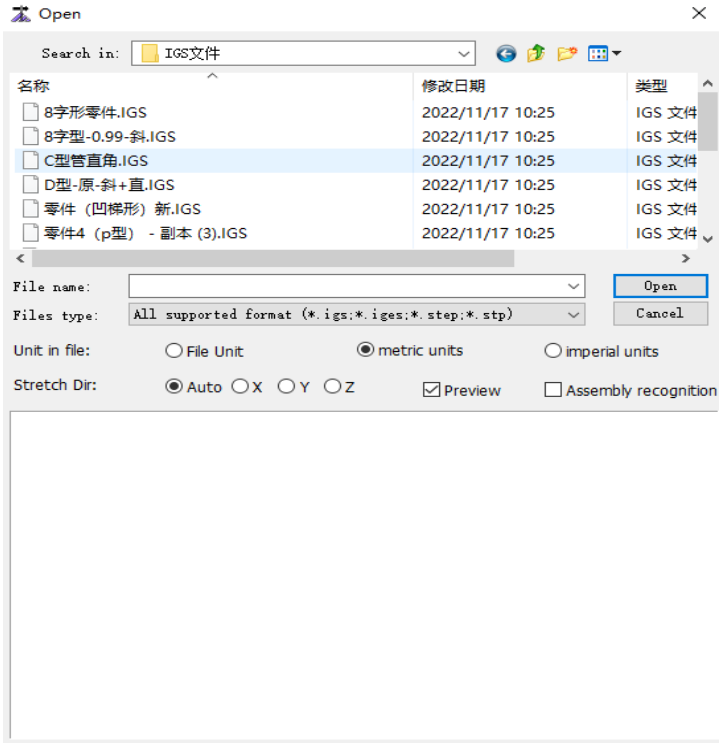
- If the pre-process in 2D edit uses insert part space, the inserted part starts from the end of the existing part plus the set space position.

Insert part file format as .igs, .iges, .step, .stp.

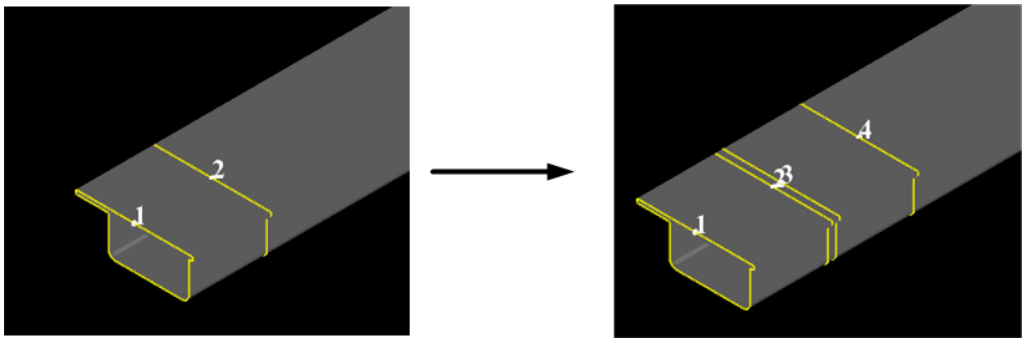
Note: The inserted part tube type must be consistent with the current tube type and section size.

Operating Steps:

1. In the menu bar, click **Common** → Insert Part → **Insert File** to open the **Open** dialog box:



2. Select the part you want to import.
3. Set file unit and stretch direction.
4. Click **Open**, and the software will identify whether the tube type of the inserted part is consistent with the current tube type and section size.
   - Consistent: The part is inserted on the current tube.



   - Inconsistent: **The Alarm/Log Message Bar** prompts the cause of the error.

3.6.2. Insert 3D Wrapping

Insert a 3D wrapping graphic on the current tube at:

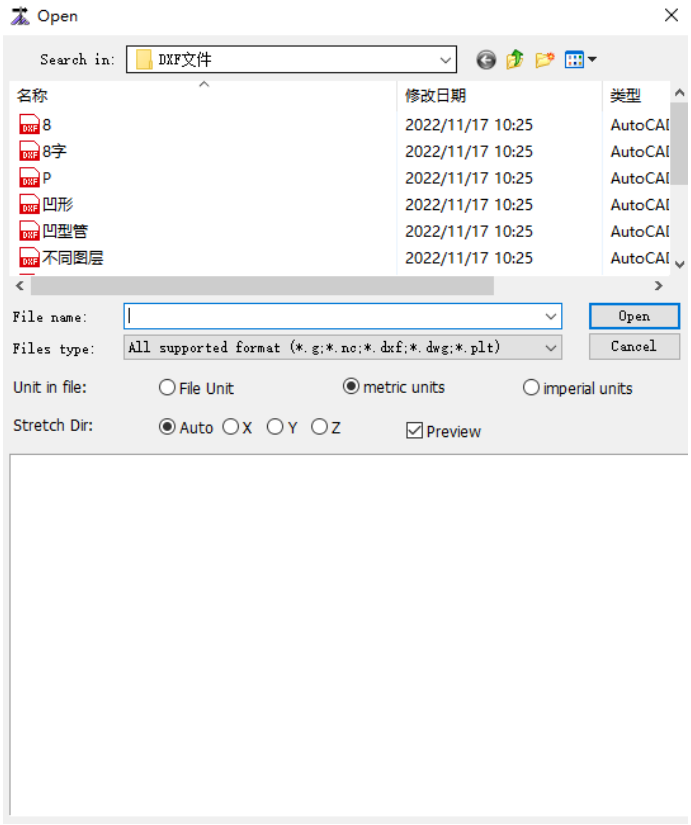
- If the current tube has no part tool path, insert from the head of the tube.

- If the pre-process in 2D edit uses insert part space, the inserted part starts from the end of the existing part plus the set space position.

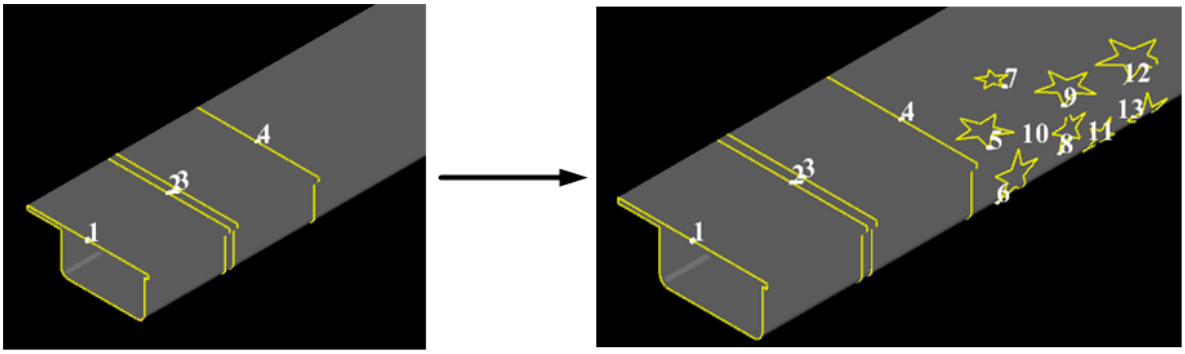
The supported formats for wrapping graphics files are .g, .nc, .dxf, .dwg, .plt.

Operating Steps：

1. In the menu bar, click **Common** → Insert Part → **3D Wrapping** to open the **Open** dialog box:



2. Select the file to import.
3. Set file unit and stretch direction.
4. Click **Open**.
   Effect picture:

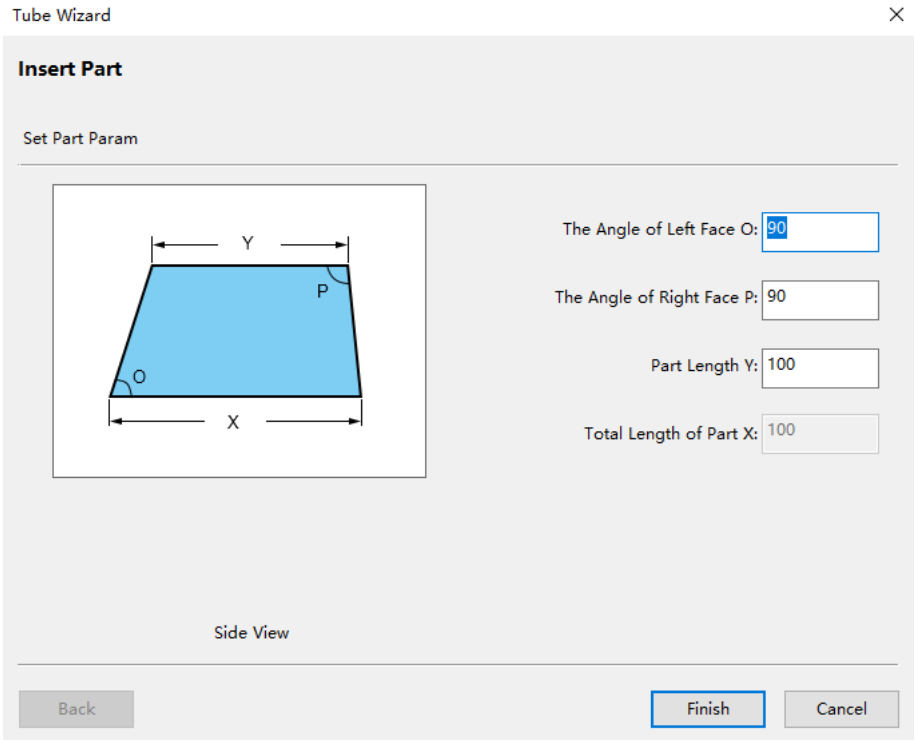


3.6.3. Insert Standard Part

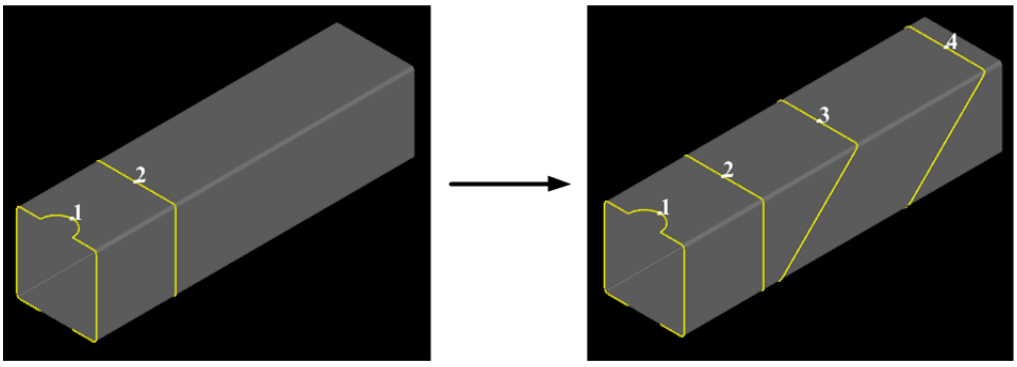
Insert standard part of the same type of tube on the current standard tube at:

- If the current tube has no part tool path, insert from the head of the tube.
- If the pre-process in 2D edit uses insert part space, the inserted part starts from the end of the existing part plus the set space position.

1. In the menu bar, click **Common** → Insert Part → **Standard Part** to open the **Pipe Wizard** dialog box:



2. Set the part parameters. The left diagram shows the meaning of the parameters.
3. Click **Finish**.
   Effect picture:



4. Edit Path File

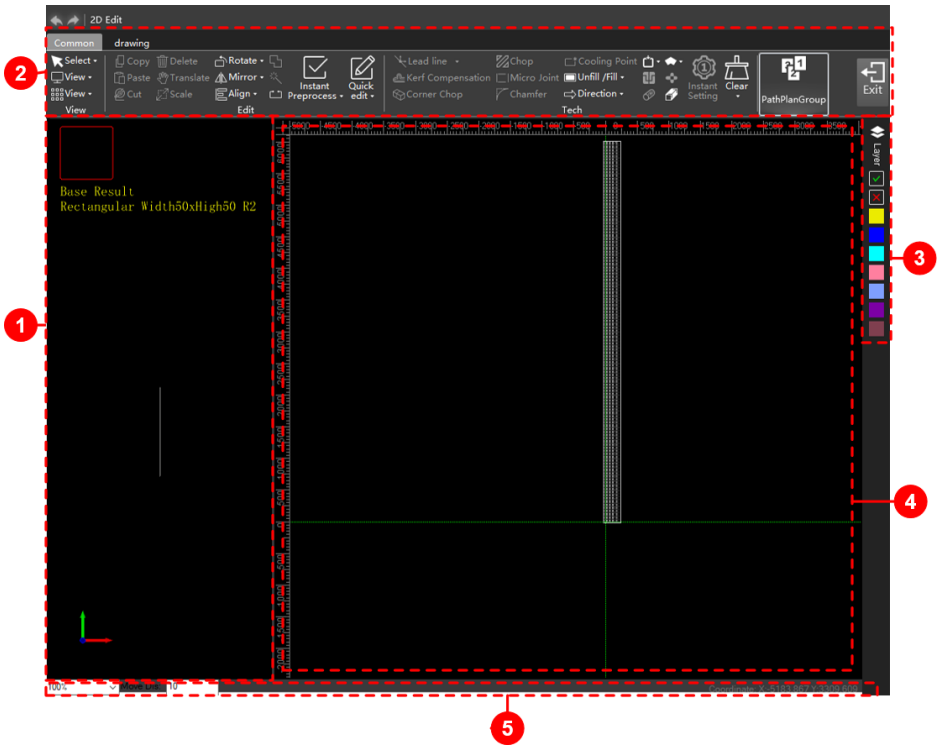
The tool path file can be edited in the **Software Main Interface** and **2D Edit**. When editing and importing, the main differences between the two pages:

- View display
  - Main interface of software: parts are displayed in 3D.
  - 2D edit: 2D expand the part along the centerline of the 3D view to form a 2D view.
- Edit function
  The editing functions supported by the **Software Main Interface** and **2D Edit** are different, as shown in the following table. For some supported functions, see the corresponding chapters for details.

Function	Software Main Interface	2D Edit
View Operation	Support	Support
Draw Graphic	Nonsupport	Support
Make Array	Nonsupport	Support
Auxiliary Tool	Nonsupport	Support
Edit Graphic	Nonsupport	Support
Preprocess Graphic	Nonsupport	Support
Quick Edit	Partial support	Support
Layer Technic	Support	Support
Machining Technic	Support	Support
Arrange Group	Partial support	Support

- Page layout
  - Software main interface: For details, see Introduce the Main Interface of the Software.
  - 2D edit page: In the **Common** menu bar of the software main interface, click 

to open the **2D Edit** page.



Page layout description:

No	Name	Description
1	3D view	View the tube from different perspectives and preview the cutting effect. The tube can be scaled, rotated and translated.
2	Menu bar	Collection of function buttons.
3	Layer toolbar	Layer related operation tool button.
4	Drawing area	Preview and draw graphics in this area.
5	Status bar	Current operation related information: operation steps and significance, operation success, etc. Display coordinate position, adjust view zoom, etc.

Preview 3D view:

- Scale: Scroll the mouse wheel, scroll up to zoom in, and scroll down to zoom out.
- Translation: Press **Ctrl** and click the left mouse button to move the mouse.
- Rapid rotate around the tube stretch direction: Press **Ctrl** and scroll the mouse wheel.
- Slowly rotate around the tube stretch direction: Press **Shift** and scroll the mouse wheel.
- Rotate around the tube stretch direction: Hold down the left mouse button and drag the mouse.
- Rotation in any direction: Hold down the mouse wheel and drag the mouse.

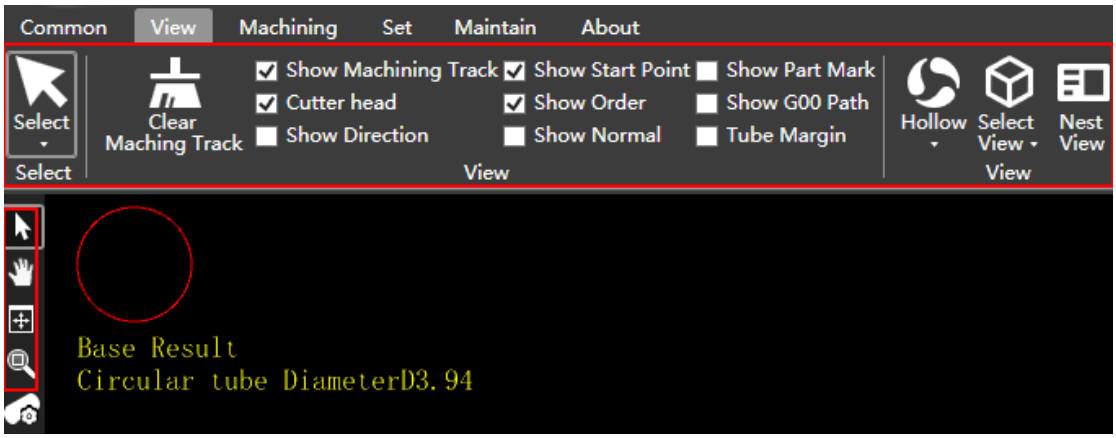
4.1. Graphic Operation

4.1.1. View Operation

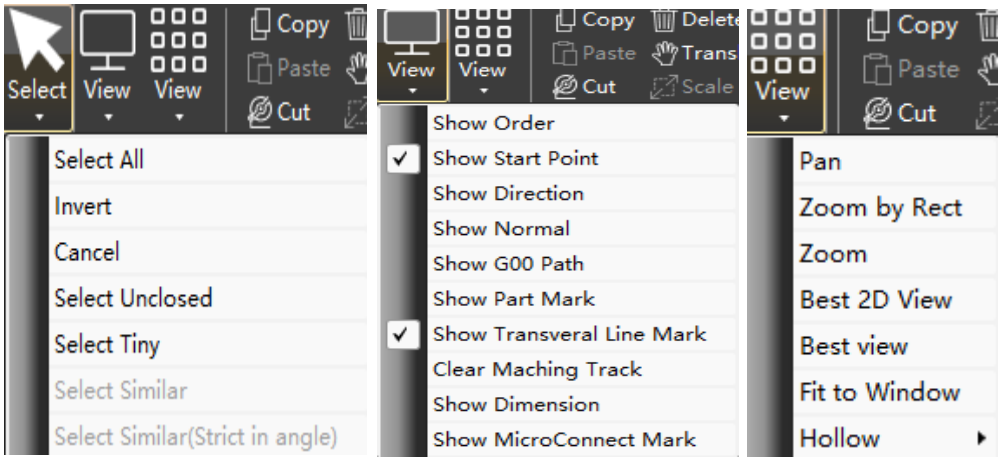
4.1.2. Overview

View display operation can be performed on the **Software Main Interface** and **2D Edit** pages, and the operation effect is updated synchronously. This article takes the **Software Main Interface** as an example to introduce the method and effect of view display operation.

Related buttons:



Related buttons for **2D Edit**:



4.1.3. Select Graph

Select graphic for easy editing. Two graphic selection methods are supported:

- Manual selection: Manually select any graphic.
- Automatic selection: Automatically select the graphic that meets the conditions.

4.1.3.1. Manually Select Graphic

Operating Steps:

1. On the common toolbar, click  .
2. Select any of the following methods to select graphic:
   - Click the left mouse button to select a single graphic.
   - Press and hold the **Ctrl** key, click the left mouse button in turn, and select multiple graphics.

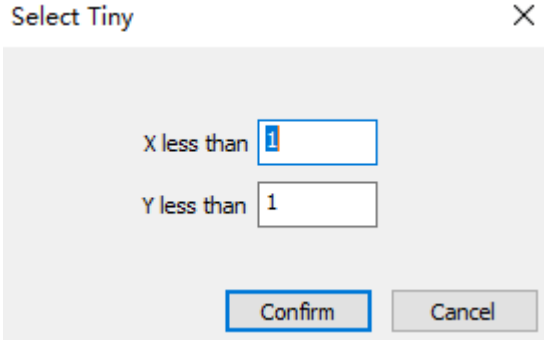
- Press the right mouse button and drag the mouse to select graphics. Select all the graphics that intersect the box and are included in the box.

4.1.3.2. Automatically Select Graphic

Operating Steps：

1. In the menu bar, click **View** → 

, select:
   - **Select All**: The system automatically selects all graphics.
   - **Invert**: Inverts the selection of unselected graphics.
   - **Cancel**: The system automatically deselects all graphics.
   - **Select Unclosed**: Select all unclosed graphics in the file.
   - **Select Tiny**: Select the graphic whose X axis and Y axis dimensions are smaller than the set value.
     i. Select **Select Tiny** to open the following dialog box:



     ii. Enter X size and Y size, and click **Confirm**.
   - **Select Similar**: After manually select a graphic, click **Select Similar**, and the system will automatically select the graphic with the same type and size as the selected graphic.

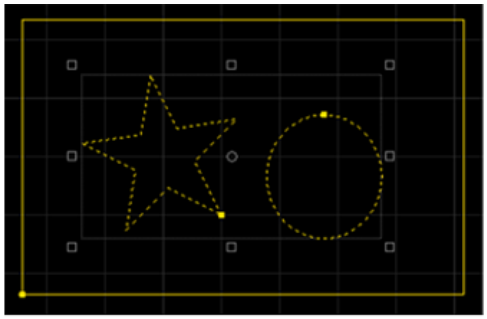
This operation does not distinguish between angles.

   - **Select Similar（Strict in Angle）**: After manually select a graphic, click **Select Similar（Strict in Angle）**, and the system will automatically select graphics with the same type, size, and angle as the selected graphics.

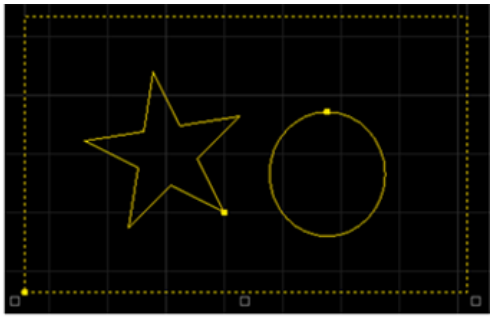
This operation distinguishes angles.

   - **Select by Layer**: Select the corresponding layer under the submenu, and the system will automatically select the graphic in the layer.
   - **Select by Inner-Outer：**: Select the corresponding inner graphic (included graphic) or outer graphic (excluded graphic) in the submenu. The system

automatically selects the corresponding graphic, and the effect picture is as follows:



Select by Inner-Outer - Inner Graphic



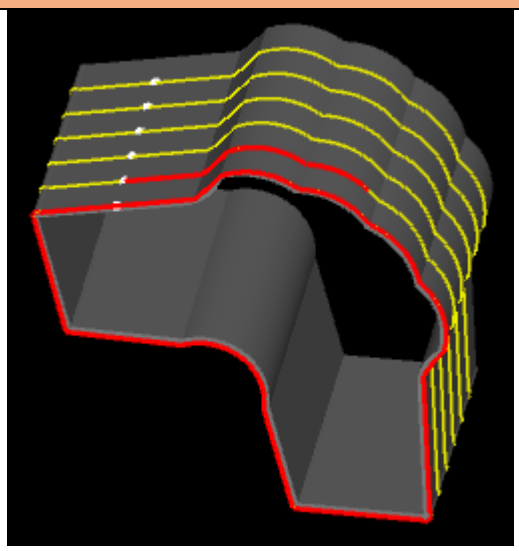
Select by Inner-Outer - Outer Graphic

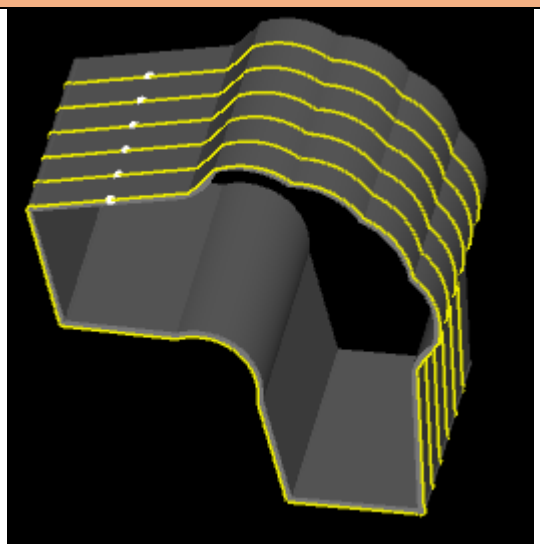
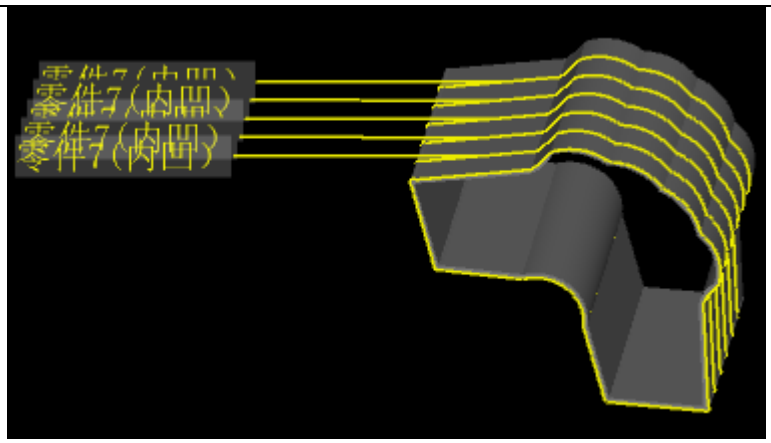
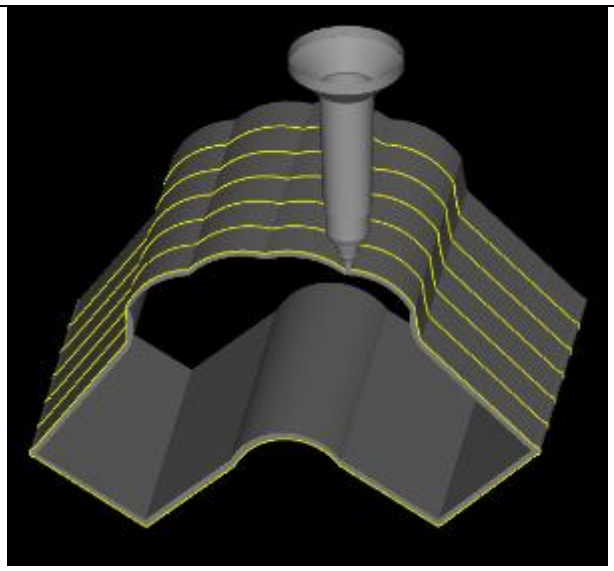
4.1.4. Show

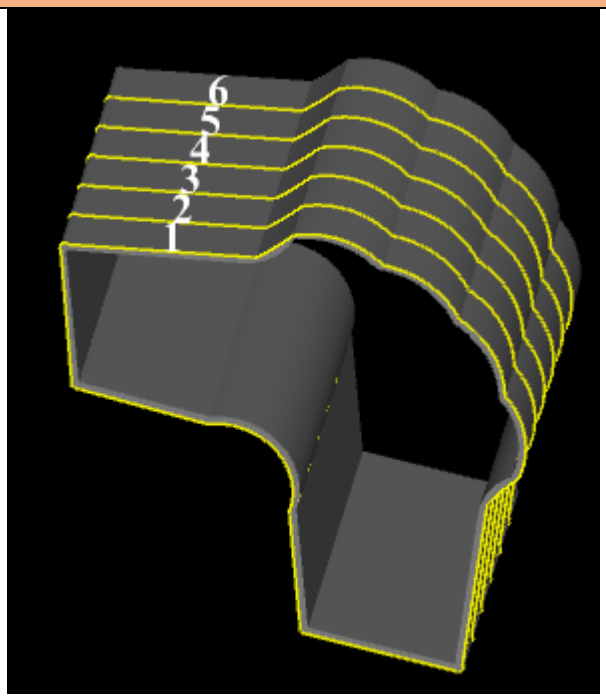
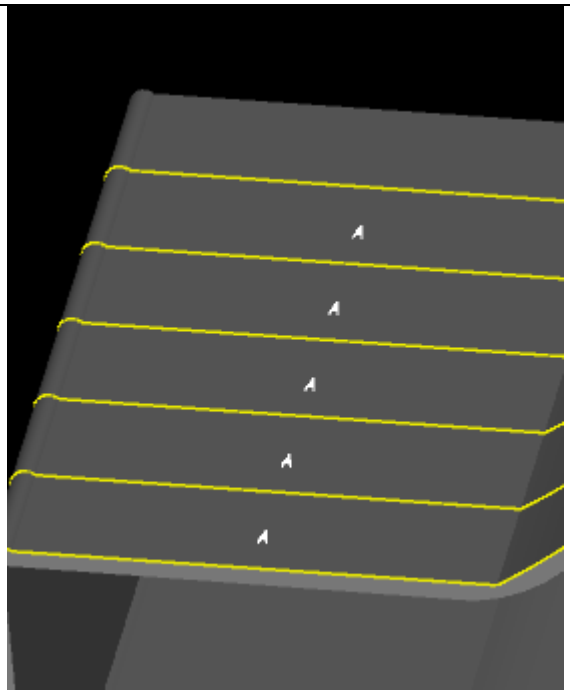
In order to better observe the technic effect, the software provides the display/hiding of various technic effects.

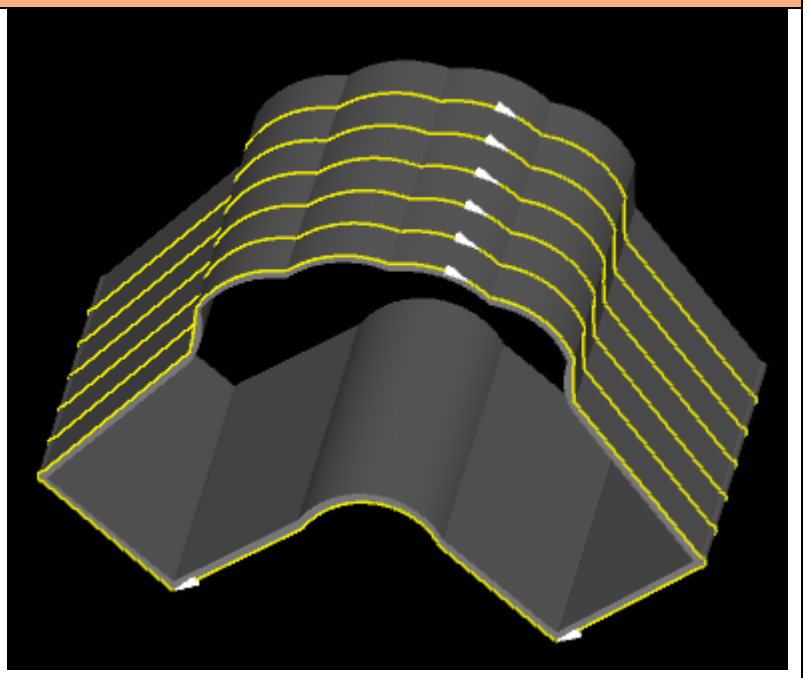
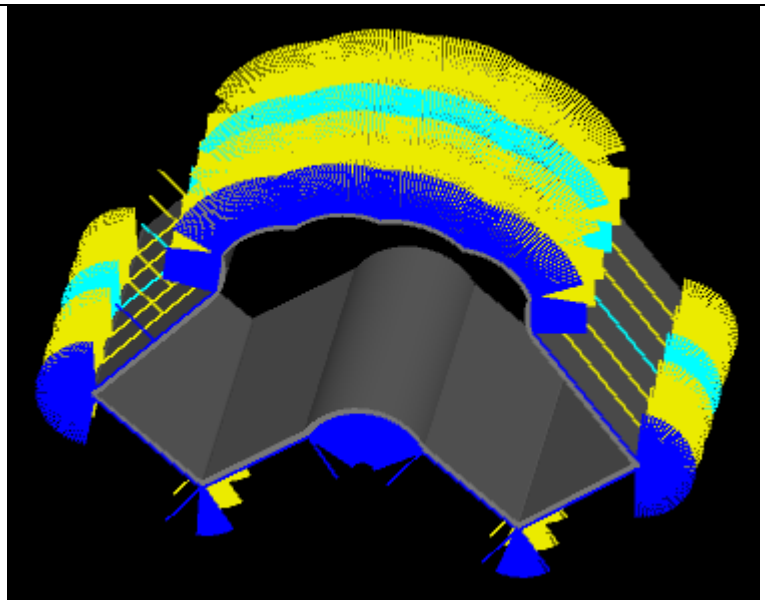
Operating Steps:

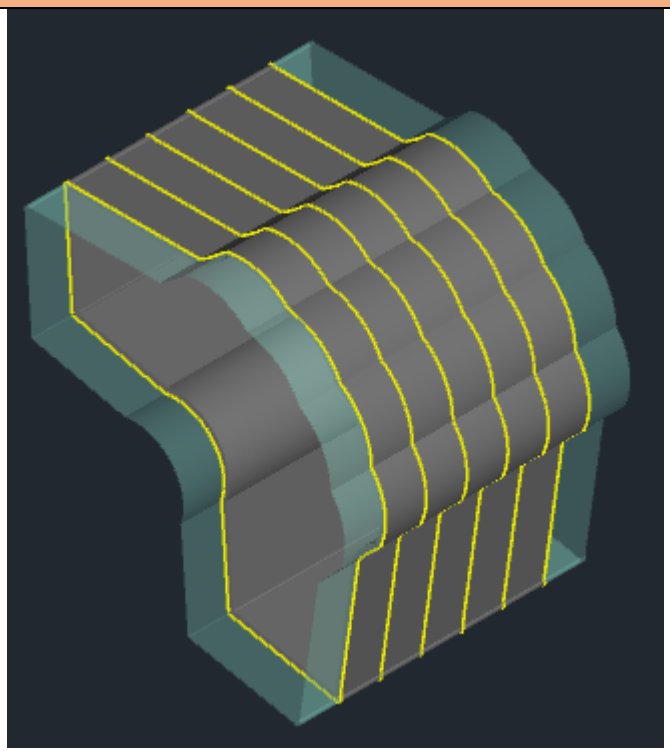
1. Check the technic effect to be displayed in the display area of the **View** menu bar, and display the corresponding technic effect in the drawing area:

Check	Description	Effect Picture
Show machining track	The red line is used to display the machining track, especially in the simulation, the simulated machining track is displayed in real time. If you want to clear the machining path, click the  .	

Check	Description	Effect Picture
Show start point	Use white dots to display the machining start point of the graphic.	
Show part mark	Display part information in the way of mark.	
Cutter head	During the simulation, the real time simulation shows the running track of the cutter head.	

Check	Description	Effect Picture
Show order	Use numbers to indicate the machining sequence.	
Show G00 path	G00 path is the cutting head movement path from the end of one part cutting to the new one. Use the white arrow to display the moving direction of the machining head.	

Check	Description	Effect Picture
Display direction	Use the white arrow to display the machining direction of the part.	
Show normal	Use the color corresponding to the layer to display the effect of normal adjustment.	

Check	Description	Effect Picture
Tube margin	The distance from the starting cutoff edge of the cutting part to the tube edge, and the effect is displayed with unused tube.	

4.1.5. View

4.1.5.1.1. Select View Angle

Select a standard angle to view the graphic.

Operating Steps：

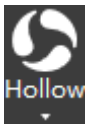
1. In the menu bar, click **View** → 

and select the following angles in the **Select View** submenu bar to view the graphic：
   - Front view
   - Rear view
   - Left view
   - Right view
   - Top view
   - Bottom view

4.1.5.1.2. Hollow

Three hollow modes are supported: solid, hollow out and wireframe. Solid hollow mode is displayed by default.

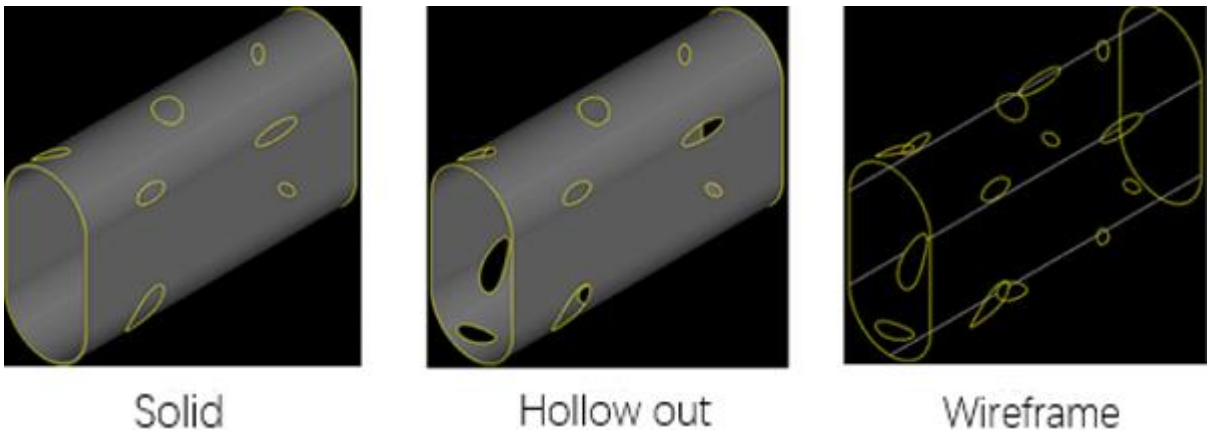
Operating Steps：

1. In the menu bar, click **View** → 

And select the following hollow modes in the **Hollow** submenu bar：
   - Solid

- Hollow out
- Wireframe

Effect picture:



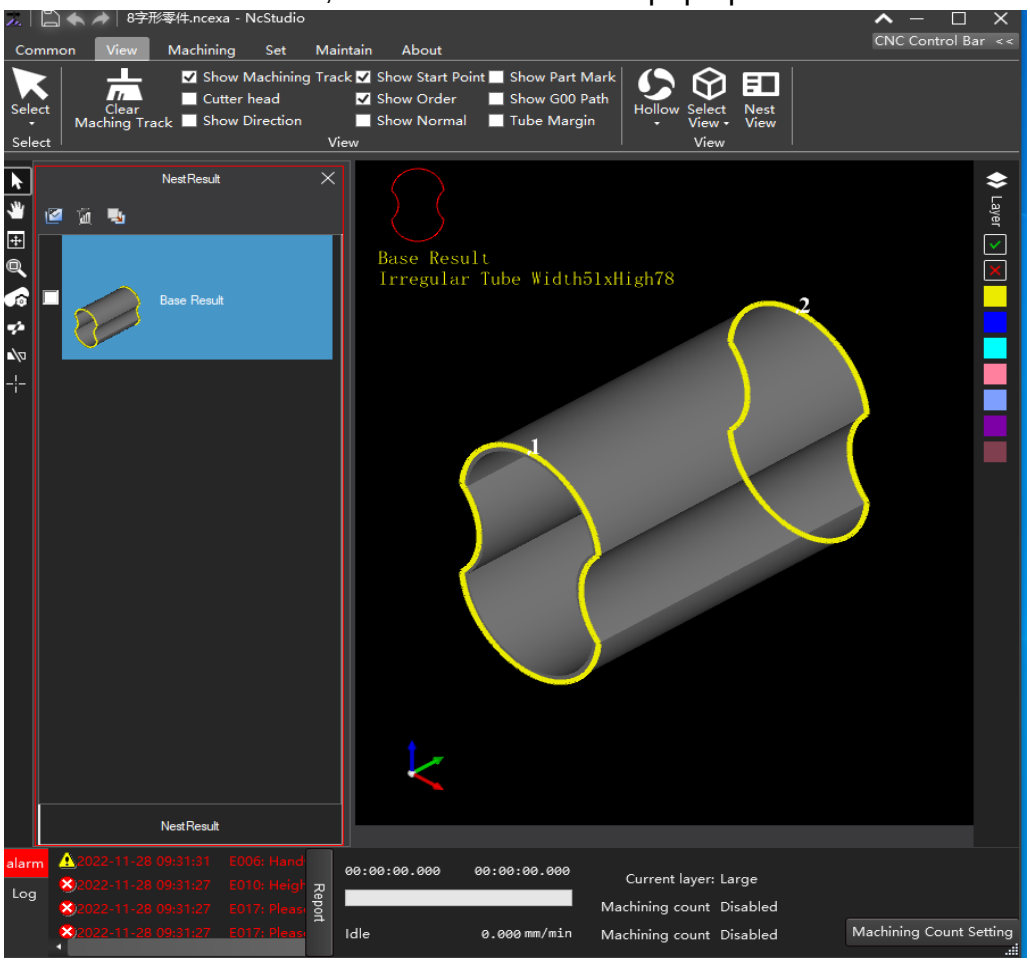
4.1.5.1.3. Nest View

Displays or hides the nest result list.

Operating Steps：

1. In the menu bar, click **View→** 

To pop up the **Nest Result**:



1. Description of the nest result list operation button:

-  Select: Select all or not select nesting results.
-  Delete: Delete the selected nest result.
-  Base result: Switch to the base view.

4.1.5.2. Other Common Operations

4.1.5.2.1. Pan

Relocate the position of graphic in the drawing area, so as to observe different parts of the current graphic.

Operating Steps:

1. On the common toolbar, click  to use pan function.
2. Select a point, press the left mouse button, drag to the target location and release the mouse.
3. Press **Esc** or right click to exit the tool.

4.1.5.2.2. Best View

The graphics are adjusted to the starting position and the default size, and all are displayed in the drawing area.

Operating Steps:

1. Select one of the following methods to use the best view function:
   - On the common toolbar, click  .
   - In the drawing area, right click and select **Best View**.

4.1.5.2.3. Zoom by Rect

Enlarge the part of the graphic to the size of the view window.

Operating Steps:

1. On the common toolbar, click  to use zoom by rect function.
2. Press the left mouse button to move and select the area to be zoomed in. Release the left mouse button to zoom in.
3. Press **Esc** or right click to exit the tool.

4.1.5.2.4. Zoom View

Zoom in or out to see the added technic effect more clearly.

Scroll the mouse wheel to zoom in and zoom out.

4.1.5.2.5. Rotate View

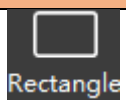
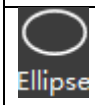
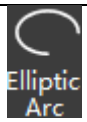
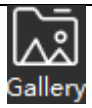
Rotate the view 360 ° to view the tube more comprehensively.

Operating Steps:

1. Select one of the following methods to rotate the view:
   - Press the mouse wheel and drag the mouse to rotate the tube in any direction.
   - Press **Ctrl** and roll the mouse wheel to quickly rotate around the tube stretching direction.
   - Press **Shift** and scroll the mouse wheel to slowly rotate around the tube stretching direction.

4.1.6. Draw Graphic

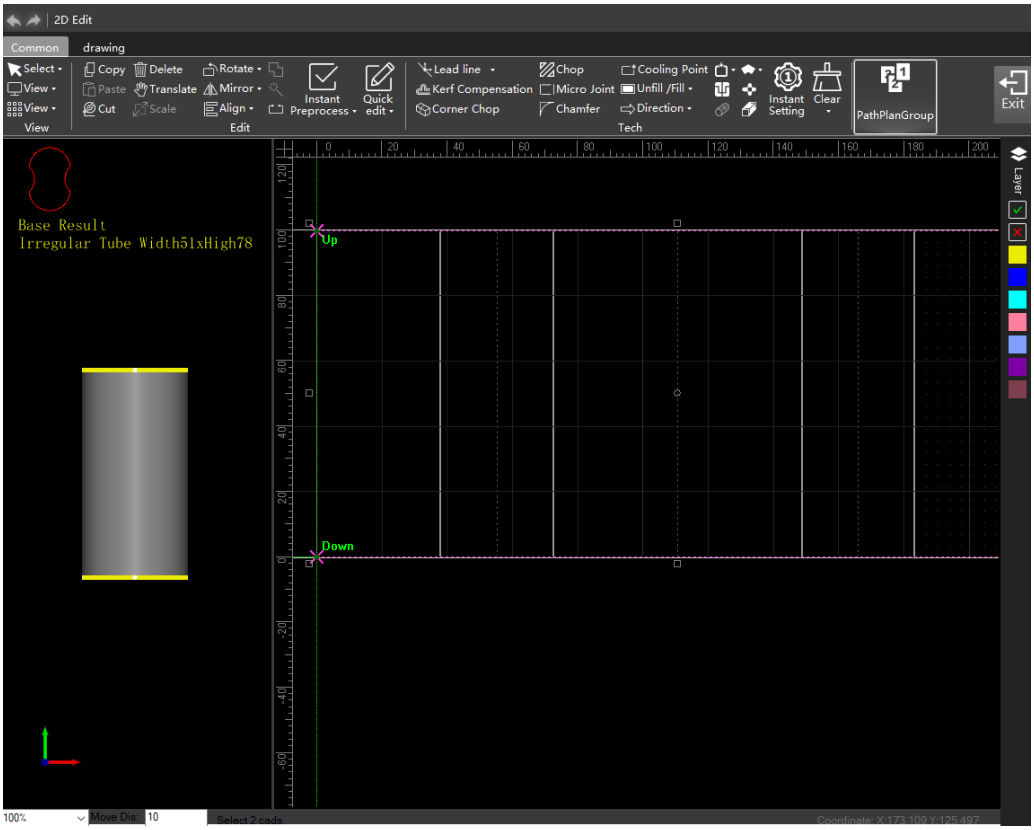
The software supports to draw the following graphics, and the system provides a graphic library function, which provides commonly used graphics for users to use directly.

Tool	Name	Tool	Name	Tool	Name
	Dot		Line		Rectangle
	Polyline		Circle		Arc
	Ellipse		Elliptic arc		Polygon
	Star		Text		Gallery

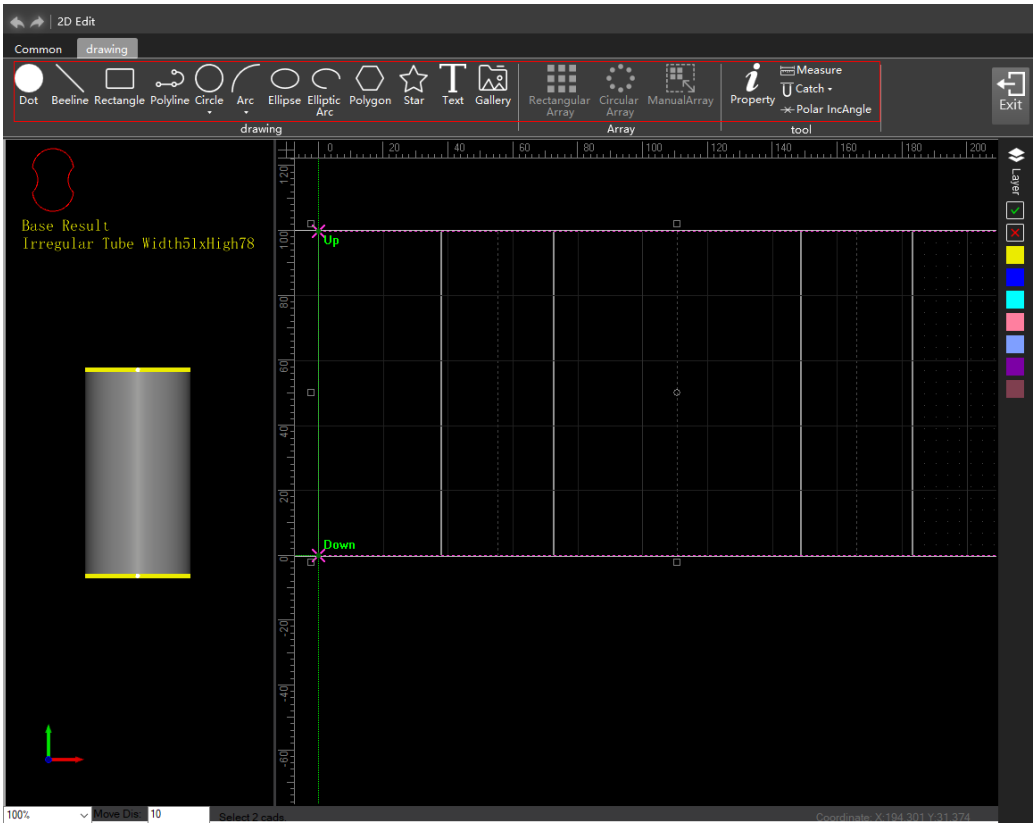
The following describes how to use drawing tools, which will not be described in subsequent chapters.

Operating Steps：

1. In the menu bar, click **Common** →  to open the **2D Edit** page:



2. In the menu bar, click the **Drawing** tab to switch to the drawing menu bar, as shown in the following picture:



3. Click the corresponding drawing tool to use it.

4.1.6.1. Dot

Operating Steps:

1. In the **2D Edit** page, click the left mouse button to select a point to add a dot graphic.
2. Click the left mouse button to draw the next dot.
3. Right click to exit the tool.

4.1.6.2. Beeline

Operating Steps:

1. In the **2D Edit** page, click the left mouse button to select the start point.
2. Click the left mouse button to select the next point.
3. Right click to exit beeline drawing.

4.1.6.3. Rectangle

Operating Steps:

1. In the **2D Edit** page, click the left mouse button to select the start point.
2. Move the mouse to the corresponding position and click the left mouse button to select the end point.
3. Right click to exit the tool.

4.1.6.4. Polyline

Polyline refers to a single object composed of multiple lines and arcs. This software supports the switching between lines and arcs.

Operating Steps:

1. In the **2D Edit** page, click the left mouse button to select the start point.
2. **（Optional：）** Right click to draw an arc, and click **Tangent Arc** in context menu. By default, the software draws **Lines**.

   Note

   The initial default for a polyline is to draw **Line**.

3. Click the left mouse button to select the next point and repeat. Click the right mouse button and click **Line** or **Tangent Arc** in context menu to switch the drawing tool.
4. After the drawing is complete, right click the mouse button and select the following operations in the context menu as required：
   - To set the current point as the end point of the polyline and the polyline as an unclosed object, click **Confirm**.

     Note

     To close it, check **Closed** in the information above the view after select the graphic.

   - To set the current point as the end point of the polyline and the polyline as a closed object, click **Closed**.
   - To cancel all previous selections and exit polyline, click **Cancel**.
5. Continue to click the right mouse button to exit drawing.

4.1.6.5. Circle

Operating Steps：

- Use radius to draw circle：
  a. In the menu bar of the **2D Edit** page, click **Drawing** → 

 → **Radius Circle**.
  b. Click the left mouse button and select the center of the circle.
  c. Click the left mouse button to select a point, and the distance from the point to the center of the circle is the radius.
  d. Click the right mouse button to exit tool.
- Use three point circle to draw a circle：
  a. In the menu bar of the **2D Edit** page, click **Drawing** → 

 → **Three Point Circle**.
  b. Click the left mouse button three times to select three points to form a circle.

  Note

Three points can form a triangle if they are not on a straight line and do not overlap. Draw the intersection of any two vertical centerlines to be the center of a circle.

c. Click the right mouse button to exit tool.

4.1.6.6. Arc

Operating Steps：

- Use radius to draw arc：

a. In the menu bar of the **2D Edit** page, click **Drawing** → 

→ **Radius Arc**.
b. Click the left mouse button and select the center of the circle.
c. Click the left mouse button, select a point as the start point of the arc, and the distance to the center of the circle is the radius.
d. a. Click the left mouse button to select the end point of the arc. The default machining direction of the tool path generated by the software is counterclockwise.
e. Click the right mouse button to exit tool.

- Use three point arc to draw arc：

a. In the menu bar of the **2D Edit** page, click **Drawing** → 

→ **Three Point Arc**.
b. Click the left mouse button to select a point, which is the start point of the arc.
c. Click the left mouse button to select the second point.
d. Click the left mouse button to select the third point, which is the end of the arc.
e. Click the right mouse button to exit tool.

4.1.6.7. Ellipse

Operating Steps：

1. In the **2D Edit** page, click the left mouse button to select the center point.
2. Click the left mouse button to select two points respectively. The distance from the two points to the center point is the long and short half axes of the ellipse.
3. Click the right mouse button to exit tool.

4.1.6.8. Ellipse Ace

Operating Steps：

1. In the **2D Edit** page, click the left mouse button to select the center point.
2. Click the left mouse button to select the long half axis and the short half axis distance respectively.
3. Click the left mouse button twice to select the start point and the end point respectively, and the default machining direction of the generated tool path is counterclockwise.
4. Click the right mouse button to exit tool.

4.1.6.9. Polygon

Operating Steps：

1. In the **2D Edit** page, click the left mouse button to select the center point.
2. Click the left mouse button to select the end point.
3. Click the right mouse button to exit tool.

4.1.6.10. Star

Operating Steps：

1. In the **2D Edit** page, click the left mouse button to select the center point.
2. Click the left mouse button to select the end point.
3. Click the right mouse button to exit tool.

4.1.6.11. Text

Operating Steps：

1. In the **2D Edit** page, drag the cursor to determine a rectangular text box.
2. Enter text in the text box.
3. **（Optional：）** Press **Ctrl+Enter** to switch to the next line.
4. Press **Enter** or click **Confirm** to complete the text drawing.
5. Click the lift mouse button to exit tool.

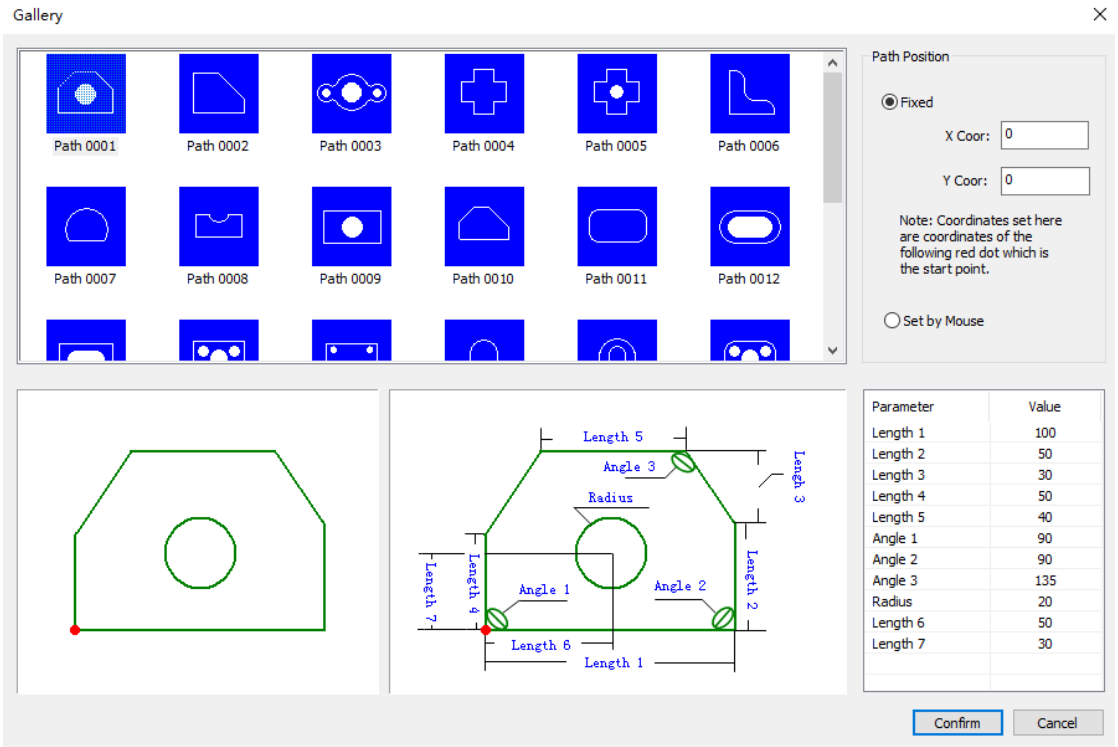
4.1.6.12. Gallery

The software provides common graphic templates, and can set graphic parameters and graphic locations.

Operating Steps：

1. In the menu bar of the **2D Edit** page, click **Drawing** → 

to open the **Gallery** dialog box:



2. Click to select a graphic, and the preview effect of the graphic will be displayed below.
3. Double click the corresponding parameter in the parameter bar as required to set the parameter value.

4. In the **Path Position** area, select any of the following methods to set the position of the graphic.
   - Check **Fixed**, set the values of X coordinate and Y coordinate, and click **Confirm**.

     Note

     The X coordinate and Y coordinate values set are the red dot positions on the preview effect picture.

   - Check **Set by Mouse** and click **Confirm** to close the **Gallery** dialog box. In the **Drawing Area**, select a place and click the mouse.

4.1.7. Make Array

Array is one of the simple nested forms, which can copy the graphic into several identical and neatly arranged graphics to improve the machining efficiency.

Array modes are divided into:

- Rectangular Array
- Circular Array
- Manual Array

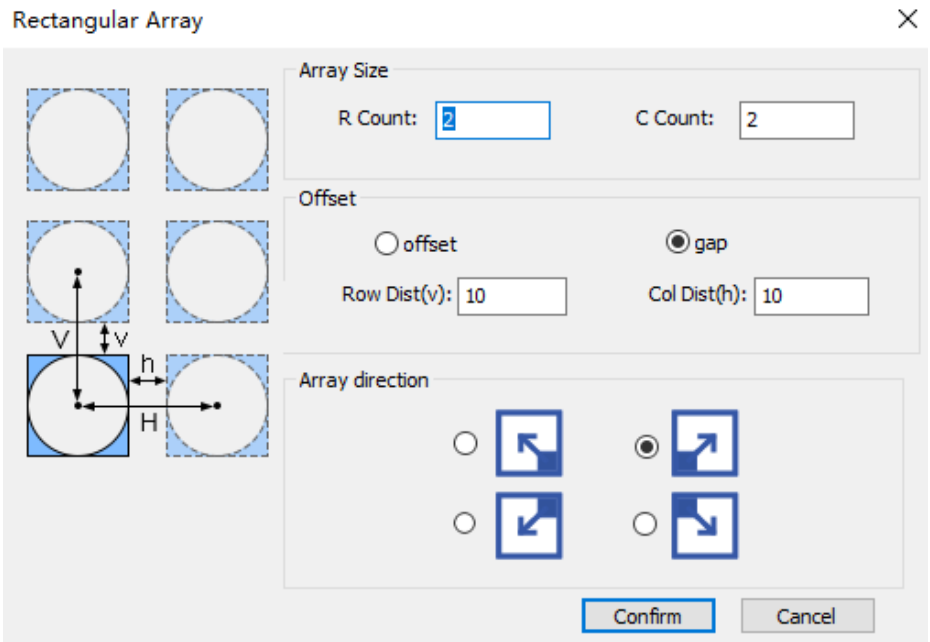
4.1.7.1. Rectangular Array

Copy the graph along the rectangular array.

Operating Steps:

1. Select one or more graphics in the **2D Edit** page.
2. In the menu bar, click **Drawing** → 

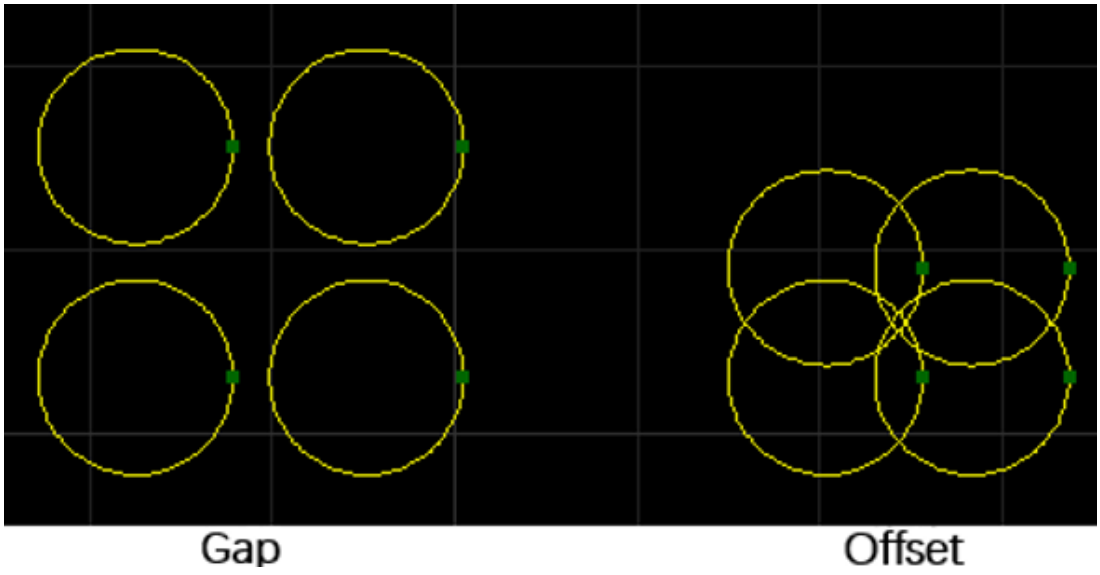
to open the **Rectangular Array** dialog box:



3. Sets the r count and c count for a rectangular array.

4. In the **Offset** area, set the offset mode:
   - **Offset**: Translate based on the center of the graphic.
   - **Gap**: Translate based on the graphic frame.
5. Select the row dist(v) and col dist(h) for the rectangular array.

Effect picture:



4.1.7.2. Circular Array

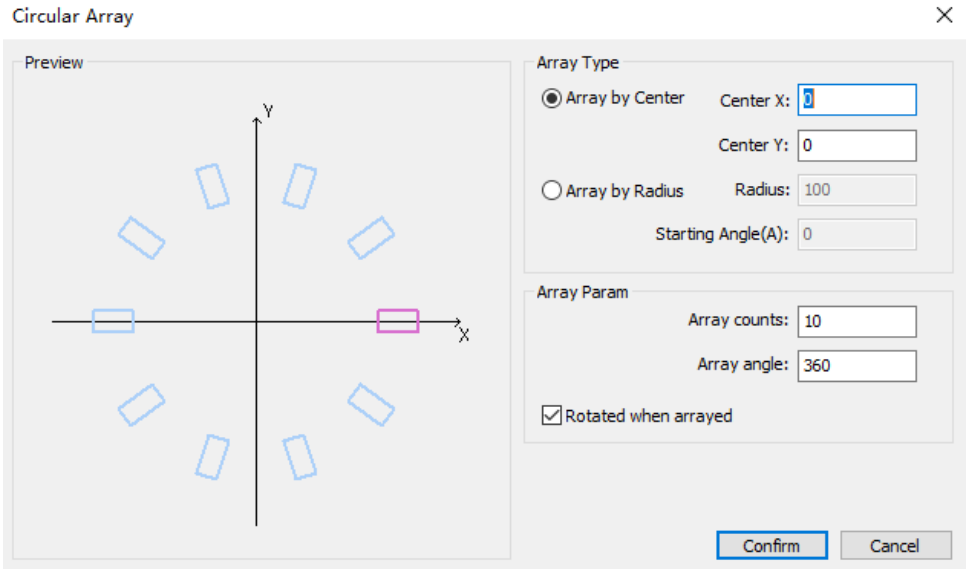
Circular array has two modes:

- Array by center: Make the array based on the selected center coordinates.
- Array by radius: Make the array around the currently selected graphic.

Operating Steps:

1. Select one or more graphics in the **2D Edit** page.
2. In the menu bar, click **Drawing** → 

to open the **Circular Array** dialog box:

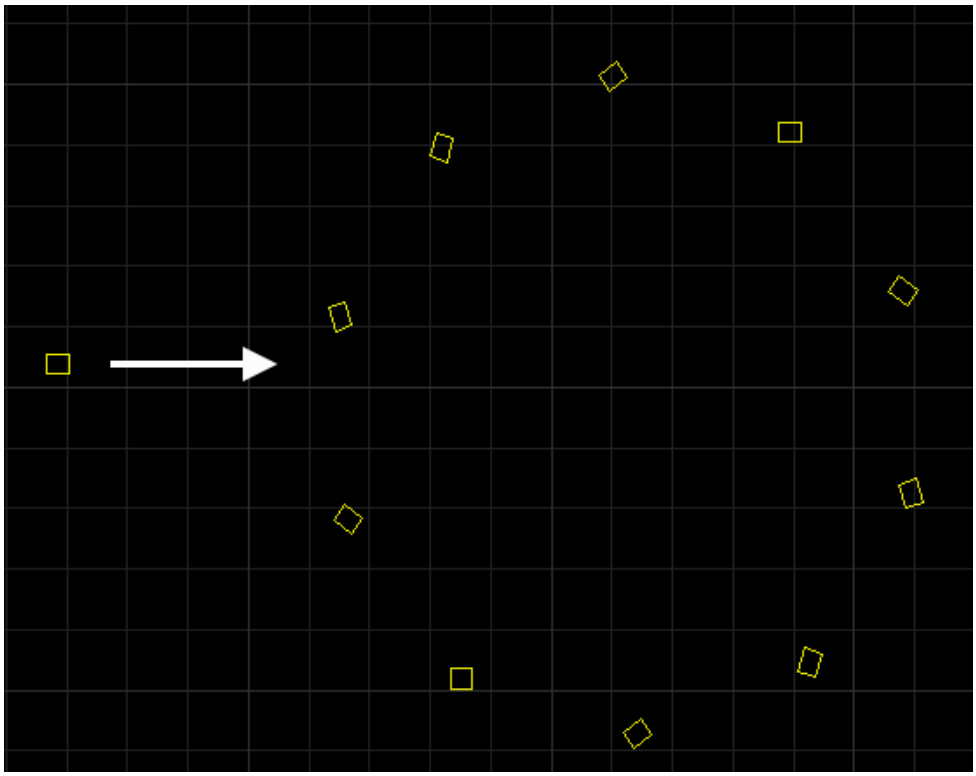


3. Set the array mode.

- Array by center: Check **Array by Center**. Set the rotation center point coordinates of the circular array.
- Array by radius: Check **Array by Radius**. Set parameter for **Radius** and **Stating Angle(A)**.

4. Set the parameters of the **Array Param** area.
   - **Array Counts**: The total number of copied circles.
   - **Array Angle**: Total column offset angle.

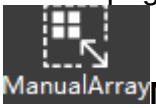
Effect picture:



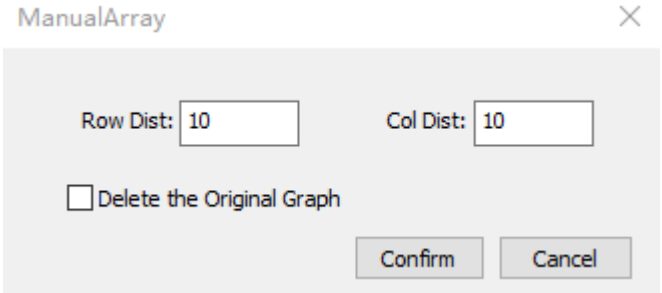
4.1.7.3. Manual Array

Manually select the area range of the array, and the graphics in this area are copied as a rectangle array.

Operating Steps:

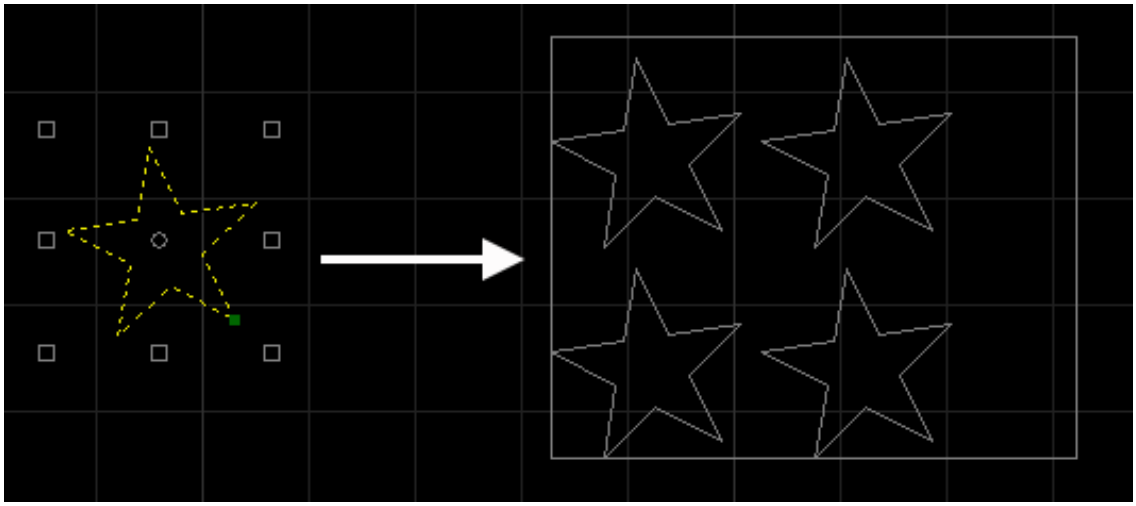
1. Select one or more graphs in the **2D Edit** page.
2. In the menu bar, click **Drawing** → 

Manual Array to open the **Manual Array** dialog box:



3. **（Optional：）** If need to delete the original graphic, check **Delete the Original Graph**.
4. Set **Row Dist** and **Col Dist**.
   - **Row Dist**: Translate left and right based on the graphic frame.
   - **Col Dist**: Translate up and down based on the graphic frame.
5. Click **Confirm** and the cursor becomes  .
6. Click the left mouse button to select the start position.
7. **（Optional：）** If need to reselect the start position, click the right mouse button.
8. Drag the mouse to select the destination and click the left mouse button.

Effect picture:



4.1.8. Auxiliary Tool

The system provides the following auxiliary tools.

- Property
- Measure
- Catch

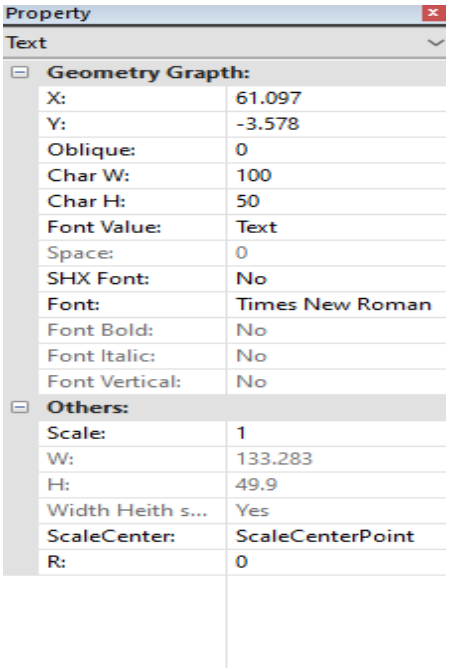
4.1.8.1. Property

You can change the size and tiling angle of a graphic by viewing and modifying the properties of a single or multiple graphics.

Operating Steps：

1. On the **2D Edit** page, select one or more graphics.
2. Select any of the following methods to open the **Property** page:
   - In the menu bar, click **Drawing**→ 

.
   - Right click and select **Property**.



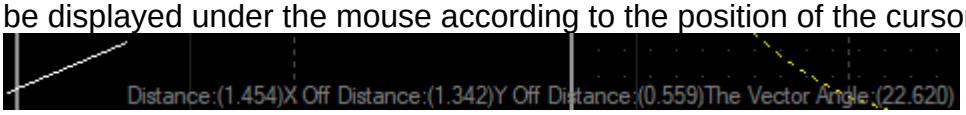
3. Modify Property Parameters:
   - Geometry graphic:
     - The X and Y values show the X axis and Y axis coordinates of the start point of the selected graphic.
     - The width and length of the rectangle are the width and length of the selected graphic.
     - Tilting angle is the angle of the graphic relative to the X axis.
   - Others:
     - You can view and modify the scale value to scale the size of the selected graphic.
     - The values of width and height change with the set scaling.
     - Scale center can be set center, left, right, up, bottom, left top, left bottom, right top, right bottom.
     - The R angle can be set manually.
4. After the modification is complete, click to close the **Property** page.

4.1.8.2. Measure

Measure the specified distance between two points, the X/Y offset, and the angle forward to the X axis.

Operating Steps:

1. In the menu bar of the **2D Edit** page, click **Drawing** → Measure and will appear behind the mouse.
2. Click the left mouse button to select the start point.
3. Move the cursor to the measurement end point, and the measurement results will be displayed under the mouse according to the position of the cursor.



4. Right click to exit the tool.

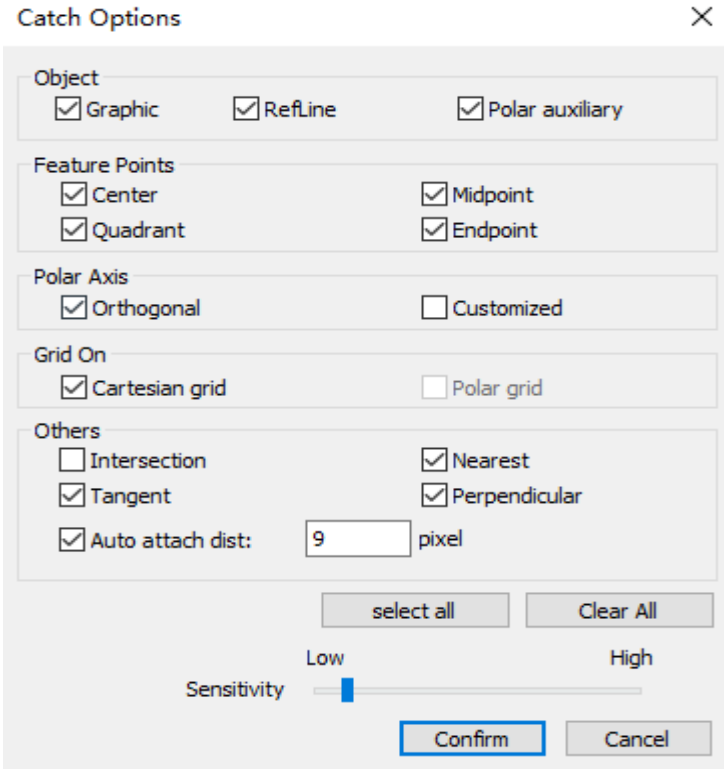
4.1.8.3. Catch

The feature points of existing graphics can be located more accurately when drawing objects.

When the mouse is close to the feature point, the system can easily catch it, which convenience the accurate connection between graphics.

Operating Steps：

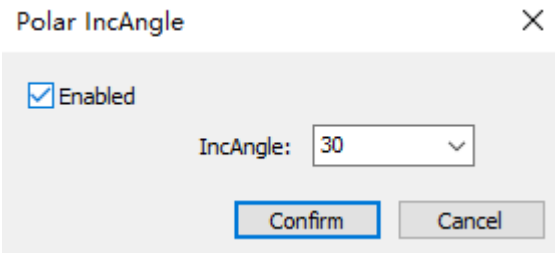
1. In the menu bar of the **2D Edit** page, click **Drawing** → click Catch → **Catch Options** and open the **Catch Options** dialog box:



2. Select the required options and set **Sensitivity**.
   The higher sensitivity can easier to catch feature points.
3. After setting, click **Confirm**.

   Note： If **Customized** is checked, follow the steps to set the incremental angle:

   a. In the **Tools** bar, click Polar IncAngle to open the **Polar IncAngle** dialog box:



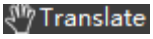
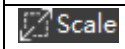
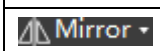
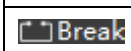
   b. Set incremental angle.

   The system will catch at the preset **Incremental Angle**. For each **Incremental Angle** rotation, the system will give the corresponding auxiliary line prompt.

   c. Click **Confirm**.

4.1.9. Edit Graphic

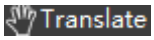
In the software can not only copy, paste, cut and delete the graphic, but also the following editing operations:

Tool	Name
	Translate
	Scale
	Rotate
	Mirror
	Align
	Join
	Explode
	Break

4.1.9.1. Translate

Translate the graphic means moving a graphic in a straight direction, changing its coordinate position without changing its shape or size.

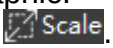
Operating Steps:

1. In the **2D Edit** page, select the graphic.
2. Select any of the following methods to translate graphic:
   - In the menu bar, click **Drawing** →  , then click anywhere in the graph, and click the left mouse button to select the target location.
   - Hold down the left mouse button to drag the graphic.

4.1.9.2. Scale

Scale the graphic means to scale the graphic equally and change the size of the graphic.

Operating Steps:

1. In the **2D Edit** page, select the graphic.
2. In the menu bar, click **Drawing** →  .
3. Click the left mouse button to select the scale center point.
4. Click the left mouse button to select the target point.
5. Move the cursor to adjust the scale ratio.
6. Click the left mouse button to confirm.

4.1.9.3. Rotate

Rotate the graphic means to rotate the graphic in any direction by any angle with a certain point as the selected center.

Operating Steps:

1. In the **2D Edit** page, select the graphic.
2. Select the following operations as required:
   - Follow these steps to rotate the graphic around any point:
     i. In the menu bar, click **Drawing** →  .
     ii. Click the left mouse button to select the rotate center point.
     iii. Move the cursor to adjust the rotation angle.

iv. Click the left mouse button to confirm.

- Hold down the **Ctrl** key and drag the rectangle point on any of the four corners of the node edit box.
- If you want to rotate 90 deg CW, click Rotate → **Rotate 90 deg CW** in the **Menu Bar**.
- If you want to rotate 90 deg CCW, click Rotate → **Rotate 90 deg CCW** in the **Menu Bar**.
- If you want to rotate180 deg, click Rotate → **Rotate 180 deg** in the **Menu Bar**.

4.1.9.4. Mirror

Mirror includes the following three modes:

- Mirror horizontal: With the vertical central axis of the graphic as the center, the left and right parts of the figure interchanged.
- Mirror vertical: With the horizontal axis of the graphic as the center, the upper and lower parts of the figure are interchanged.
- Mirror any angle: Take a line at any angle of the graphic as its axis, the left and right parts of the graphic can be interchanged and rotated by any angle.

Operating Steps:

1. In the **2D Edit** page, select the graphic.
2. Select the following operations as required
   - If you need mirror horizontal, click **Drawing** → Mirror → **Mirror Horizontal** in the menu bar.
   - If you need mirror vertical, click **Drawing** → Mirror → **Mirror Vertical** in the menu bar.
   - If you need mirror any angle, complete the following steps:
     i. Click **Drawing** → Mirror → **Mirror any Angle** in the menu bar.
     ii. Click the left mouse button to select the mirror center.
     iii. Move the cursor to adjust the rotation angle.
     iv. Click the left mouse button to confirm.

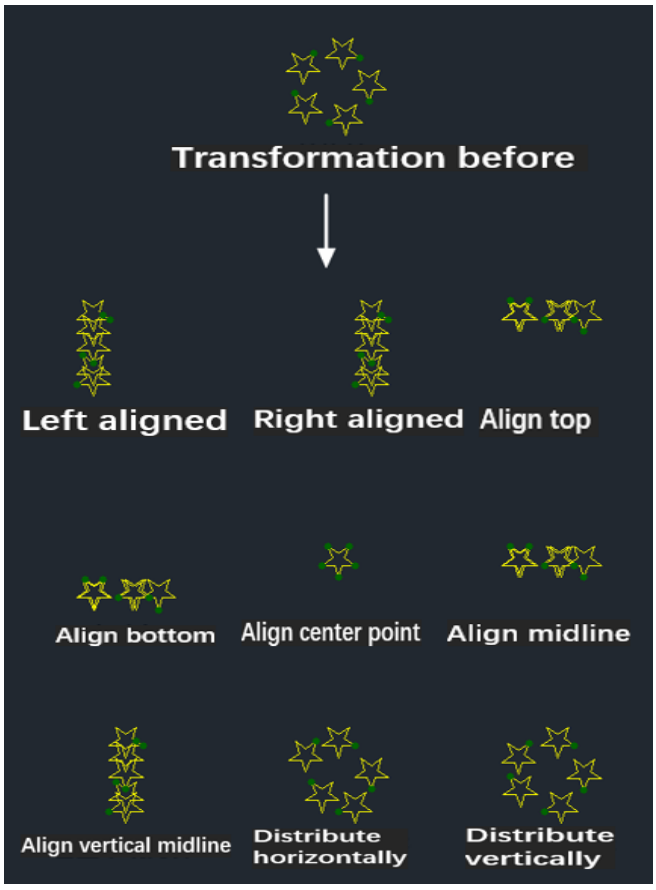
4.1.9.5. Align

Change the relative position of multiple graphics to align them.

Operating Steps:

1. In the **2D Edit** page, select multiple graphics.
2. Click **Drawing** → Align **Align** in the menu bar and select the alignment in the submenu:
   - Left aligned
   - Right aligned
   - Align top
   - Align bottom
   - Align center point
   - Align midline
   - Align vertical midline
   - Distribute horizontally
   - Distribute vertically

The system automatically performs the alignment. Effect picture:



4.1.9.6. Join

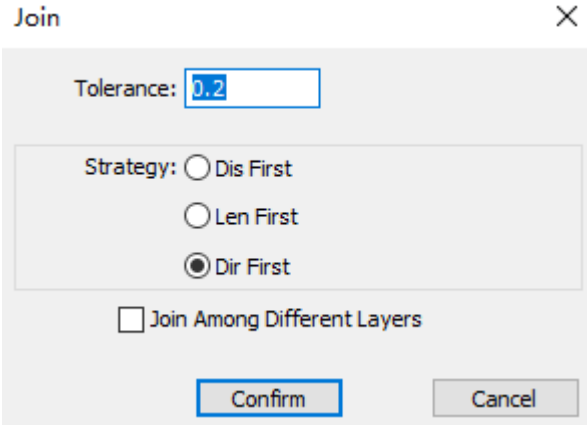
Join graphics are used to join unconnected graphics into a single graphic.

This function applies only to unclosed objects, non-text, and groups.

It is advised to open catch before use. For details, please click Catch.

Operating Steps：

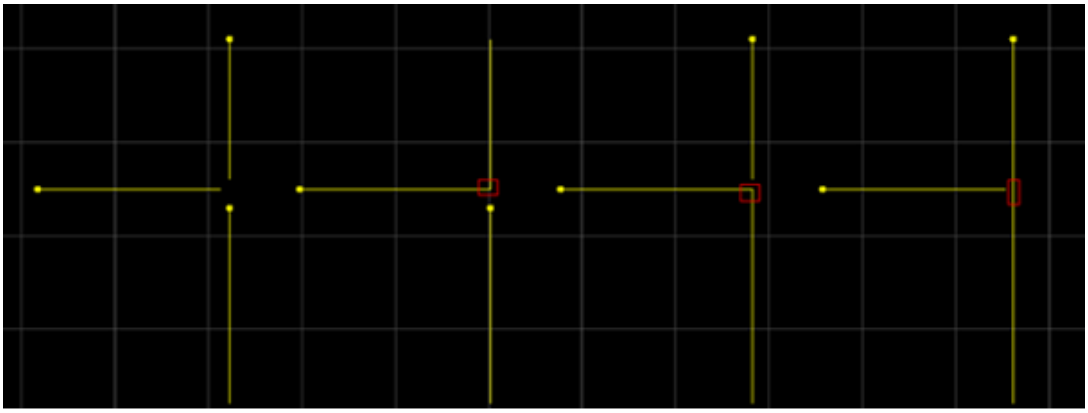
1. In the **2D Edit** page, select multiple graphics.
2. Select any of the following methods to open the **Join** dialog box:
   - In the menu bar, click **Drawing** → Join.
   - Click the right mouse button and click **Join**.



3. Enter the value in the **Tolerance** settings box.
   Tolerance is the maximum value of spacing between graphics that needs to be satisfied for joining.

Defaults merge tolerance range: [0.01, 10]mm.

4. To set join **Strategy**, click **Dis First/ Len First/ Dir First**.
   When more than three endpoints in the same join position meet the join tolerance, the graphic with the closest join distance/longest join length/same direction is preferred. Graphics will preferentially join nearest/longest/same direction.
   Effect picture:



5. **（Optional：）** If need to join two lines on different layers, check **Join Among Different Layers**.

4.1.9.7. Explode

Delete redundant lines to achieve the purpose of modifying tool path, mostly used for polylines.

When used together with Join, it can correct the errors in drawing graphics and ensure the quality of machining.

Explode has the following characteristics:

- When the object is a graphical group, **Explode** is equivalent to **Dissolving the Group**.
- When the object is text, **Explode** is equivalent to **Turning Text into Graphics**.

Operating Steps：

1. In the **2D Edit** page, select the graphic.
2. In the menu bar, click **Drawing** → Explode.

4.1.9.8. Break

Used to break the graphic into multiple polylines. There are two methods to break graphic:

- Auto: Break the selected object automatically based on the value set.
- Manual: Manually select the break location to perform the break on single graphic at one time.

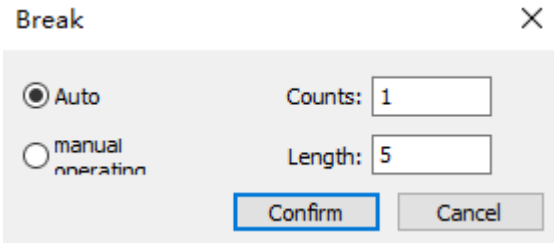
Common usage scenarios:

- Break can make the cut part connected with the surrounding material, which has the same effect as micro joint.
- When drawing the graphic, remove the excess graphic to facilitate the appearance of the ideal shape.

Operating Steps：

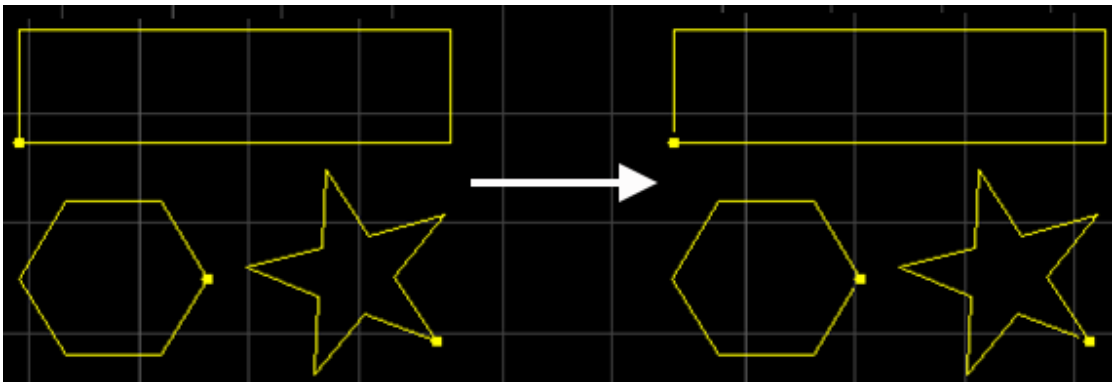
1. In the **2D Edit** page, select the graphic.

2. In the menu bar, click **Drawing** → Break.



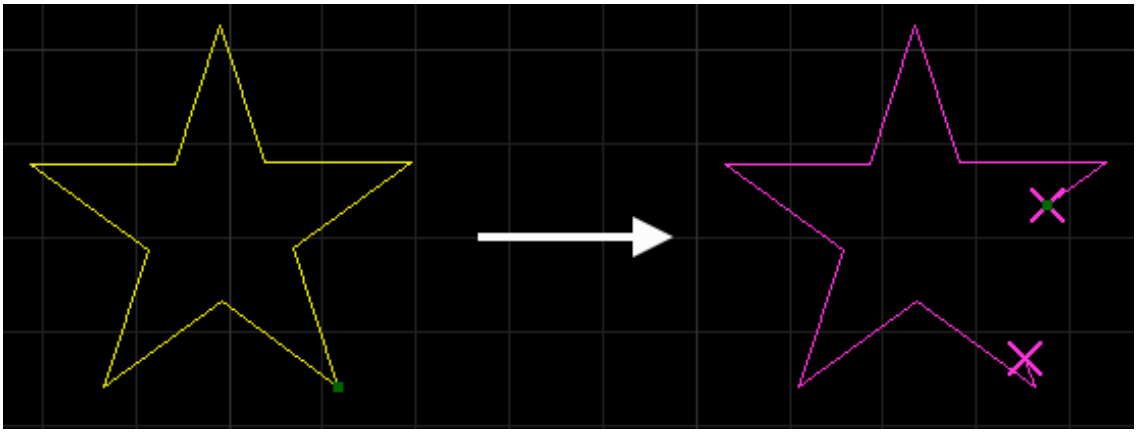
3. Perform the following operations based on the break mode selected:
   - Automatic break mode:
     i. Click **Auto**. Enter **Counts** and **Length**.
     ii. Click **Confirm**.

   Effect picture:



   - Manual break mode:
     i. Click **Manual**. Enter **Length**.
     ii. Click **Confirm**, the cursor will become .
     iii. Click the left mouse button to select the break location.
     iv. Click the right mouse button to manually break.

   Effect picture:



4.1.10. Preprocess Graphic

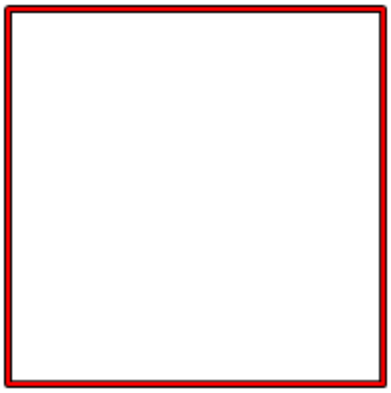
The preprocessing operation of the graphics makes the graphics achieve better machining effect.

The system supports single pretreatment of drawings and batch pretreatment of multiple projects.

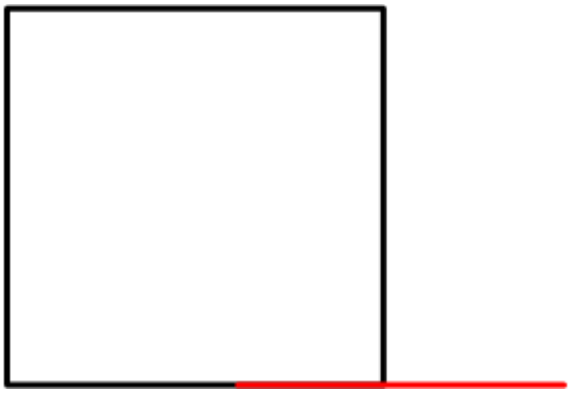
4.1.10.1. Delete Overlap lines

Objects of overlap lines can be deleted as follows:

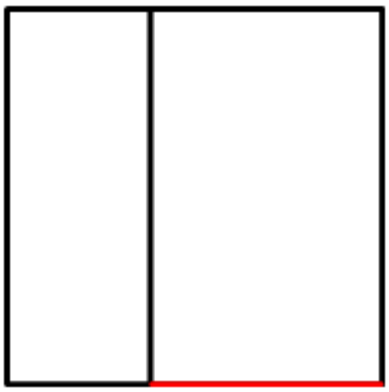
- Two completely coincident graphics:



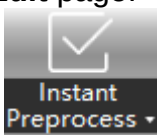
- The overlap between a line segment and the graphic:



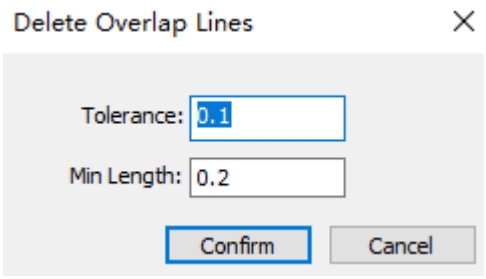
- The overlap of the graphic itself:



Operating Steps:

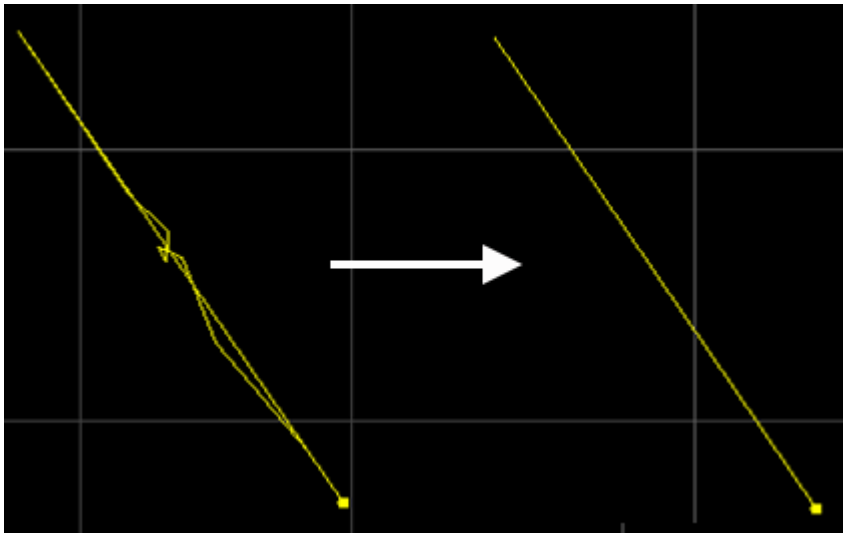
1. Select multiple graphics in the **2D Edit** page.
2. In the menu bar, click **Common** → 

→ **Delete Overlap lines** to open the **Delete Overlap lines** dialog box:



3. Set the following:
   - **Tolerance**: The distance between the two lines is within the maximum tolerance range.
   - **Min Length**: The coincidence length of two lines is greater than the minimum length.

Effect picture:

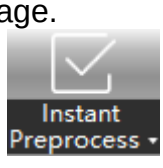


4.1.10.2. Simplify Curve

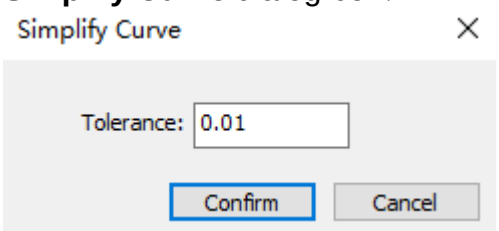
A single object consisting of many lines and arcs in which nodes control and adjust the shape of the curve.

Simplify curve means that the system automatically reduces the excess part in the graphic within the tolerance range, to speed up the response speed of the graphic operation.

Operating Steps:

1. Select the graphic in the **2D Edit** page.
2. In the menu bar, click **Common** → 

→ **Simplify Curve** to open the **Simplify Curve** dialog box:



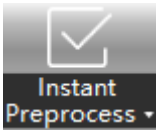
3. Enter **Tolerance** value in the box and click **Confirm**.

Curve Simplifying Successful will appear in the **Information Bar**.

4.1.10.3. Smooth Curve

Smooth a number of polylines to ensure smooth machining.

Operating Steps：

1. Select the graphic in the **2D Edit** page.
2. Select either of the following to smooth curve：
   - In the menu bar, click **Common** → 

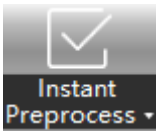
→ **Smooth Curve**.
   - Click the right mouse button and click **Smooth Curve**.

   Curve Smoothing Successful will appear in the **Information Bar**.

4.1.10.4. Convert Text to Graphic

It is used to convert text into polylines, and technics can be added later.

Operating Steps：

1. Select the graphic in the **2D Edit** page.
2. Select either of the following to convert text to graphic：
   - In the menu bar, click **Common** → 

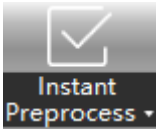
→ **Convert Text to Graphic**.
   - Click the right mouse button and click **Convert Text to Graphic**.

   The number of graphics selected in the **Information Bar** will increase.

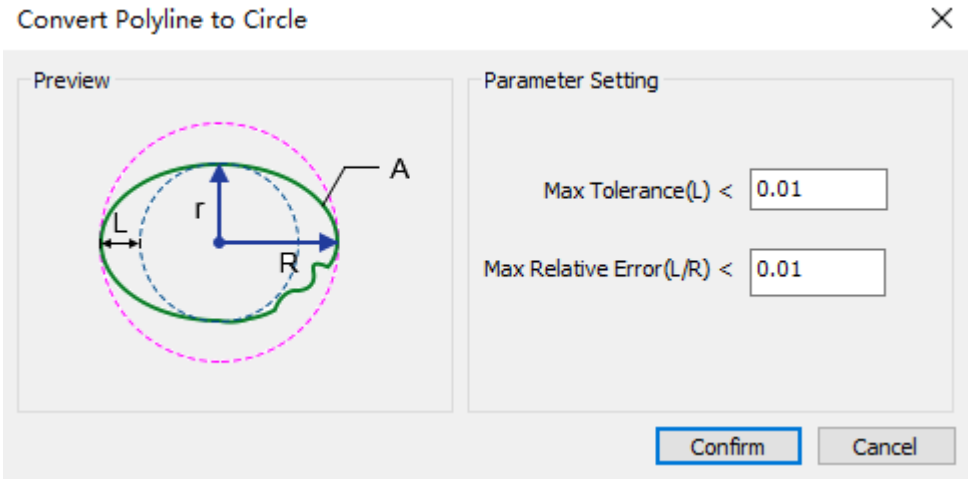
4.1.10.5. Convert Polyline to Circle

Used to convert a closed polyline that looks like a circle to a circle.

Operating Steps：

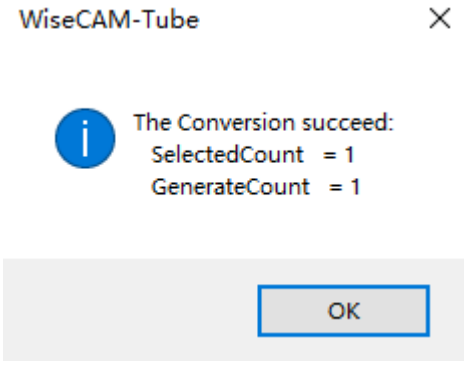
1. Select the graphic in the **2D Edit** page.
2. Select either of the following to open the **Convert Polyline to Circle** dialog box：
   - In the menu bar, click **Common** → 

→ **Convert Polyline to Circle**.
   - Click the right mouse button and click **Polyline to Circle**.

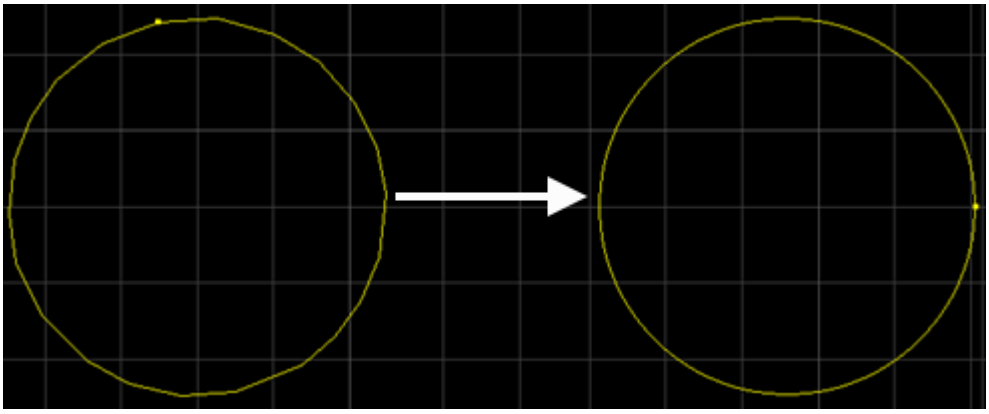


3. Enter **Max Tolerance** and **Max Relative Error** in the box.

4. Click **Confirm** and the following dialog box will pop up after the conversion is successful:



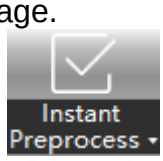
Effect picture:



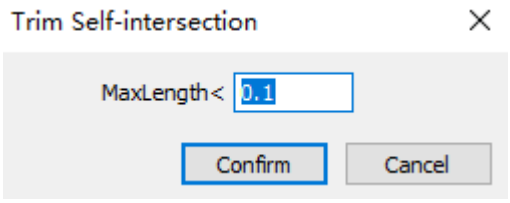
4.1.10.6. Trim Self-intersection

Split the self-intersecting polyline graphic and cut out the excess lines.

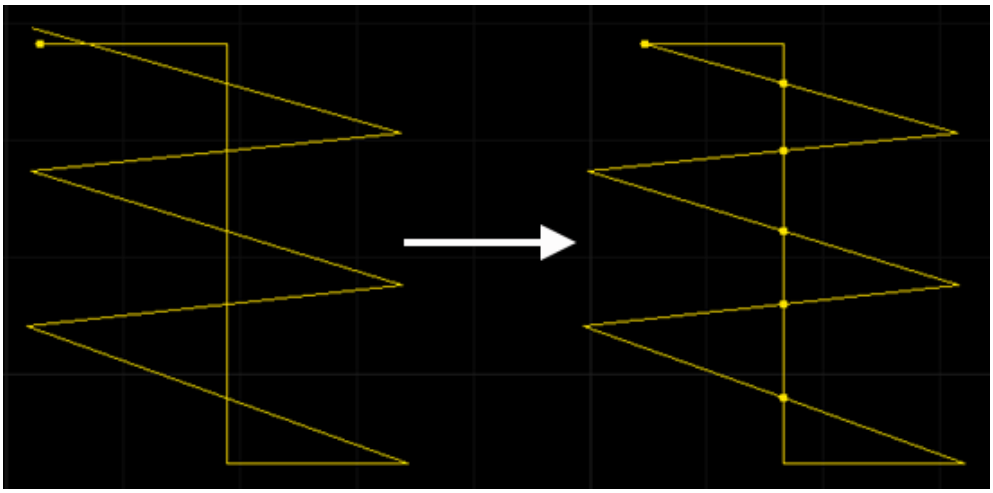
Operating Steps:

1. Select the graphic in the **2D Edit** page.
2. In the menu bar, click **Common** → 

→ **Trim Self-intersection** to open the **Trim Self-intersection** dialog box:



3. Set the length and click **Confirm**. The software will automatically cut out the lines within the length range.
   Effect picture:



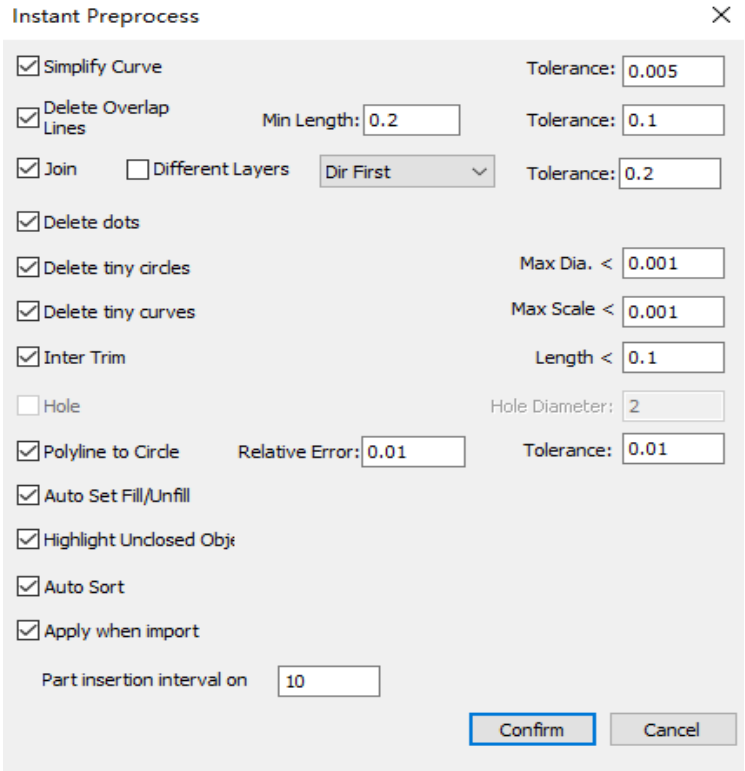
4.1.10.7. Instant Preprocess

The instant preprocess function sets error-prone items in the common graphics machining process. Users can automatically process the graphics by selecting the items to be processed according to their needs, so that the graphics can achieve better machining effects.

The instant preprocessing function items mainly include: simplify curve, delete overlap lines, join, delete dots, delete tiny circles, delete tiny curves, inter trim, polyline to circle, auto set fill/unfill, etc.

Operating Steps：

1. Select the graphic in the **2D Edit** page.
2. Select either of the following to open the **Instant Preprocess** dialog box:
   - In the menu bar, click **Common** →  .
   - Click the right mouse button and click **Instant Pre-process**.



3. Check the desired option and enter a value.
4. If you want to automatically process graphics when importing files according to the above options, select **Apply When Importing**.
5. Set the **Part Insertion Interval On**. When inserting a part with the base layer of **Nest Result List**, the distance between the part and the existing part.
6. Click **Confirm** and use instant preprocess function.

4.1.11. Quick Edit

In the **2D Edit** page, the **Quick Edit** function button gathers common functions. The function entries are as follows: **Quick Edit** refers to the collection of common editing functions for the convenience of users. It provides the following functions:

- Tube Intersection
- Tube Cutting
- Tube Partition: The function entry is in the **2D Edit** page.
- Tube Cutting Setting
- Part Mark: The function entry is in the **2D Edit** page.

The function entries are as follows:

- At the entrance of **Software Main Interface**

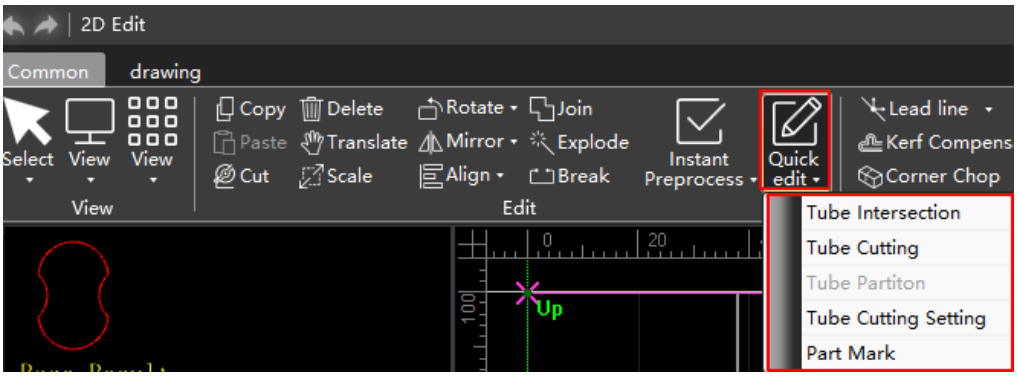
  In the common toolbar of **Software Main Interface**, click  **Tube Intersection** /  **Tube Cutting** /  **Tube Cutting Setting**.

- At the entrance of the **2D Edit** page

  In the menu bar of the **2D Edit** page, click **Common** → 

→ **Tube Intersection** / **Tube Cutting** / **Tube Partition** / **Tube Cutting Setting** / **Part Mark**.



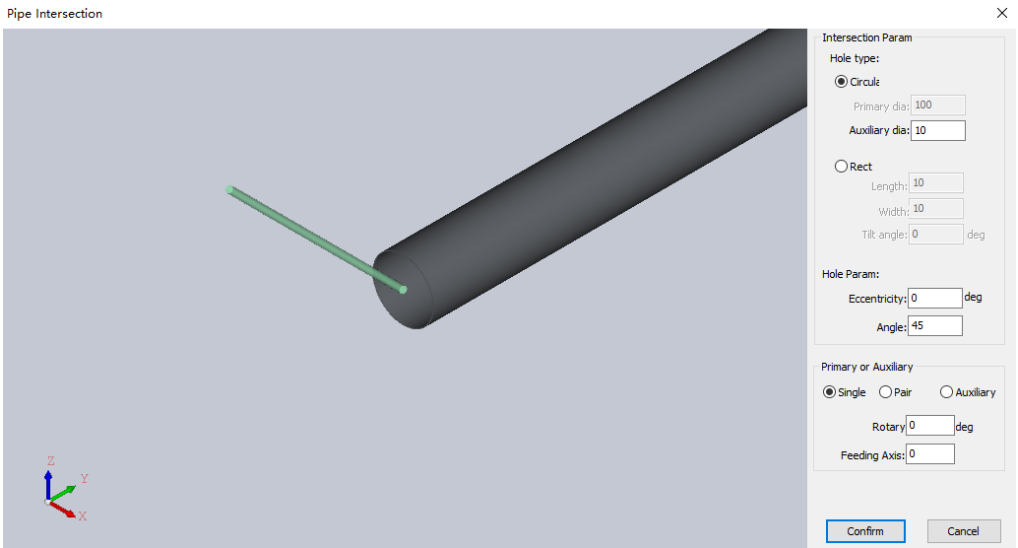
4.1.11.1. Tube Intersection

Generate intersecting hole tool path on tube, including circular hole and rectangular hole.

This section takes the intersection of circular tube as an example.

Operating Steps：

1. Select any of the following methods to open the **Pipe Intersection** dialog box:
   - In the common toolbar of **Software Main Interface**, click  **Tube Intersection**.
   - In the menu bar of the **2D Edit** page, click **Common** →  → **Tube Intersection**.



2. Select hole type and set hole param:

Parameter	Description
Circular	Type of cut hole.
Rect	Type of cut hole.
Primary dia	The diameter of the main tube, that is, the diameter of the tube to be cut. ▪ When the cutting type is selected as **Single** or **Pair**, it cannot be set. The fixed value is the tube diameter. ▪ It can be set when the cutting type is selected as **Auxiliary**.

Parameter	Description
Auxiliary dia	The diameter of the branch tube, that is, the diameter of the tube that needs to run through the main tube. The setting shall meet the following requirements: branch tube diameter ≤ main tube diameter. ▪ It can be set when the cutting type is **Single** or **Pair**. ▪ When the cutting type is selected as **Auxiliary**, it cannot be set. The fixed value is the tube diameter.
Eccentricity	The distance between the main tube centerline and the branch tube centerline. The maximum value should be **± (Main Tube Diameter - Branch Tube Diameter)/2**.
Angle	The inclination angle when the main tube intersects the branch tube. Setting range: 5 º~175 º.

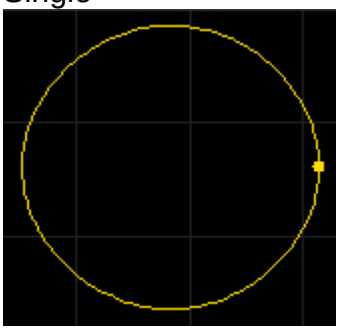
3. In the **Primary or Auxiliary** area, set the following parameters:

Parameter	Description
Hole type	You can select to generate **Single**, **Pair** or **Auxiliary** circular holes.
Rotary	The rotation angle of the cutting graphic relative to the workpiece origin.
Feeding axis	The feed axis distance of the cutting graphic relative to the workpiece origin.

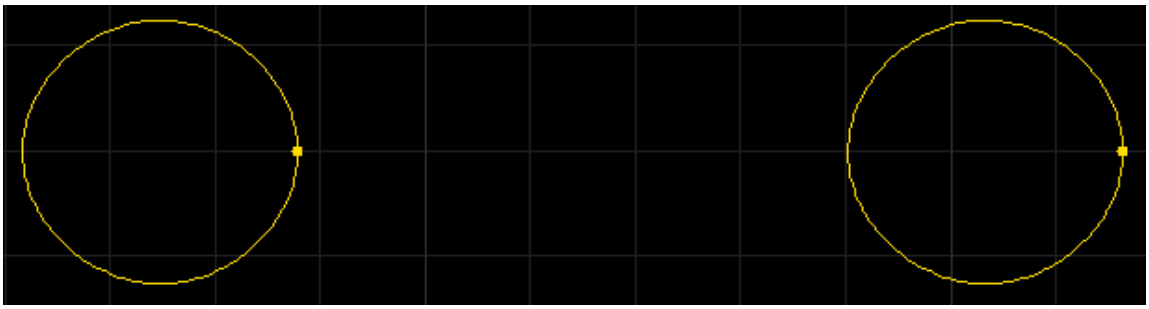
4. Click **Confirm**.

Effect picture:

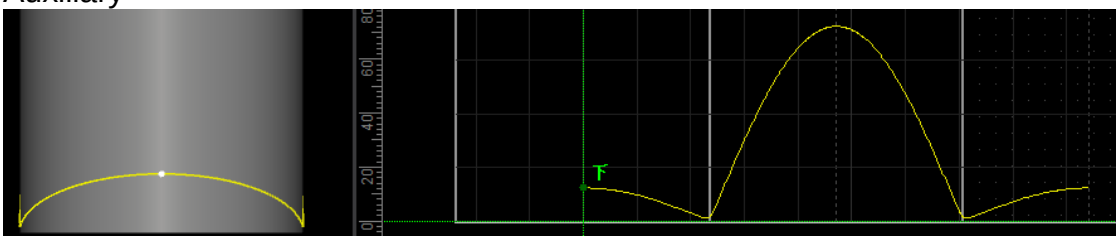
- Single



- Pair



- Auxiliary



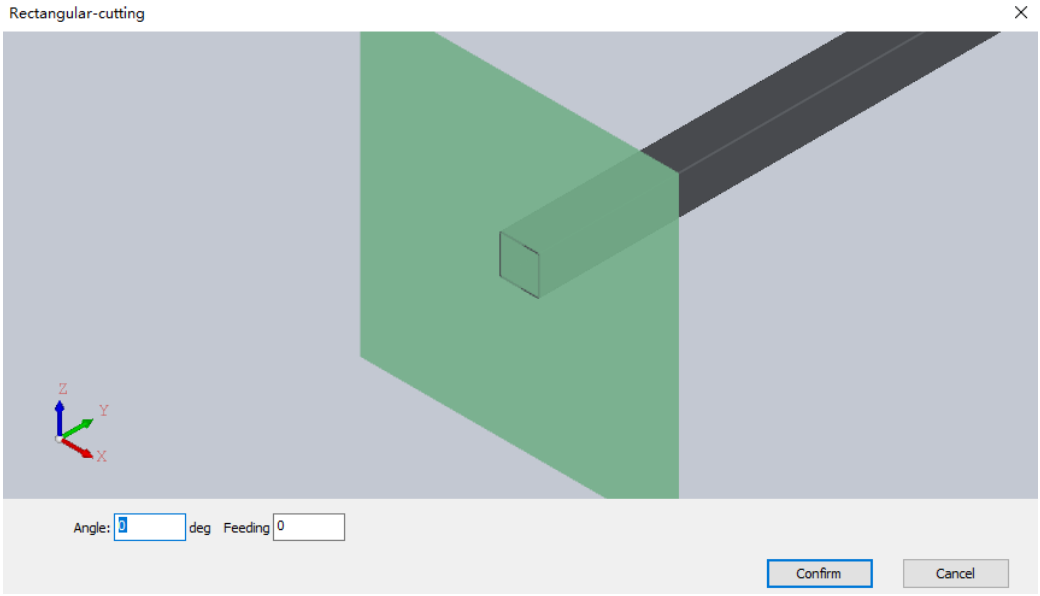
4.1.11.2. Tube Cutting

Generate a cutoff path on the tube.

This section takes rectangular tube cutting as an example.

Operating Steps：

1. Select any of the following methods to open the **Rectangular-cutting** dialog box:
   - In the common toolbar of **Software Main Interface**, click **Tube Cutting**.
   - In the menu bar of the **2D Edit** page, click **Common** → Quick edit → **Tube Cutting**.



2. Set parameters for angle and feeding axis:

Parameter	Description
Angle	The angle between the cutting surface and the tube centerline. Setting range: 5 º~175 º.
Feeding	Feed axis coordinates of the cutting position on the tube.

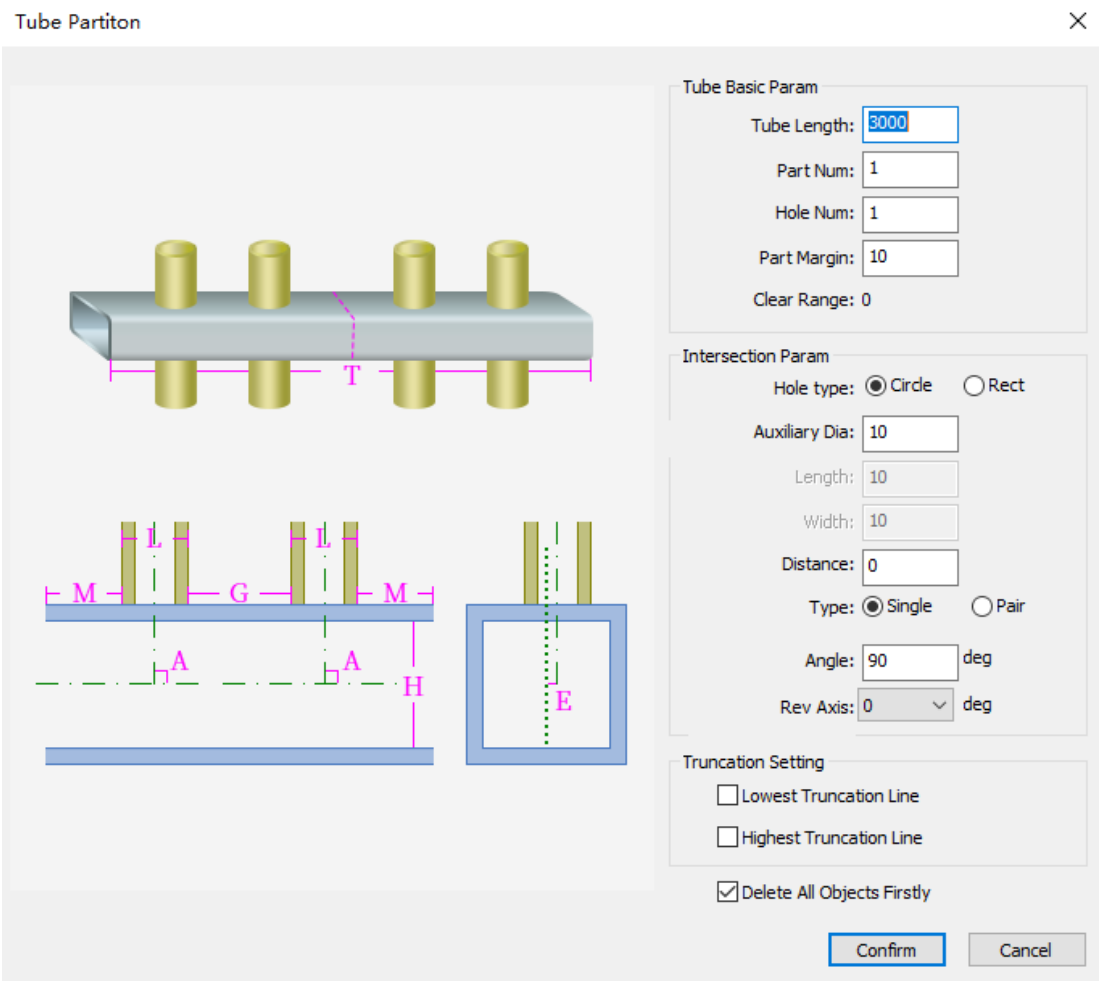
3. Click **Confirm**.

4.1.11.3. Tube Partition

Generate intersecting and cutting array tool paths on tubes.

Operating Steps：

1. In the menu bar of the **2D Edit** page, click **Common** → Quick edit → **Tube Partition** to open the **Tube Partition** dialog box:



1. In the **Tube Basic Param** area, set the following parameters:

Parameter	Description
Tube length	Set the length of the tube.
Part num	The number of parts set on the tube.
Hole num	Number of holes per part.
Part margin	The distance reserved between parts.

Note

Clear range: After setting the **Tube Partition** parameter, you need to ensure that the clear range is less than 0.

1. In the **Intersection Param** area, set the following parameters:

Parameter	Description
Hole type	Select **Circle** or **Rect**.
Auxiliary dia	**Circle** parameter, the diameter of the branch tube, that is, the diameter of the tube that needs to run through the tube.

Parameter	Description
Length	**Rect** parameter, the length of the rectangular hole.
Width	**Rect** parameter, the width of the rectangular hole.
Distance	The distance between the tube centerline and the branch centerline.
Type	Optional when the cutting type is **Circle**. You can select to generate **Single** or **Pair** circular holes.
Angle	The inclination angle when the tube intersects the branch tube.
Rev axis	The rotation angle of the cutting graphic relative to the workpiece origin.

2. Check the following options in the **Truncation Setting** area as required:
   - Lowest truncation line
   - Highest truncation line
3. Check **Delete All Objects Firstly** as required.
4. Click **Confirm**.

4.1.11.4. Tube Cutting Setting

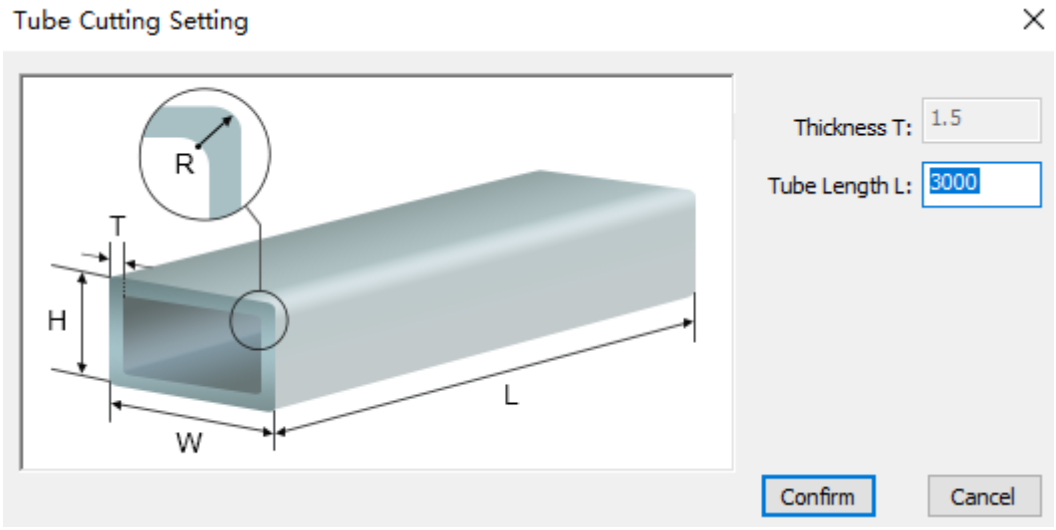
Set the type and size of the machining tube in the software according to the actual situation.

This section takes rectangular tubes as an example.

Operating Steps:

1. Select any of the following methods to open the **Tube Cutting Setting** dialog box:
   - In the common toolbar of **Software Main Interface**, click  **Tube Cutting Setting**.
   - In the menu bar of the **2D Edit** page, click **Common** → 

→ **Tube Cutting Setting**.



2. Modify the tube length.
3. Click **Confirm**.

4.1.11.5. Part Mark

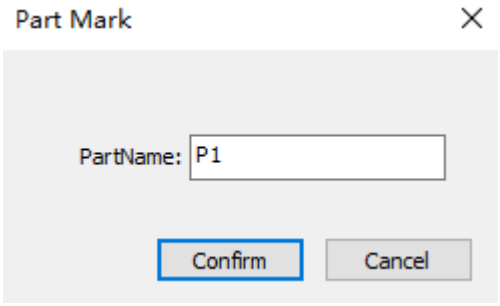
Modify the part name.

Operating Steps：

1. In the menu bar of the **2D Edit** page, click **Common** → 

to check **Show Part Mark**.
2. Click 

→ **Part Mark** to open the **Part Mark** dialog box:



3. Set the part name.
4. Click **Confirm**, and the mouse becomes to  .
5. Click the line marked with the part to change the corresponding part name.
6. Click the right mouse button to exit the function.

4.2. Layer Technic

4.2.1. Overview

The layer function is mainly used to set the layer process. That is, parameters such as cutting and piercing, so as to ensure the cutting effect.

The system provides seven color layers. Each layer can be set with different layer technics. By default, the layer technics of the same color object are the same.

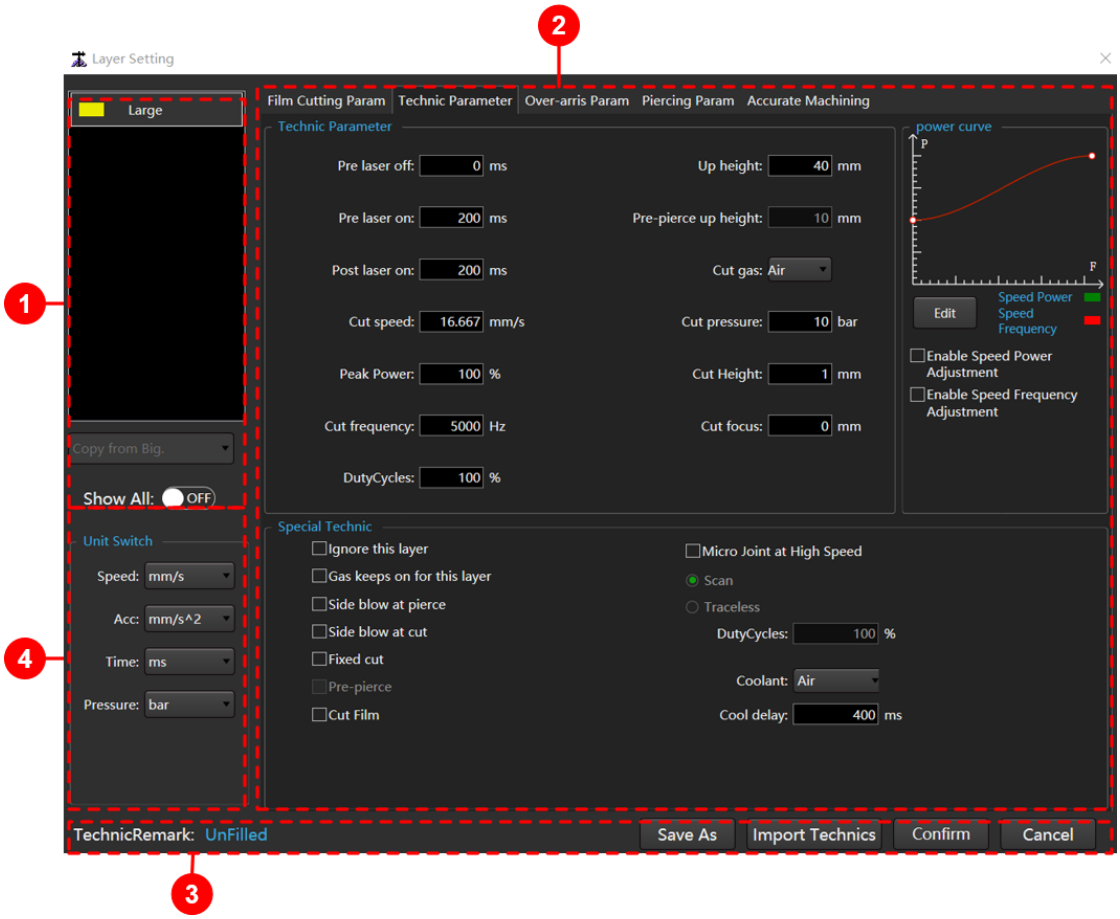
The system provides the management function of layer technic library, which can create new layer technics, modify, backup, restore and delete existing layer technics, and add and delete material types.

4.2.2. Set Layer Technic

4.2.2.1. Layer Parameter

In the layer bar of the **Software Main Interface** or **2D Edit** page, click 

to open the **Layer Setting** dialog box:



1.Layer selection box 2. Parameter setting page 3. Technic operation bar 4. Unit switch

- **Layer Selection Box**
  The following operations can be performed in the layer selection box:
  - Click the layer: Set the layer technic of the target layer.
  - Click to select the target layer, select a layer in the **Copy From XX** drop-down box, copy and apply the layer parameters to the target layer.

    Note

    When this operation is performed, multiple layers must exist in the current tool path.

  - Set **Show All** to **ON** status: The system displays all layers; Otherwise, only the layers contained in the current path file are displayed.
- **Parameter Setting Page**
  Click the page switch button above to switch to the corresponding parameter setting page, including:
  - **Film Cutting Param**
    Page: Set the film cutting parameters of the layer (pre laser on, post laser on) and the over arris parameters (power, frequency, etc.).
    For details, see Set Film Cutting Parameter.
  - **Technic Param**

Page: Set the technic parameters of the layer (pre laser off, up height, etc.) and special technic parameters (side blow, fixed cut, etc.).
For details, see Set Technic Parameter.

- **Over-arris Param**
  Page: Set the over-arris parameters of the layer (peak power, pressure, etc.).
  For details, see Set Over-arris Parameter.
- **Piercing Param**
  Page: Set piercing parameters (increment speed, peak power, etc.).
  For details, see Set Piercing Parameter.
- **Accurate Machining**
  Page: Set accurate machining parameters (slowly start and slowly stop).
  For details, see Set Accurate Machining.

- **Technic Operation Bar**
  Operation: Save as technic, import technic, save or cancel modification of current layer parameters.
- **Unit Switch**
  Switch the unit of the parameter here as needed.

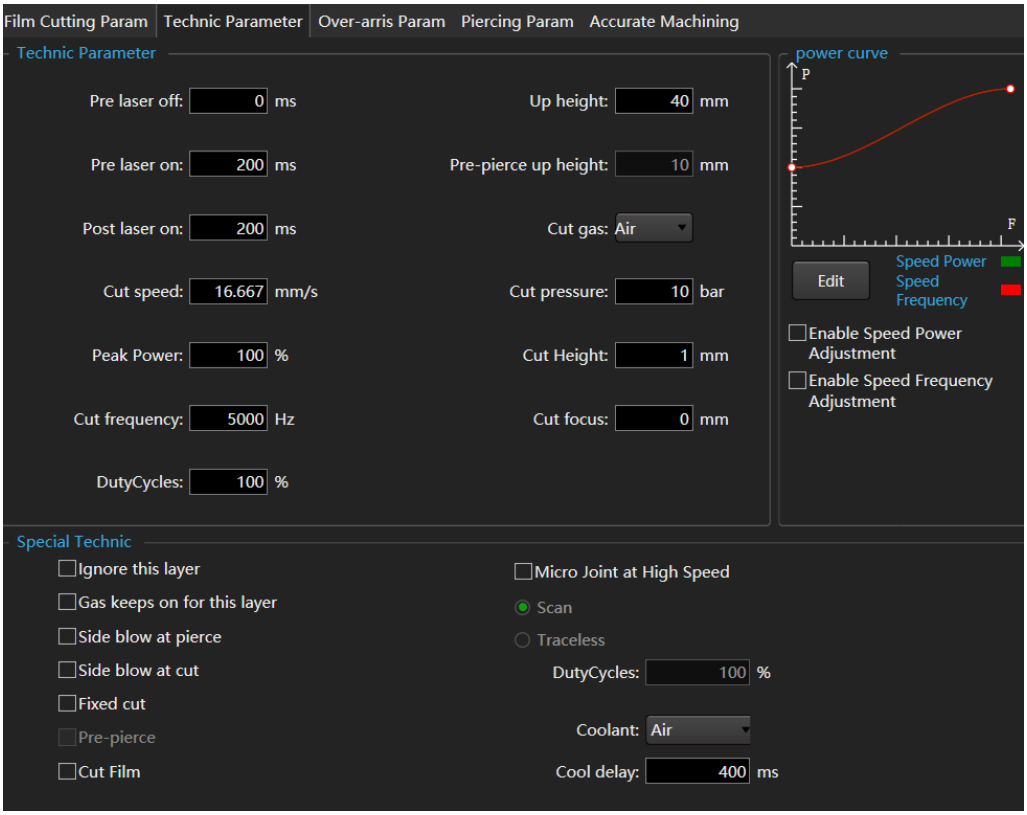
4.2.2.2. Set Operation

This includes the operations of assigning and setting layers and setting layer technics.

Operating Steps:

1. In the layer bar of the **Software Main Interface** or **2D Edit** page, click 

to open the **Layer Setting** dialog box:



2. **（Optional：）** If the target technic file needs to be imported from the technic library, execute the import technic operation. For details, see Import Technics.
3. Click the layer to be set in the **Layer Selection Box**.
4. **（Optional：）** If the layer parameters currently set are similar to other layers, select a layer in the **Copy From XX** drop-down box, copy and apply the layer parameters to the target layer.
5. **（Optional：）** In the **Unit Switch** area, switch the unit of the parameter as required.
6. In the **Parameter Setting Page** area, set the technic parameters. For details, see the following:
   - Set Film Cutting Parameter.
   - Set Technic Parameter, it includes setting technic parameters and editing power curve.
   - Set Over-arris Parameter.
   - Set Piercing Parameter.
   - Set Accurate Machining.
7. After setting the parameters, repeat steps 3 to 6 as needed to set the parameters of the next layer.
8. Click **Confirm** to close the **Layer Setting** dialog box and complete the layer technic setting.

Related Tasks：

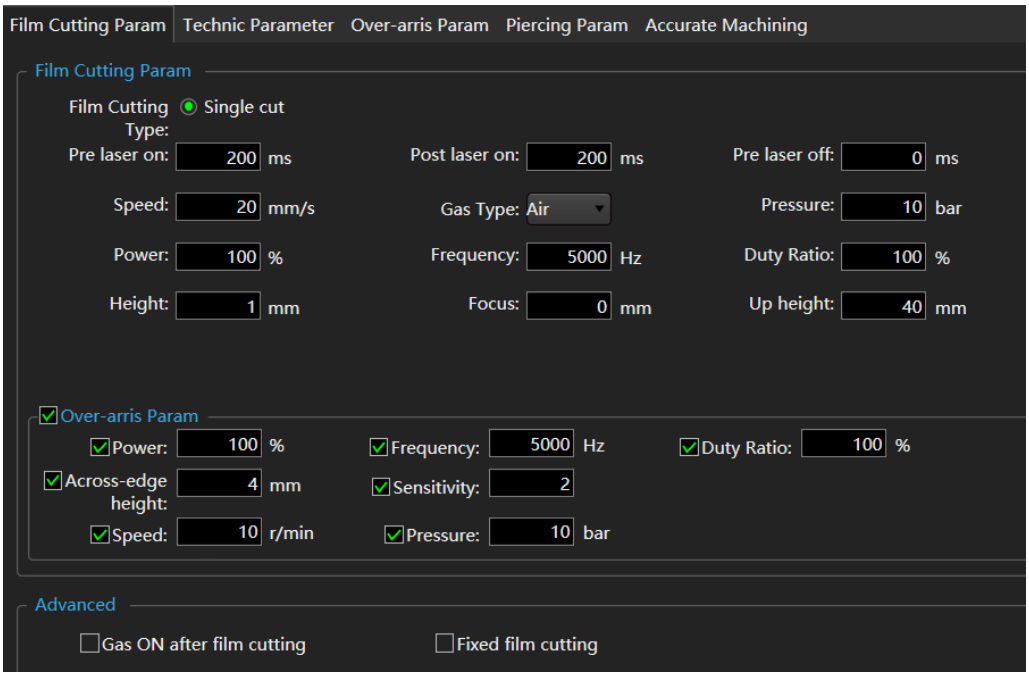
- Save as technic: Save the current technic to the technic library. For details, see Save As.
- Apply layer technic: For details, see Apply Layer Tchnic.
- Technic library management: For details, see Manage Technic Library.

4.2.2.2.1. Set Film Cutting Parameter

It is used to select the film cutting type and cut the protective film on the tube surface.

Operating Steps：

1. In the **Layer Setting** dialog box, click **Film Cutting Param** to switch to the **Film Cutting Param** page:



2. Set parameters. The parameters are described as follows:

   Parameter description of film cutting parameter area:

Parameter	Description
Film cutting type (single cut)	A single graphic is cutting first, and then the graphic is machined.
Pre laser on	Turn on the laser before delay. It can be set to a negative number to turn off the laser in advance to solve the problem of over burning at the end of the thin tube.
Post laser on	After turned on the laser, continue to set the time before performing the next step.
Pre laser off	Turn off the laser before delay.
Speed	The actual speed of film cutting.
Gas type	The type of gas used of film cutting.
Pressure	When cutting the film, the gas pressure should be used with the proportional valve.
Power	Peak power when cutting film.
Frequency	When cutting the film, the carrier frequency of the PWM modulation signal is also the number of light output in one second.
Duty Ratio	The duty ratio of cutting film is setting by adjusting the laser with PWM.
Height	The height of the nozzle from the tube when cutting film.
Focus	This takes effect after focus control is enabled. Position of focal point when cutting film.
Up height	The height of the laser head when switch cutting film.

   Parameter description of over-arris parameter area:

Parameter	Description
Power	Peak power when over-arris cutting.
Frequency	When over-arris cutting, the carrier frequency of the PWM modulation signal is also the number of light output in one second.
Duty Ratio	The duty ratio of over-arris cutting.
Across-edge height	The height of the nozzle from the tube when over-arris cutting.
Sensitivity	Follow the torch lifter dynamic sensitivity when over-arris cutting.
Speed	The speed of the axis of rotation used when over-arris cutting. （In case of over arris, the rotation axis speed is controlled separately. In other cases, the rotation axis speed is planned together with other axes according to the cutting speed）.
Pressure	The gas pressure of over-arris cutting should be used with proportional valve or multi-valve.

Parameter description of advanced area：

Parameter	Description
Gas on after film cutting	The gas output port is always open during the film cutting process and the film cutting to cutting process.
Fixed film cutting	The film will be cut at the **Fixed Height Position** in the common parameters.

3. **（Optional：）** If the film needs to be cut at a fixed Z-axis coordinate at all times, select the **Fixed Film Cutting**.
4. **（Optional：）** If the gas output port should always be open during the film cutting process and the film cutting to cutting process, select the **Gas On after Film Cutting**.

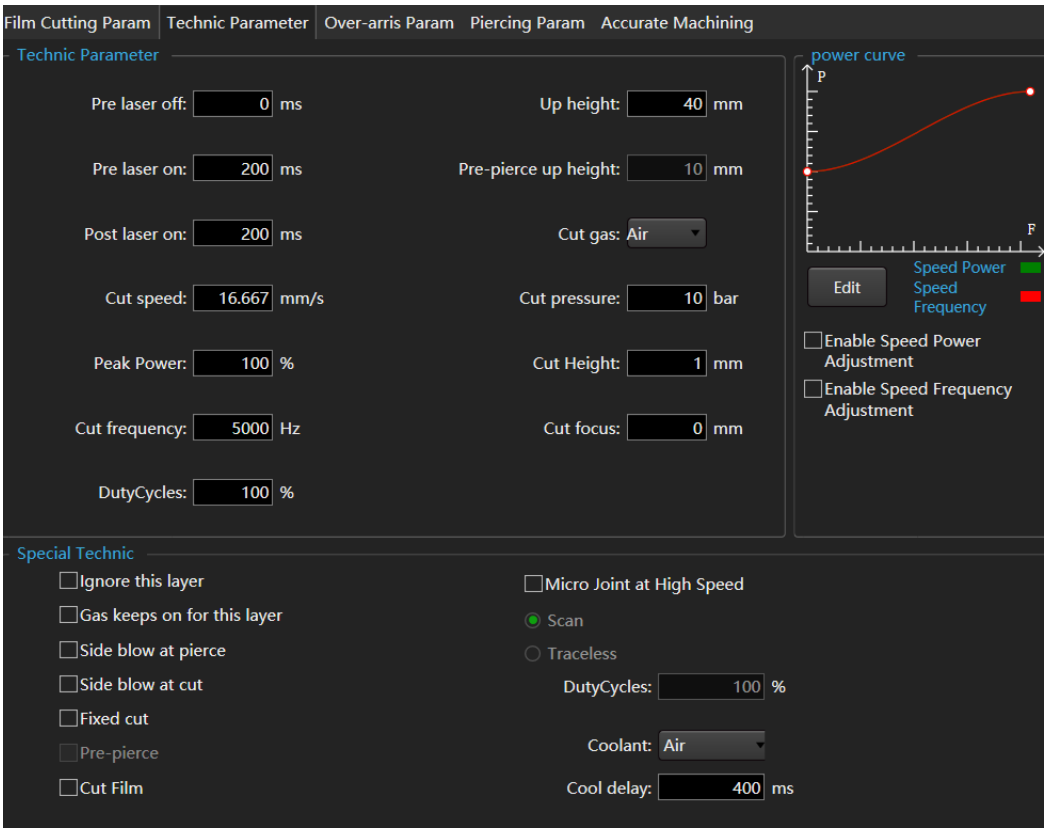
4.2.2.2.2. Set Technic Parameter

Used to set technic parameters, special technic parameters and edit power curve.

By editing the power curve, the problems such as over-burning of sharp corners and inconsistency of cutting effect with different thickness in laser cutting can be solved. The cutting power can be adjusted to change with the cutting speed, so as to ensure that the heat power absorbed per unit area is the same and the ideal cutting effect can be achieved.

Operating Steps：

1. In the **Layer Setting** dialog box, click **Technic Param** to switch to the **Technic Param** page：



2. Set parameters. The parameters are described as follows:
   Parameter description of technic parameter area:

Parameter	Description
Pre laser off	Turn off the laser before delay.
Pre laser on	Turn on the laser before delay.
Post laser on	After turned on the laser, continue to set the time before performing the next step.
Cut speed	The actual cutting speed.
Peak power	Adjust the laser by analog quantity, set the laser intensity when cutting.
Cut frequency	The carrier frequency of PWM modulation signal is also the number of light output in one second when cutting.
Duty cycles	The duty ratio of cutting is setting by adjusting the laser with PWM.
Up height	The height of the laser head when switch cut graphics.
Pre-pierce up height	In pre-pierce, the height of the cut head is raised after each hole is pierced. If the total number of tool path pierce is 1, this parameter does not take effect.
Cut gas	The type of gas used to cut.
Cut pressure	The gas pressure of cutting should be used with proportional valve.
Cut height	The height of the nozzle from the tube when cutting.
Cut focus	This takes effect after focus control is enabled. Position of focus point when cutting.

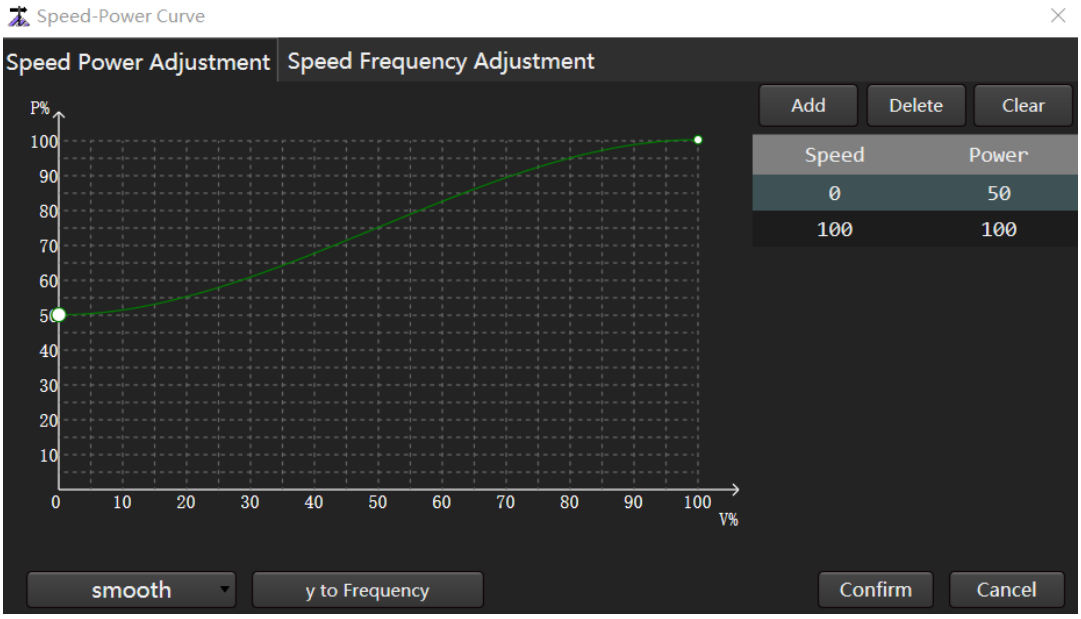
Parameter description of special technic parameter area:

Parameter	Description
Ignore this layer	Do not machining all graphics in the current layer.
Gas keeps on for this layer	Do not close the blowing port during the machining of the graphics in this layer.
Side blow at pierce	When piercing, open the side blowing port.
Side blow at cut	When cutting, open the side blowing port.
Fixed cut	Whether the cutting has been kept in the fixed Z axis coordinates for cutting.
Pre-piercing	All machined objects in the current layer are enabled pre-piercing so that all tool paths are perforated before the actual machined objects.
Micro joint at high speed	The laser is not opened at the micro connection, and the cutting head continues to move without slowing down.
Cut film	Enabled when cutting the metallic material of the top film.
Coolant	Gas used for cooling.
Cool delay	When the machining to the cooling point is blowing cooling time.

3. In the power curve area, select **Enable Speed Power Adjustment** and **Enable Speed Frequency Adjustment** as required. The following table describes the parameters:

Parameter	Description
Enable speed power adjustment	When cutting, the cutting power will change with the cutting speed, and the specific value is determined by the speed power curve.
Enable speed frequency adjustment	When cutting, the cutting power will change with the cutting speed, and the specific value is determined by the speed frequency curve.

4. Using **Enable Speed Power Adjustment** as an example, click **Edit** to open the **Speed-power Curve** dialog box:



5. Select the following methods to edit the power curve:
   - Edit in curve box:
     - Double-click the target position to add a curve node.
       The more nodes you add, the more accurate the curve will be.
     - Double-click the location of the added node to delete it.
       As nodes are added or deleted, the corresponding speed and power values will added or deleted in the list on the right.
   - Edit in the list on the right:
     - Click **Add** and the list will automatically add a set of speed power values. Then double-click to change the value and click on the blank area.

       Note

       The speed power curve is in increasing mode. The values added must increase in ascending order, and **0** and **100** cannot be modified.

     - Select a set of speed power values and click **Delete** to delete the set of values.
       The left curve box adds or deletes the corresponding nodes synchronously.
       To restore the curve to the default curve, click **Clear**.

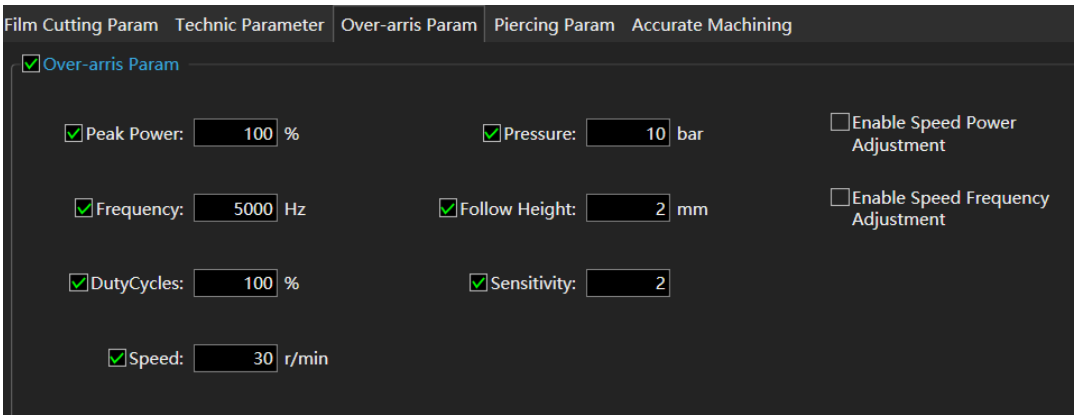
   During machining, the system will automatically adjust the speed and power/frequency matching relationship according to this curve, without other manual operation.

4.2.2.2.3. Set Over-arris Parameter

Used to set the **Peak Power**, **Pressure** and other over-arris parameters.

Operating Steps:

1. In the **Layer Setting** dialog box, click **Over-arris Param** to switch to the **Over-arris Param** page:



2. The parameters are described as follows：

Parameter	Description
Peak power	By adjusting the setting of the laser with the analog quantity, the peak current and peak power during over-arris cutting can be corresponding.
Pressure	When over-arris cutting the film. The gas pressure of cutting should be used with proportional valve or multi-valve.
Frequency	When over-arris cutting the film, the carrier frequency of the PWM modulation signal is also the number of light output in one second.
Follow height	The height of the nozzle from the tube when over-arris cutting.
Duty cycles	The duty ratio of over-assis cutting.
Sensitivity	Follow the torch lifter dynamic sensitivity when over-arris cutting.
Speed	The speed of the axis of rotation used when over-arris cutting（In case of over arris, the rotation axis speed is controlled separately. In other cases, the rotation axis speed is planned together with other axes according to the cutting speed）.

3. In the power curve area, select **Enable Speed Power Adjustment** and **Enable Speed Frequency Adjustment** as required. The following table describes the parameters：

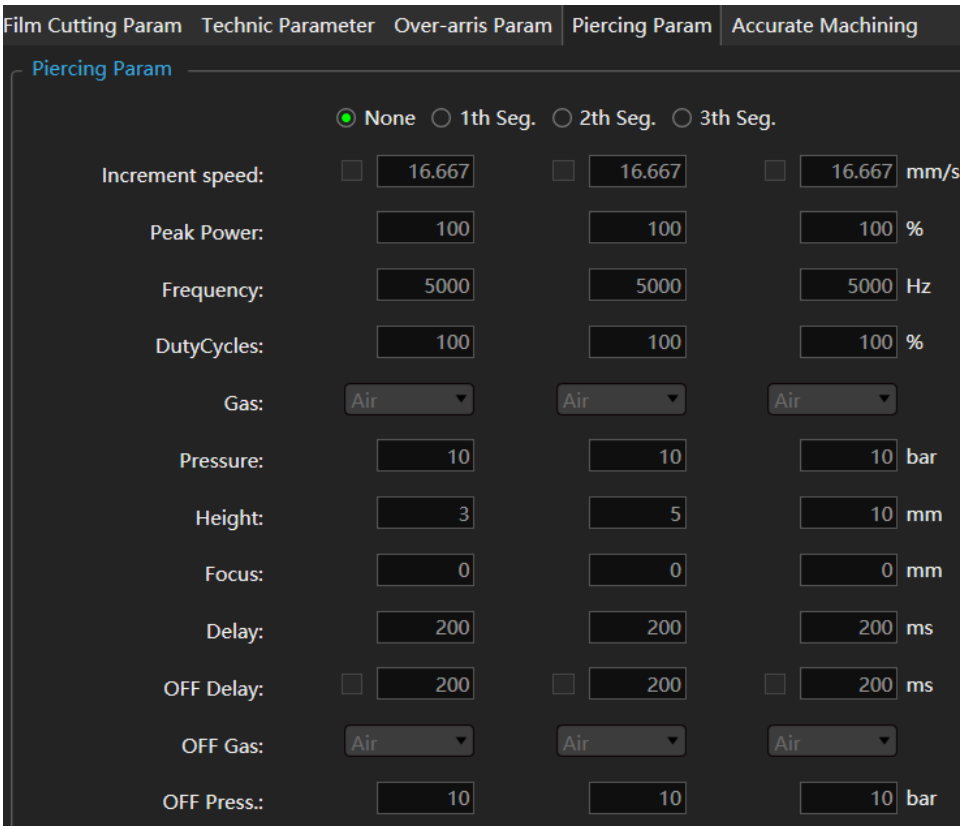
Parameter	Description
Enable speed power adjustment	When cutting, the cutting power will change with the cutting speed, and the specific value is determined by the speed power curve.
Enable speed frequency adjustment	When cutting, the cutting power will change with the cutting speed, and the specific value is determined by the speed frequency curve.

4.2.2.2.4. Set Piercing Parameter

Used to select the piercing mode and set the piercing parameters.

Operating Steps：

1. In the **Layer Setting** dialog box, click **Piercing Param** to switch to the **Piercing Param** page：



2. Select the method of piercing as follows:
   - **None**
     The system automatically performs the following processing actions:
     a. Open the follow-up valve and blow valve.
     b. After the cutting head moves down to the **Cutting Height**, wait for the **Blowing Delay** time set in **Common Parameters**.
     c. Open the laser valve and start cutting.
   - **1st Segment / 2nd Segment**
     The system automatically performs the following processing actions:
     a. Open the follow-up valve and blow valve.
     b. After the cutting head moves down to the **Cutting Height**, wait for the **Blowing Delay** time.
     c. Open the laser valve and start pierce. The duration is the **Pierce Delay** time.
     d. Select the method of piercing as follows:
        - **Increment Pierce:** Without closing the laser valve, start the cutting process at the **Increment Speed** to the **Cutting Height**.
        - **Section pierce:** Close the laser valve, control the cutting head to move down to the **Cutting Height**, then open the laser valve to start cutting.
   - **3rd Segment**
     The system automatically performs the following processing actions:
     a. Perform 3rd segment.
     b. Perform 2nd segment.
     c. Perform 1st segment.
3. The parameters are described as follows.

Parameter	Description
Increment speed	Sets the speed at which the piercing height descends to the cutting height when using increment pierce.
Peak power	The laser intensity during piercing is set by adjusting the laser by analog quantity.
Frequency	In piercing, generally adopts a low PWM modulation signal carrier frequency to avoid mistakes.
Duty cycles	The duty ratio of piercing.
Gas	The type of gas used to pierce.
Pressure	The gas pressure of cutting should be used with proportional valve.
Height	The height of the pierce from the tube.
Focus	This takes effect after focus control is enabled. Position of focal point when piercing.
Delay	The time that increment pierce and section pierce turn on the laser at the piercing height.
Off delay	The interval time between blowing after the laser is turned off.
Off gas	Air, nitrogen, oxygen.
Off pressure	The air pressure value when blowing.

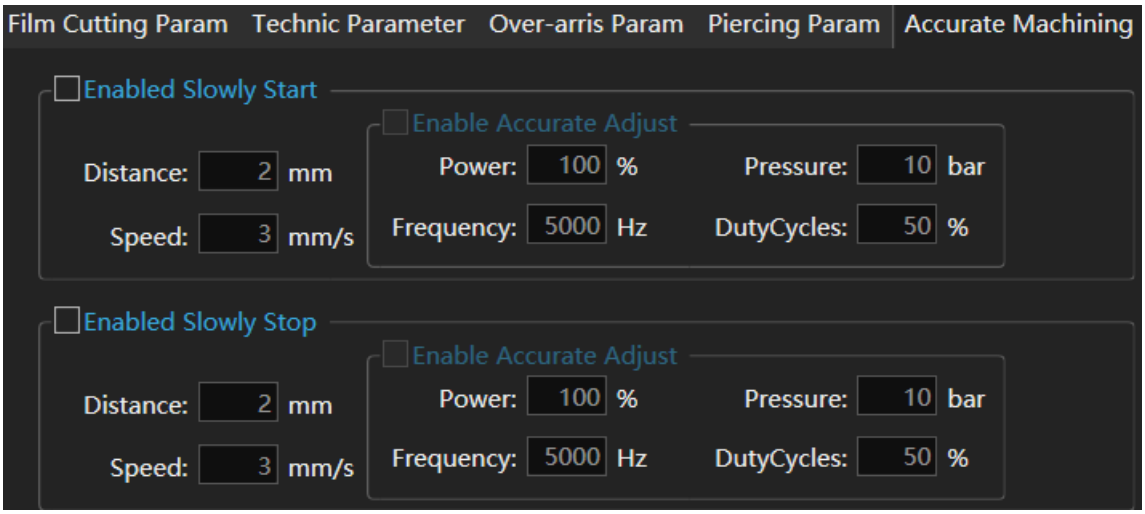
4. **（Optional：）** If click the **Increment Speed**, the cutting head will move from the piercing height to the cutting height at that speed when Increment piercing is used. If not selected, the cutting head follows to the **Cutting Height**.
5. **（Optional：）** If click the **Off Delay** , **Off Delay**, **Off Gas** and **Off Pressure** can be set. If not clicked, the corresponding pierce will not stop blowing after the end.

4.2.2.2.5. Set Accurate Machining

Used to set slowly start and slowly stop, and select whether to enable slowly start and slowly stop.

Operating Steps：

1. In the **Layer Setting** dialog box, click **Accurate Machining** to switch to the **Accurate Machining** page：



2. Check **Enable Slowly Start** and **Enable Slowly Stop** as required, and set the parameters. The parameter description follows.

Parameter	Description
Enable slowly start	The speed, frequency, duty cycle, power and pressure used when cutting a short distance from the starting point. If you do not enable accurate adjustment, the effect is equivalent to the original slow start.
Enable slowly stop	Speed, frequency, duty cycle, power and pressure used for a short distance before the end of cutting. If you do not enable accurate adjustment, the effect is equivalent to the original speed.

4.2.3. Apply Layer Technic

When drawing parts, the default is to use the large graphics layer technique. This section describes how to apply the layer technic.

Operating Steps：

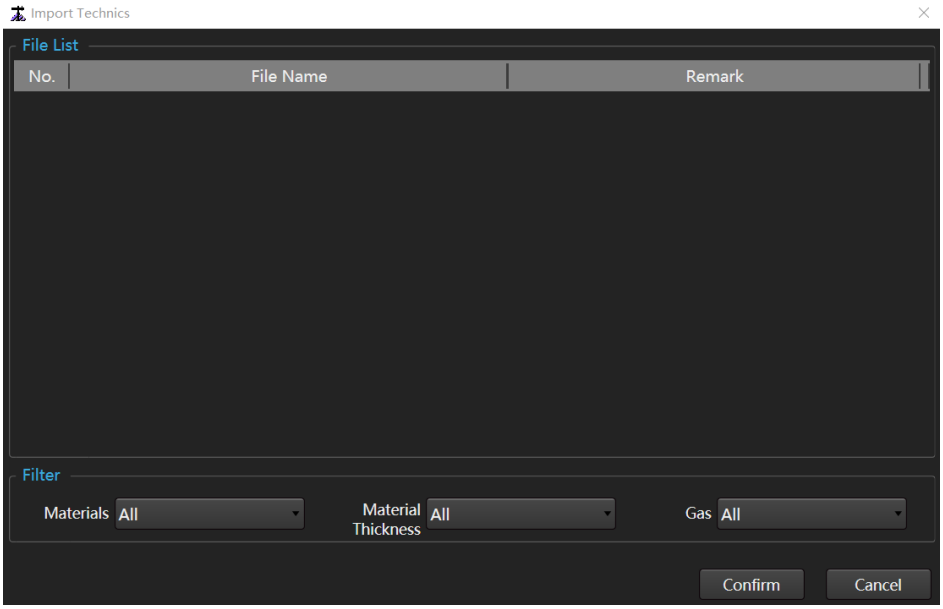
1. Select the graphic or part.
2. Click the layer's color in the **Layer** to set the selected object as the layer's technic.

4.2.4. Import Technics

Import the target technic file from the technic library, that is, quickly apply the layer technic parameter information to the system.

Operating Steps：

1. In the **Layer Setting** dialog box, click **Import Technics** to open the **Import Technics** dialog box:



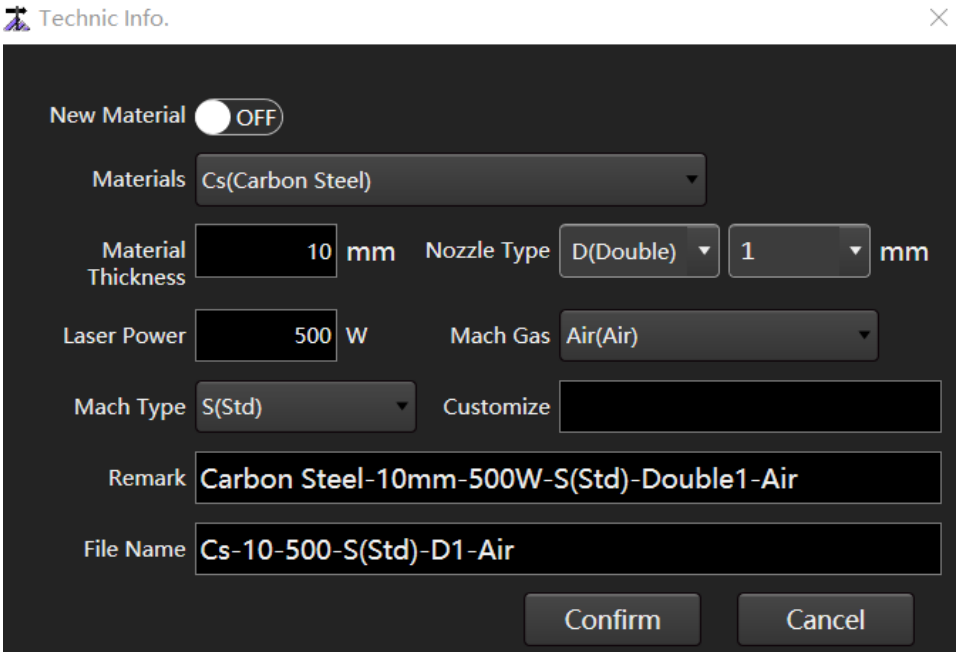
2. Click the technic to be imported and it becomes highlighted, click **Confirm**, and a confirmation prompt box pops up. Click **Confirm**.

4.2.5. Save As

Save the current technic to the technic library.

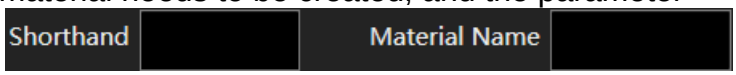
Operating Steps：

1. In the **Layer Setting** dialog box, click **Save As** to open the **Technics Info.** dialog box：



2. Set the technic information parameters. The parameters are described as follows：

Parameter	Description
New material	New material button. ▪ When the button is set to OFF, the new material function is not enabled. ▪ When the button is set to ON, the new material function is enabled.

Parameter	Description
Materials	The material of the tube. Perform different operations according to the status of the new material button. ▪ When the button is set to OFF, select the existing material from the **Materials** drop-down box. ▪ When the button is set to ON, a new material needs to be created, and the parameter **Materials** becomes  to fill in the shorthand and material name.
Material thickness	You can manually set the thickness of the tube in millimeters (mm).
Nozzle type	Select the nozzle type. If there is no suitable type, you can add it in **Nozzle Information Management** and then select it. For details, see Nozzle Info. Management.
Laser power	The power of laser cutting can be manually set in watts (W).
Mach gas	The cutting gas air, nitrogen $N_2$and oxygen O_2can be selected.
Mach type	S, H, Q can be selected.
Customize	Additional information can be added manually.
Remark	Notes are automatically generated according to the parameters of technic information. Naming rules: Material name - material thickness - laser power - machining type - nozzle type (type+diameter) - gas type. It is named Chinese with unit.
File name	The file name is automatically generated according to the parameters of the technic information. Naming rules: Material abbreviation - material thickness - laser power - machining type - nozzle type (type+diameter) - gas type - custom. Named in English or abbreviated without units.

3. Click **Confirm** to finish saving to the technic library.

4.3. Machining Technic

4.3.1. Overview

The machining technic can be set on the **Software Main Interface** and **2D Edit** pages. The two included machining technic functions are as follows:

This chapter will introduce all the machining technic functions. If the function entries supported by the two pages are the same, it will not be emphasized in which page to set.

Technic	Software Main Interface	2D Edit
Lead line	√	√
Kerf compensation	√	√
Corner chop	√	√
Grid chop	√	√
Micro joint	√	√

Technic	Software Main Interface	2D Edit
Chamfer	√	√
Bridge	√	-
Cooling point	√	√
Unfill/fill	√	√
Direction	√	√
Centering mark	√	√
Inner compensation	√	√
Weld compensation	√	√
Vertical intersection	√	√
Over arris jiggle	√	√
Instant setting	√	√
Clear	√	√

4.3.2. Lead Line

It is used to avoid machining errors or damage to the workpiece caused by laser staying above the start position for a long time, so as to improve machining accuracy.

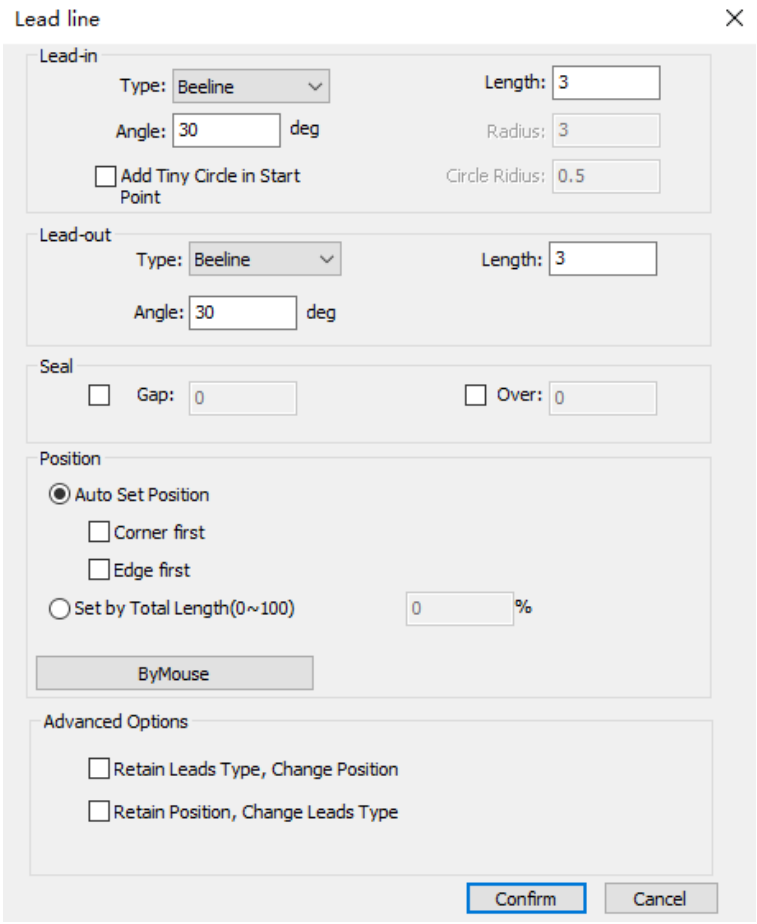
The types of lead line are divided into:

- **Lead-in**: consisting of line lead line, arc lead line and hook lead line.
- **Lead-out**: consisting of line lead line and arc lead line.

Operating Steps:

1. Select one or more graphics.
2. Select any of the following methods to open the **Lead Line** dialog box:
   - In the **Tech** area of the **Common** menu bar, click 

→ **Lead Line**.
   - Right click, and click **Lead Line** → **Set** in the shortcut menu.



3. In the **Lead-in** and **Lead-out** areas, set the lead-in and lead out types and related parameters. The parameters are described as follows:
   - **Angle**: For a line-type lead line, it refers to the angle between the lead line and the tangent line of the intersection; for an arc-type lead line, it refers to the central angle.
   - **Length**: For a line-type/arc-type lead line, it refers to the length of the line/arc; for a hook-type lead line, it refers to the sum of the radius of the arc and the length of the line.
   - **Radius**: For a hook-type lead line, it refers to the radius of the arc.
   - **Add Tiny Circle in Start Point**: Add a tiny circle at the start point of the lead-in line, so as to solve the problem that the accumulation of slag influences cutting effect during piercing a thick tube.
   - **Circle Radius**: The radius of the hole at the start point of the lead line.
4. **（Optional：）** If the target object is a closed object, in **Seal** area, select one of the following
   - **Gap**: The lead line is unclosed and the cutting head will not cut through at the sealing position.
   - **Over**: The lead line is closed and the cutting head will cut at the sealing position.

   In the menu bar, click Lead line → **Seal** → **Notch** / **Overcut** to edit or delete gaps and overcut separately.
5. Set the position of the lead line:
   - If you check **Auto Set Position**, do one of the following:
     - **Corner First**: Add lead line at the corner first.

- **Edge First**: Add lead line at the longest edge first.

- If **Set by Total Length (0 ~100)** is selected, set the percentage of the position from the start point of machining to the lead line in the total side length of the graphic.

  Note

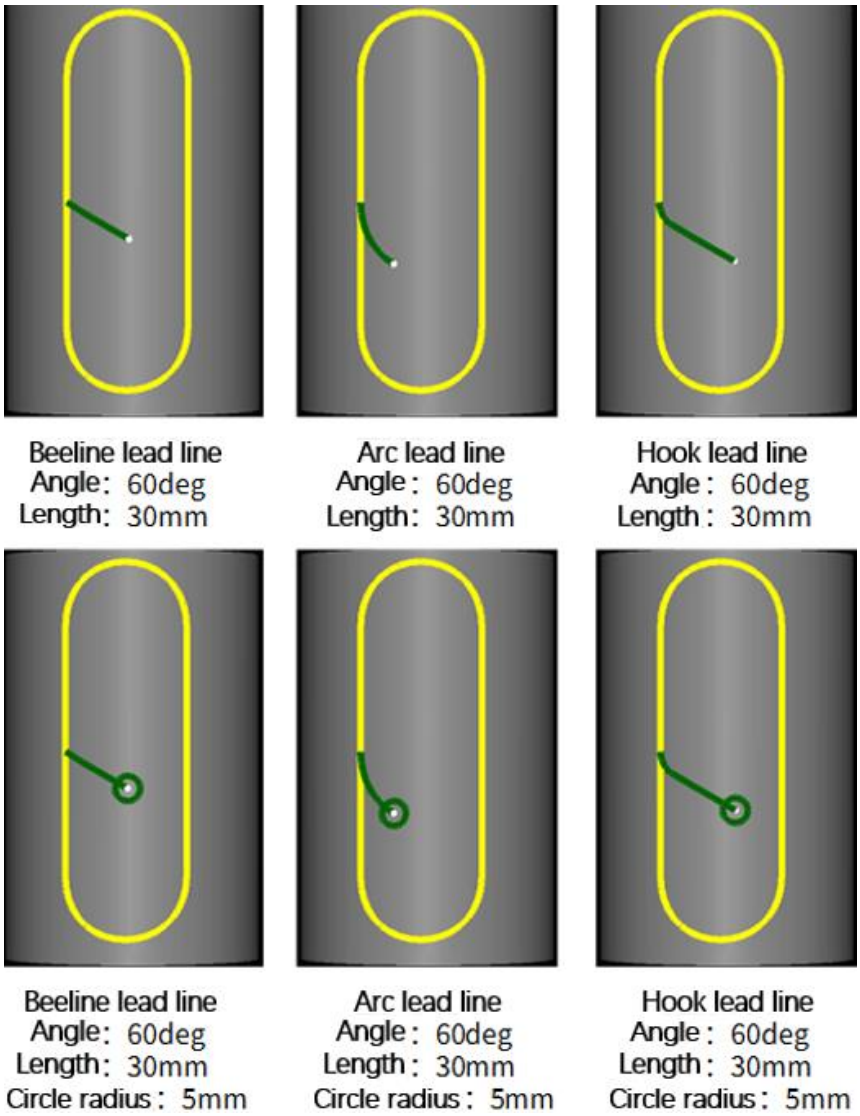
  Available only for closed graphics.

- If you click **By Mouse**, the cursor will become to  . Click the graphic edge to manually specify the position of the lead line. After setting, right click or press **Esc** to exit the tool.

6. In the **Advanced Area**, select as needed: **Retain Leads Type, Change Position** / **Retain Position, Change Leads Type**.
7. **（Optional：）** To manually modify the lead line position, perform the following steps:
   a. Select the following methods to use the manual set start point function.
      - In the menu bar, click 

→ **Set Start Point**.
      - Right click and click **Lead Line** → **Start Point**.
   b. Click the left mouse button on the graphic edge to modify the position of the lead line without modifying the angle and length.
   c. Right click or press **ESC** to exit the start point function.

Effect picture:



4.3.3. Kerf Compensation

Laser cutting has kerf (the part of loss during cutting), which makes the size of the part actually cut deviate from the theoretical size of the part. This operation can compensate the geometric dimension of the deviation.

The type of kerf compensation includes the following:

- **All Shrink**: Shrink the cutting area for all selected parts.
- **All Expand**: Expand the cutting area for all selected parts.
- **Unfill: Shrink, Fill: Expand**: Shrink the cutting area for parts with unfill attribute, and expand the cutting area for parts with fill attribute.

Operating Premise:

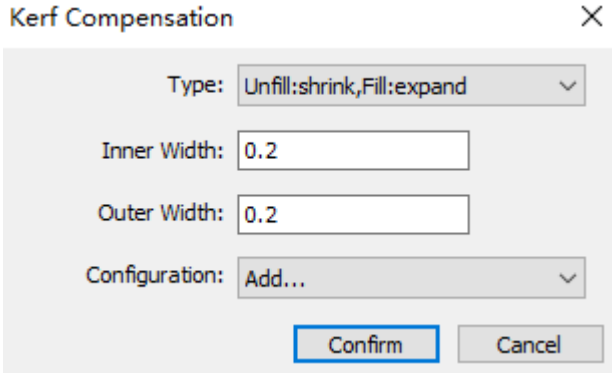
Before compensating the kerf, ensure the following:

- The text has been turned into polylines.
- It is not dot, auxiliary line, over tangent, over arris, scan, self intersection and co-edge graphic.

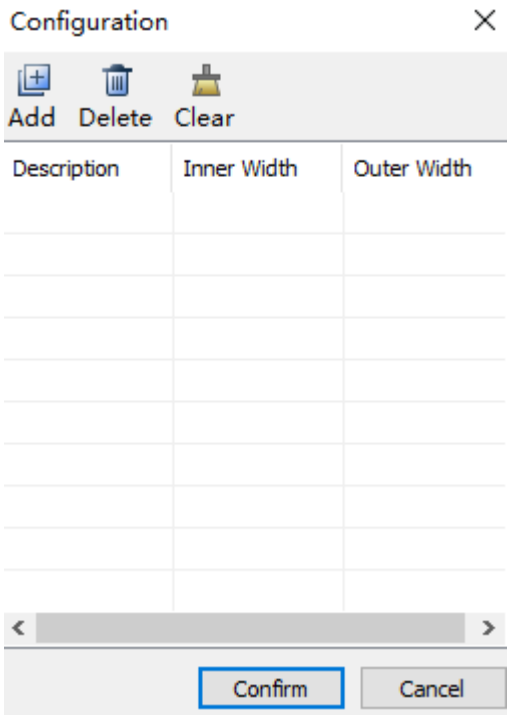
Operating Steps:

1. Select one or more graphics.

2. In the **Tech** area of the **Common** menu bar, click Kerf Compensation to open the **Kerf Compensation** dialog box:



3. Click the **Type** drop-down box to select a compensation type.
4. Set the inner width and outer width.
5. **（Optional：）** To save the commonly used inner width and outer width for later use, do the following:
   a. In the drop-down box of **Configuration**, select **Add**. **Configuration** dialog box pops up:

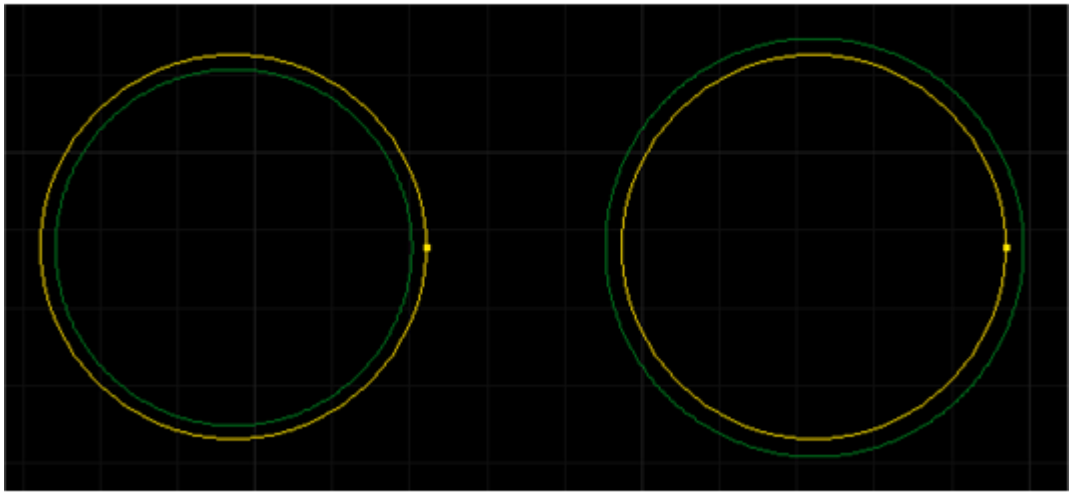


   b. Click **Add**, set a name in **Description** column, set the inner width in **Inner Width** column, and set outer width in **Outer Width** column.
   c. For later use, in the drop-down box of **Configuration**, select the name set in the **Description** column. The system automatically fills in the inner width and outer width.

Effect picture:

Original machining trace

Actual machining trace after compensating



Shrink Expand

4.3.4. Corner Chop

It is mainly used for graphics spanning 2~3 faces. In actual cutting, this kind of tool path is difficult to fall off after cutting, and it is automatically marked before cutting to facilitate the waste falling off.

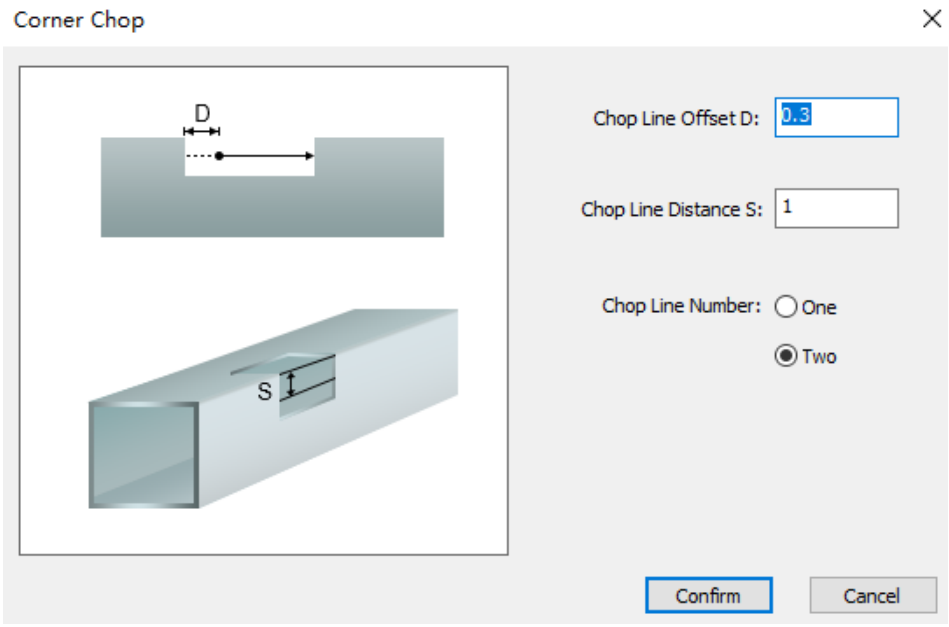
Operating Premise：

Before corner chop an object, ensure the object meets the following requires：

- Unfill.
- Non text.
- Over arris closed graphic spanning 2~3 sides.
- Micro joint and chop are not added.
- No other graphics are included.

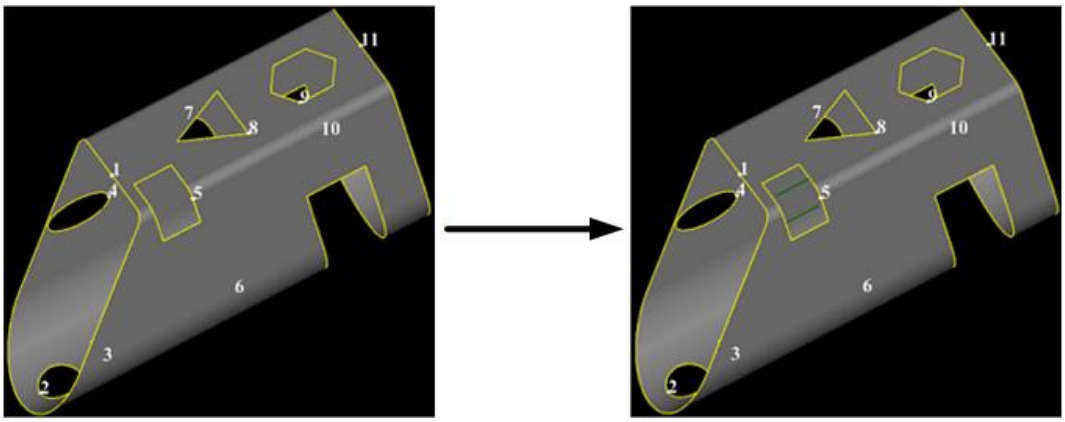
Operating Steps：

1. Select one or more graphics.
2. The path to open the **Corner Chop** dialog box varies according to the interface：
   - In the **Software Main Interface**, in the **Tech** area of the **Common** menu bar, click Chop → **Corner Chop**.
   - In the **2D Edit** page, in the **Tech** area of the **Common** menu bar, click Corner Chop.



3. Set parameters. The parameters are described as follows:
   - **Chop Line Offset**: The distance between the start point of the chopping line and the hole.
   - **Chop Line Distance**: The distance between the chopping line and the arris.
   - **Chop Line Number**: Select 1 or 2.
4. Click **Confirm**.

Effect picture:



4.3.5. Grid Chop

Divide the selected graphic into multiple blocks, that is, chop the machining waste to facilitate the waste falling off.

Operating premise:

Before chop an object, ensure the object meets the following requires:

- Non text.
- Non over arris graphic.
- Unfill.
- Close graphic.
- Large graphics, and the minimum distance between the chop line and the entity frame is 0.3mm, and the minimum length of the chop line is 1mm.

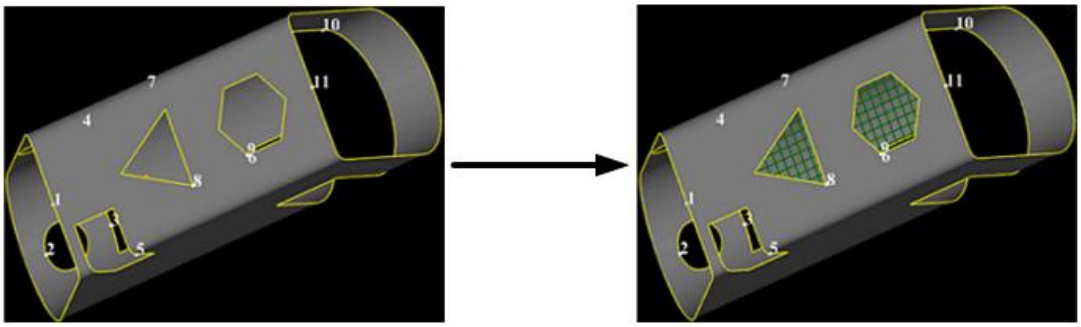
- No micro joint was added.
- No other graphics are included.

Operating Steps：

1. Select one or more drawings.
2. The path to open the **Chop** dialog box varies according to the interface：
   - In the **Software Main Interface**, in the **Tech** area of the **Common** menu bar, click Chop → **Grid Chop**.
   - In the **2D Edit** page, in the **Tech** area of the **Common** menu bar, click Chop.

3. Set the chop space.

Effect picture：



4.3.6. Micro Joint

Micro joints can be used to connect parts with surrounding materials, so that materials do not fall and sorting is not required. At present, the section steel does not support micro joint of the cut-off line.

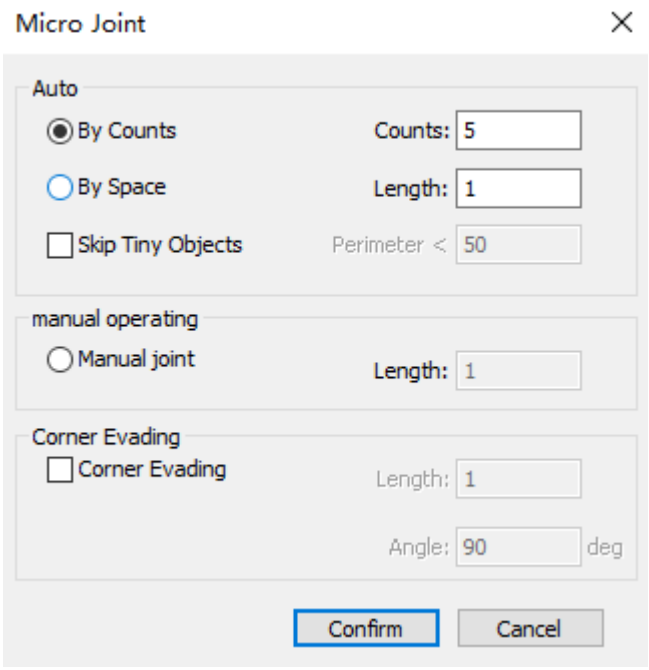
The following two micro joint settings are supported：

- Set automatic micro joint： The system automatically adds micro joint to the selected objects according to the set values.
- Set manual micro joint： Select the micro joint position by yourself.

4.3.6.1.1. Automatic Joint

Operating Steps：

1. Select the object.
2. In the **Tech** area of the **Common** menu bar, click Micro Joint to open the **Micro Joint** dialog box：



3. In the **Auto** area, select by counts or space to micro joint.

Note

The system automatically executes micro joints on the selected objects according to the set values.

---

4. **（Optional：）** Check **Skip Tiny Objects** and set the perimeter.

Note

Small graphics within the perimeter will not be micro joints.

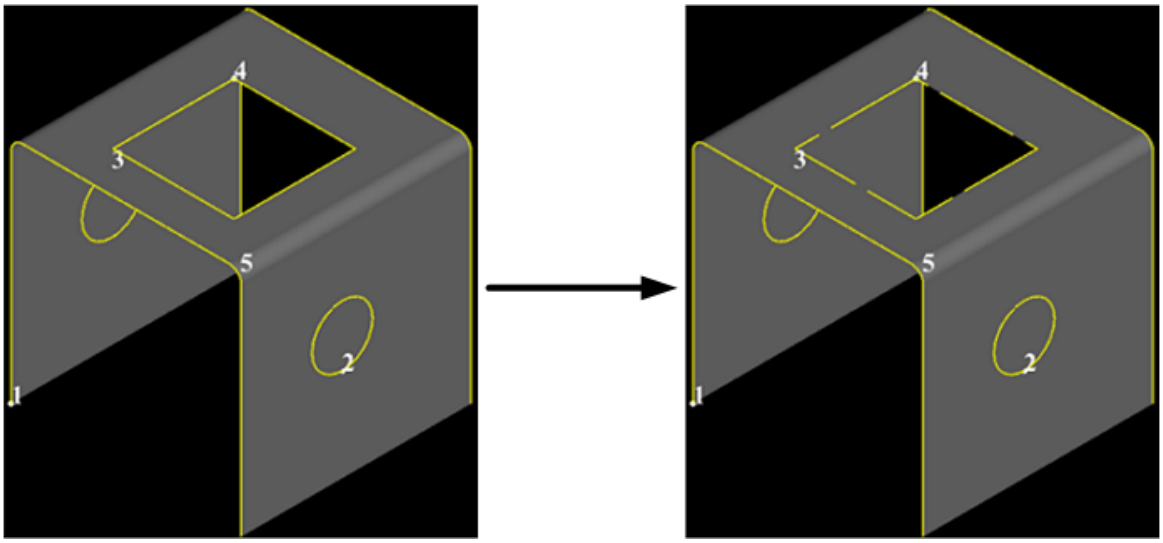
---

5. Check **Corner Evading** and set the following parameters:
   - **Length**: In the range of evade length, the corner cannot be added micro joint. Range: 0.001 mm ~10 mm.
   - **Angle**: Range: 90 ° ~180 °.

Note

If **Corner Evading** is not checked, all points support micro joint.

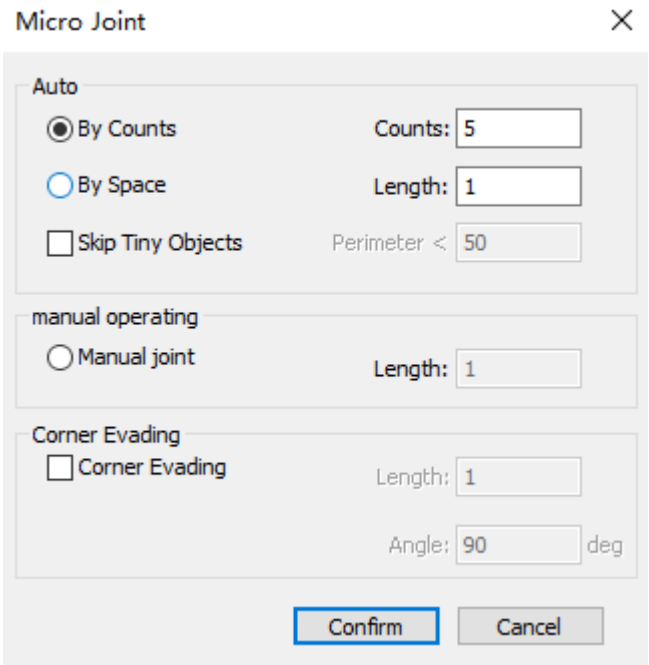
When **By Counts** micro joint is selected and the set quantity is **5**, the effect before and after the automatic micro joint is as follows:



4.3.6.1.2. Manual Joint

Operating Steps：

1. You do not need to select an object. In the **Tech** area of the **Common** menu bar, click Micro Joint to open the **Micro Joint** dialog box：



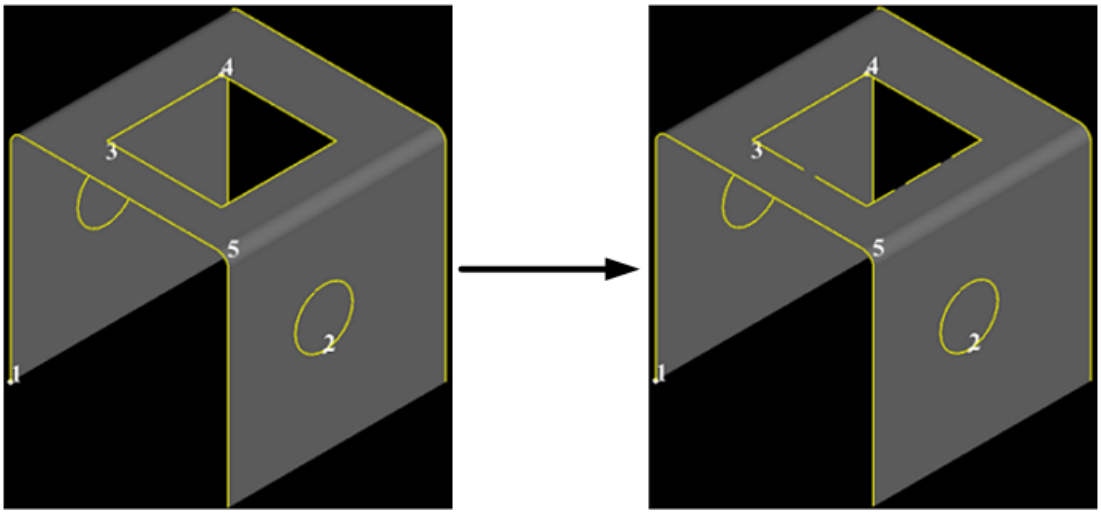
2. In **Manual Operating** area, select **Manual Joint**, and set parameter **Length**.
3. Check **Corner Evading** and set the following parameters：
   - **Length**： In the range of evade length, the corner cannot be added micro joint. Range: 0.001 mm ~10 mm.
   - **Angle**： Range: 90 ° ~180 °.

Note

If **Corner Evading** is not checked, all points support micro joint.

4. Click **Confirm**, and the cursor becomes to  .
5. Move the mouse to select the micro joint position, and click the left mouse button to add micro joint.
6. Right click or press **Esc** to exit the manual micro joint function.

Effect picture:



4.3.7. Chamfer

Perform arc chamfer on all corners within the set angle range in the graphic to improve the cutting effect of inflection points when cutting thick materials.

Select the following methods to add chamfers:

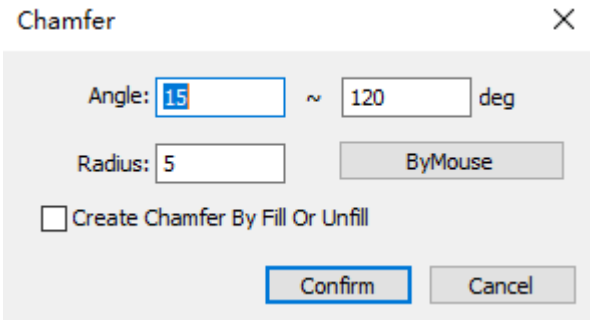
- Automatically add chamfer: Automatically chamfer the selected and qualified objects according to the set value.
- Manually add chamfer: Manually select chamfer position based on your need. Angle range: (0, 180)°

4.3.7.1.1. Automatic Chamfer

Operating Steps:

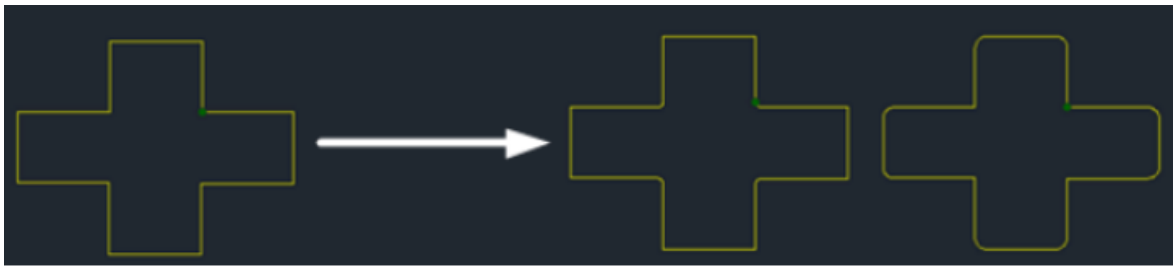
1. Select the object.
2. In the **Tech** area of the **Common** menu bar, click 

to open the **Chamfer** dialog box:



3. Set parameter angle and radius.
4. **（Optional：）** To automatically add chamfer for closed objects according to the attribute of fill/unfill, check **Create Chamfer by Fill or Unfill**.
5. Click **Confirm**. After setting, the system will automatically add chamfers to the qualified corners.

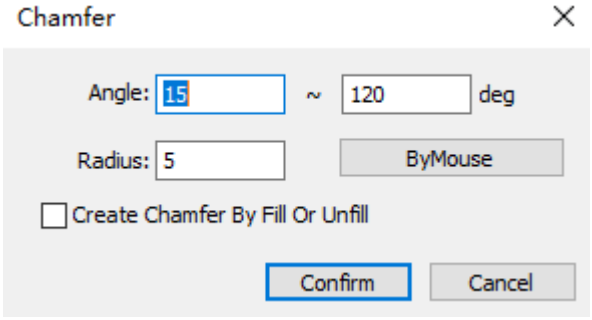
Taking the following as an example, set parameter **Angle** between 45° and 90° and check **Create Chamfer by Fill or Unfill**. The effect picture is as follows:



4.3.7.1.2. Manual Chamfer

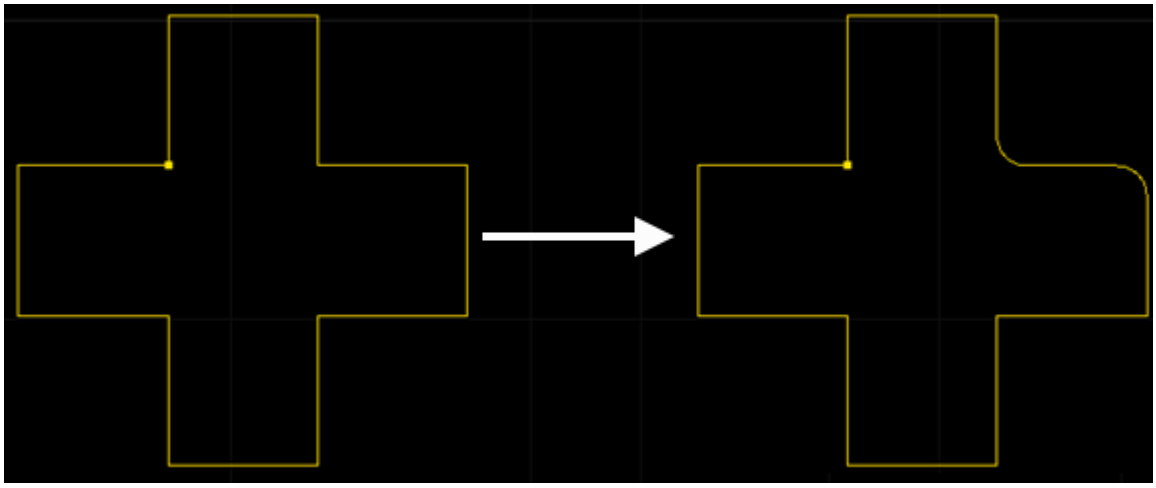
Operating Steps：

1. You do not need to select an object. In the **Tech** area of the **Common** menu bar, click Chamfer to open the **Chamfer** dialog box:



2. Sets the radius of the chamfer.
3. **（Optional：）** Check **Create Chamfer by Fill or Unfill**.
4. Click **By Mouse**, and the cursor becomes to .
5. Move the mouse to the target position and left click to select an adding position.
6. Right click to exit manually add chamfer function.

When **Create Chamfer by Fill or Unfill** is not checked, effect picture:



4.3.8. Bridge

When a part is composed of multiple parts, use this function to connect these parts, so that they will not fall off after cutting, and reduce the number of pierces. Using the **Bridge** function for many times can achieve the effect of completing all graphics at one time. It is mostly used to connect text strokes.

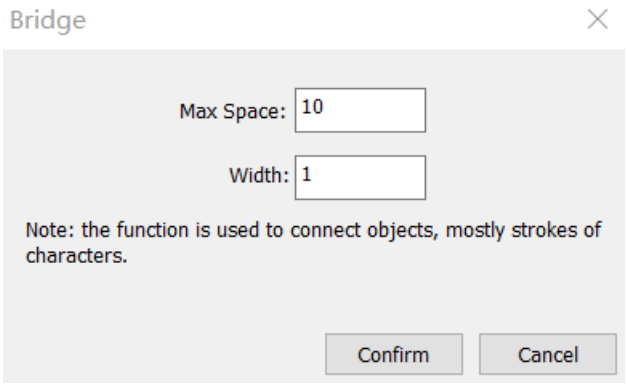
Operating Steps：

1. In the **Software Main Interface**, select the graphic.

   Note

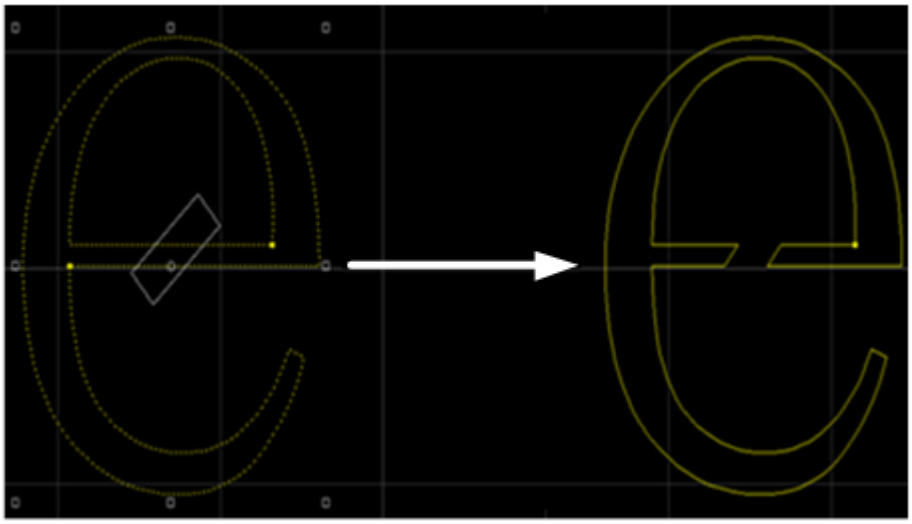
   If the bridged object is text, make sure that the text has been converted to graphics.

2. In the **Tech** area of the **Common** menu bar, click Bridge to open the **Bridge** dialog box：



3. Set parameter **Max Space** and **Width**.
4. Click **Confirm**.
5. Click the left mouse button to select the two ends of the bridge.
6. Right click or press **Esc** to exit bridge function.

Effect picture：



4.3.9. Cooling Point

Add a cooling point at the inflection point of the graphic, and only blow air without turn on the laser. Avoid slowing down at the corners of the graphic during machining, resulting in excessive local laser energy. If continuous machining occurs, such phenomena as corner burning and excessive slag will occur.

The following two methods to add cooling points are supported：

- Automatically add cooling point: Automatically add cooling points to selected and qualified objects according to the set value.
- Manually add cooling point: The position of inflection point shall be selected by yourself. Angle range: 0 °~180 °.

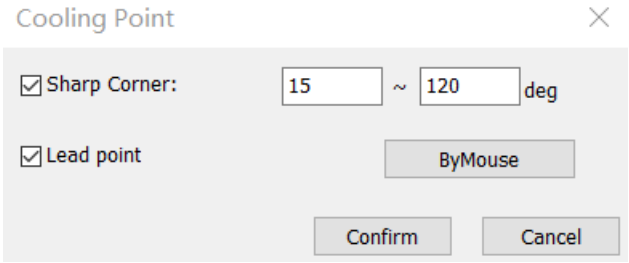
Note

Cooling point cannot be added at the start point of machining.

4.3.9.1.1. Automatic Cooling Point

Operating Steps:

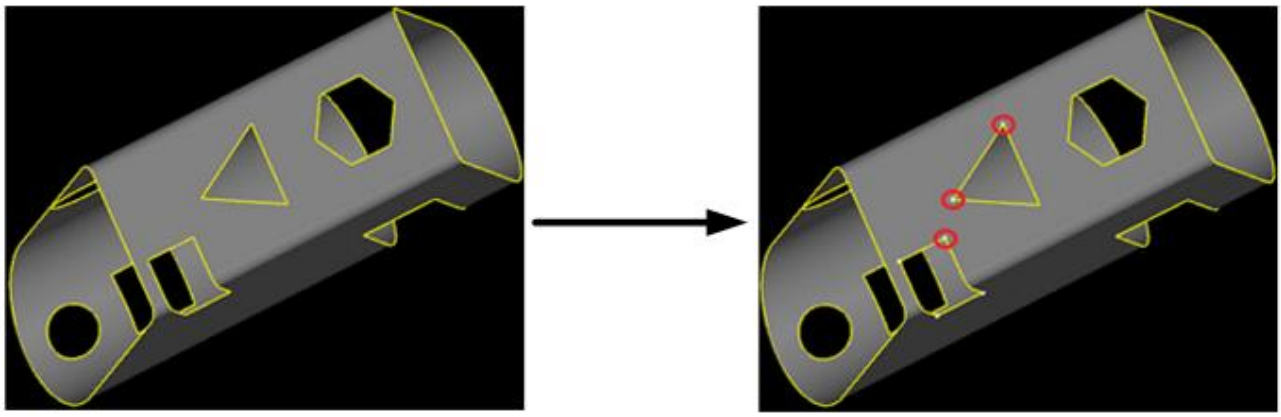
1. Select the object.
2. In the **Tech** area of the **Common** menu bar, click Cooling Point to open the **Cooling Point** dialog box:



3. Check **Sharp Corner** and set the angle range for sharp corner.
4. **（Optional：）** To add a cooling point at the position of lead-in line without being limited by the range of the angle of sharp corner, check **Lead Point**.

After setting, the system will automatic add cooling points at the inflection points that meet the conditions.

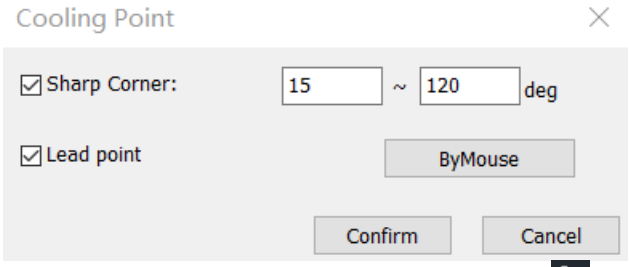
When **Sharp Corner** is set to 15 ° ~90 °, effect picture:



4.3.9.1.2. Manual Cooling Point

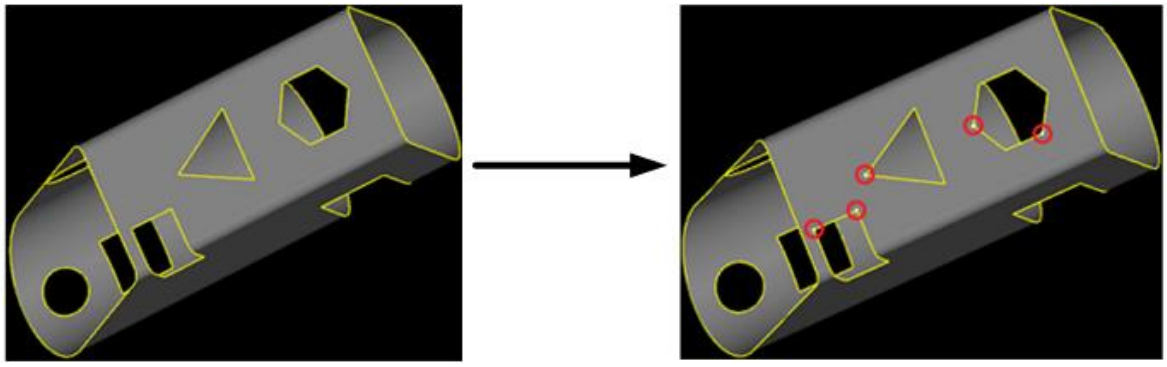
To manually add cooling points without selecting objects, follow the steps:

1. In the **Tech** area of the **Common** menu bar, click Cooling Point to open the **Cooling Point** dialog box:



2. Click **By Mouse**. The cursor becomes .
3. Move the mouse to select a cooling position, and click the left mouse button to add a cooling point.
4. Right click or press **Esc** to exit manually add cooling point function.

Effect picture:



4.3.10. Unfill/Fill

Unfill is used to keep the outside of the closed graphic during machining. **Fill ** is used to keep the inside of the closed graphic during machining.

Operating Steps:

1. Select the closed graphic.
2. Select one of the following methods to set the unfill and fill cutting:
   - In the **Tech** area of the **Common** menu bar, click Unfill /Fill → **Unfill** / **Fill**.
   - Right click and click **Unfill/Fill** → **Unfill** / **Fill** / **Auto Setting**.

Note

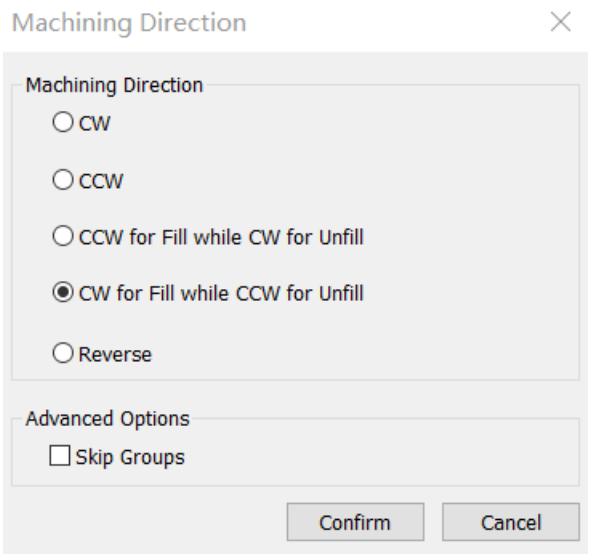
Automatic setting refers to the setting of unfill or fill according to the nesting relationship of the selected graphic.

4.3.11. Direction

Used to change the original machining path direction in the tool path.

Operating Steps:

1. Select the object.
2. Select the following method to open the **Machining Direction** dialog box:
   - In the **Tech** area of the **Common** menu bar, click Direction → **Auto Setting**.
   - Right click and click **Machining Direction** → **Set**.



3. In the **Machining Direction** area, select the machining direction.
   If you only need to reverse the machining direction, select the following method:
   - In the **Tech** area of the **Common** menu bar, click Direction → **Reverse**.
   - Right click and click **Machining Direction** → **Reverse**.
   - In the **Machining Direction** dialog box, click **Reverse**.
4. **（Optional：）** If need is setting the machining direction, the primitive machining direction in the group remains unchanged. Check **Skip Groups**.

After setting, the machining direction is automatically generated.

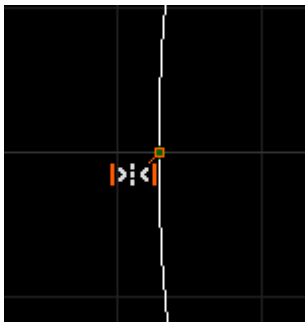
4.3.12. Centering Mark

When the tube is long, there will be some bending deformation in the middle of the front and rear chucks, resulting in that after cutting a section of length, the centering data executed before machining cannot continue to apply. This operation can eliminate this error. After cutting a certain length, the tube will be automatically leveled and divided, and then the breakpoint resume will be automatically performed.

Operating Steps：

1. In the **Tech** area of the **Common** menu bar, click Centering Mark → **Manually** / **Manually （Single Face）** / **Automatically**：
   - Manually： Specify the centering position of the tube section at this position.
   - Manually （single face）： Specifies the centering position of the tube single face.
   - Automatically： After setting the centering mark interval, the system automatically adds the leveling center mark according to the set interval.

After selecting the marking method, the system automatically adds the centering mark and centering at the specified place. Effect picture:



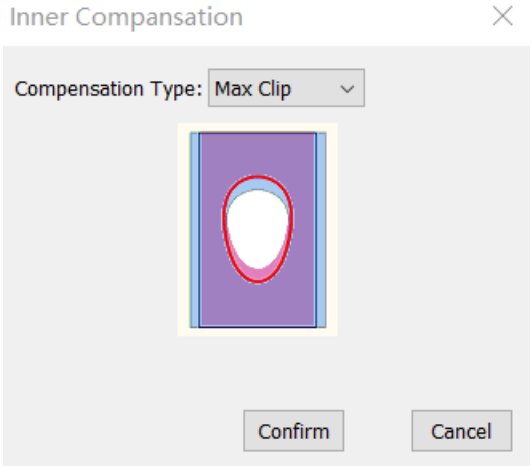
4.3.13. Inner Compensation

When performing the intersection with a certain angle of inclination, it is impossible to ensure the insertion of the inner diameter of the main tube because the laser head is cut vertically. The **Inner Compensation** function ensures that the inner diameter of the inserted main tube and the weld are as small as possible.

Note that 2D editing and drawing graphics cannot add inner compensation.

Operating Steps：

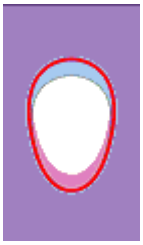
1. Select the object.
2. Select any of the following methods to open the **Inner Compensation** dialog box：
   - In the **Tech** area of the **Common** menu bar, click Inner Compansation.
   - Right click and click **Inner Compensation**.



3. Click the **Compensation Type** drop-down box to select a compensation type.

Effect picture，the red line is the cutting path：

- **Max Clip**：



- **Inner Clip**：

- **Min Clip**:

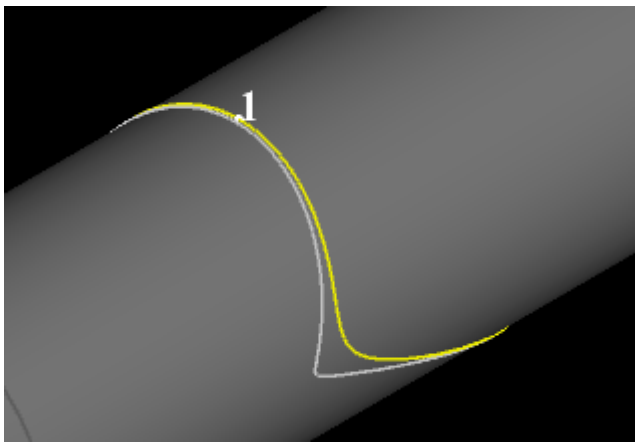
4.3.14. Weld Compensation

Used to compensate for weld shrinkage caused by welding.

Operating Steps：

1. Select any of the following methods and use the **Weld Compensation** function:
   - In the **Tech** area of the **Common** menu bar, click Weld Compensation drop down box.
   - Right click and click **Weld Compensation**.
2. Select the setting method in the submenu:
   - To set the selected break line, click **Compensation Select**.
   - To set all break lines, click **Compensation All**.

Effect picture:



4.3.15. Vertical Intersection

During cutting, the general intersecting cutting may cause the outside of the cutting hole to be large and the inside to be small, and it is impossible to insert the tube that conforms to the outside frame of the hole. The function of **Vertical Intersection** can ensure the consistency of the size of internal and external cutting holes.

The following three intersecting methods are supported:

- Primitive is cut by normal intersection at the same angle, and the following settings can be selected:

  - Vertical intersection: Automatic setting of intersection angle, 0 ° or 180 ° intersection.
  - Horizontal intersection: Automatic setting of intersection angle, 90 ° or 270 °.
  - Common intersection: For the user defined angle of general intersection, you can manually enter the intersection angle. The parameter range is 0 ° - 360 °.
- According to the normal cutting of the respective primitive vertical intersection, it is set by the automatic angle of common intersection.

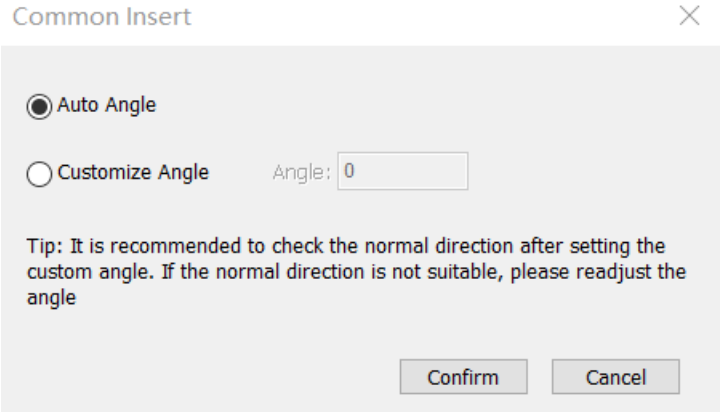
Operating premise:

Before setting vertical intersection, ensure that:

- The current cutting tube is rectangular tube, circular tube, elliptical tube or waist tube.
- The target object is non dot, text, scan group, and primitive that bypasses the entire tube by 180 °.

Operating Steps:

1. Select the object.
2. Select one of the following methods to set the vertical intersection.
   - In the **Tech** area of the **Common** menu bar, click Vertical Intersection → select the intersection method.
   - Right click and click **Vertical Intersection** → select the intersection method.
3. Perform different operations according to different intersection methods.
   - If you select **Set Vertical Insert** or **Set Horizontal Insert**, the selected graphic becomes white.
   - If **Set Common Insert** is selected, check the box in the pop-up **Common Insert** dialog box as required:
     - Auto angle.
     - Customize angle and enter the angle.



   After setting the general intersection, click **Confirm**, and the selected graphic will turn white.
   If **Show Normal** is not checked in the view page, a prompt will pop up indicating whether to show the normal.

4.3.16. Over Arris Jiggle

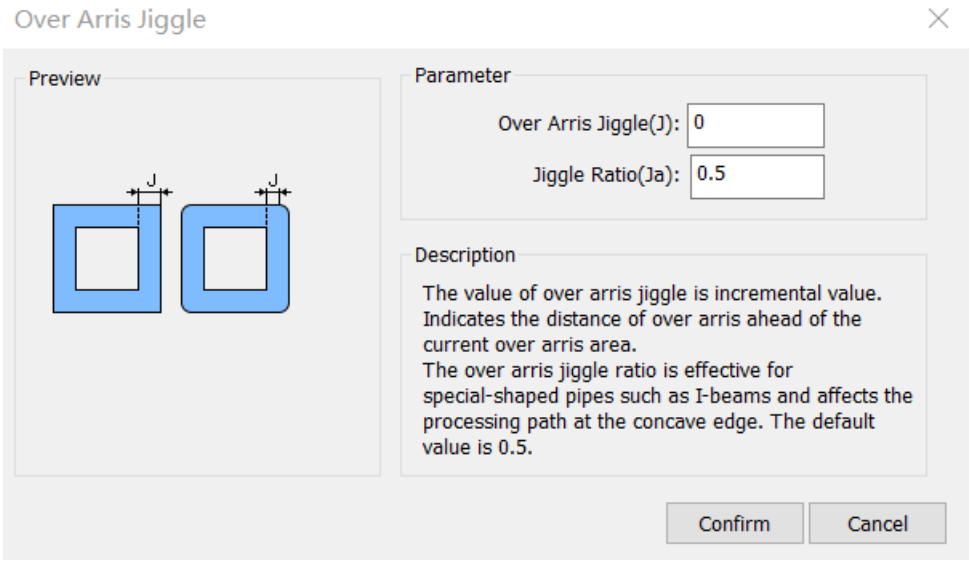
By adjusting the over arris start stop position. To solve the problem that the actual thickness of the over arris area needs to be cut becomes larger and cannot be completely

cut to the deviation between the tube drawing and the actual size of the tube or the deformation of the actual tube during machining.

Before setting over arris jiggle, you can click **View** → **Show Normal** in the **View** area of the menu bar.

Operating Steps：

1. In the **Tech** area of the **Common** menu bar, click Over Arris Jiggle to open the **Over Arris Jiggle** dialog box：



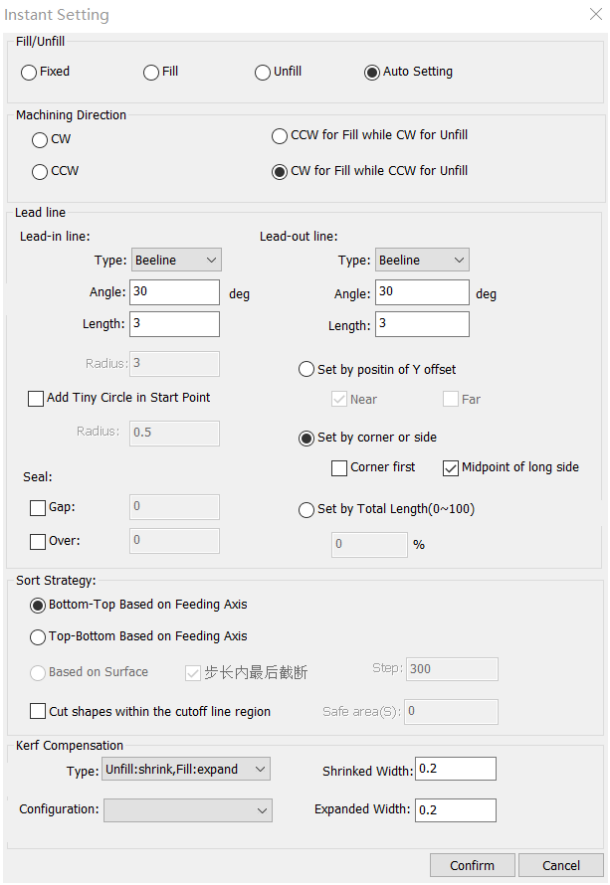
2. Set parameters **Over Arris Jiggle** and **Jiggle Ratio**.

4.3.17. Instant Setting

It is used to set unfill/fill, lead line, machining direction, machining order and kerf compensation with one click, so as to improve machining efficiency.

Operating Steps：

1. Select one or more objects.
2. Select any of the following methods to open the **Instant Setting** dialog box：
   - In the **Tech** area of the **Common** menu bar, click Instant Setting.
   - Right click and click **Instant Setting**.



3. Set as required.

4.3.18. Clear

It is used to clear some set technics.

The clearing items include the following:

- Micro joint
- Chop
- Lead line
- Cooling point
- Kerf compensation
- Inner compensation
- Weld compensation
- Vertical intersection
- Corner chop
- Centering mark

Operating Steps：

1. Select one or more objects.
2. Select the following methods to clear the technic:
   - In the **Tech** area of the **Common** menu bar, click  **Clear** to select the items to be cleared.
   - Right click and click **Clear** to select the items to be cleared.

4.4. Arrange Path

4.4.1. Sort

It is used to specify the processing order of each graphic in the tool path file.

To display the original machining sequence in the tool path, click **View** → **Show Order** in the **View** area of the menu bar.

The system supports the following sorting methods:

- Auto Sort
- Manual Sort
- Specify Sort
- Sort List
- Sort to Top/Bottom
- Mach Order

Note

Manual sort methods, such as specify sort, sort list, sort to top/bottom, and mach order can only be edited in 2D edit mode.

4.4.1.1. Auto Sort

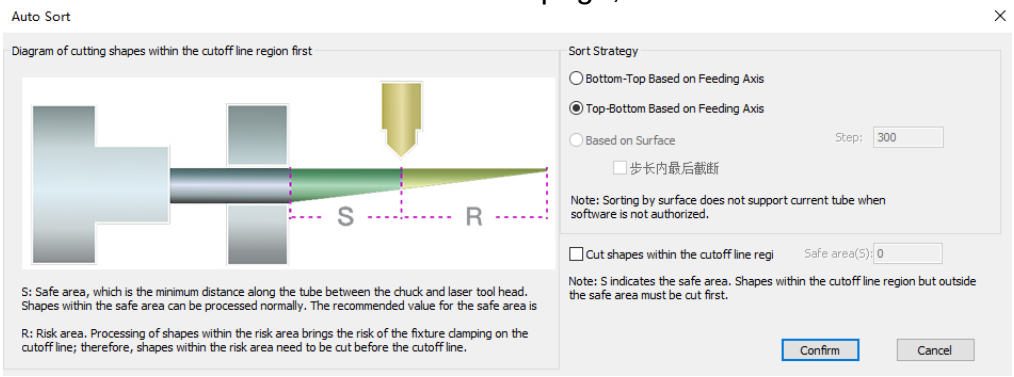
According to the selected sorting strategy, the software automatically arranges the machining sequence.

Operating Steps:

1. Select multiple objects.
2. Open the **Auto Sort** dialog box by using the following methods:
   - In the menu bar of the **Software Main Interface**, click **Common** → 

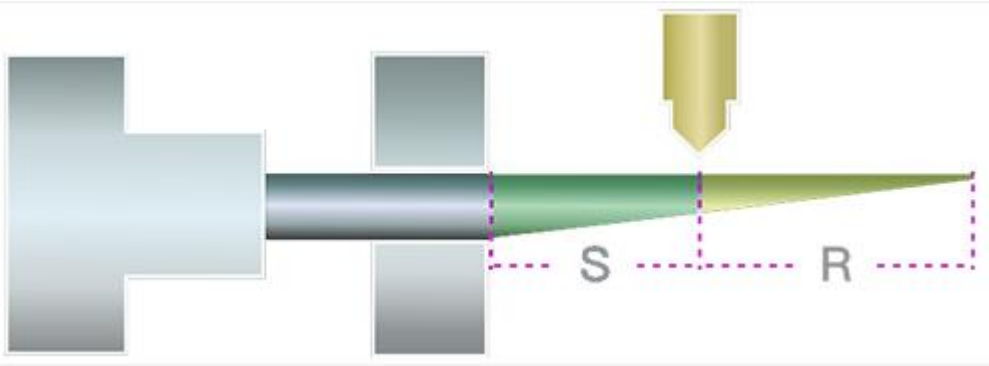
.
   - In the menu bar of the **2D Edit** page, click **Common** → 

→ **Auto Sort**.



3. Select sorting strategy:
   - Bottom-top based on feeding axis
   - Based on surface: Currently only applicable to rectangular tube.
4. **(Optional:)** Check **Cut Shapes within the Cutoff Line and set safe area (S).

Effect picture:



- S: Safe area, which is the shortest distance along the tube between the chuck and laser cutting head. Shapes within the safe area can be machined normally. The recommended value is 0.
- R: Risk area. When machining graphics in this area, there is a risk that the fixture may be holding on the cutoff line. Graphics in this area need to be cut first before the cutoff line.

5. Click **Confirm**.

4.4.1.2. Manual Sort

Use the mouse to specify or adjust the serial number to adjust the machining sequence of single or multiple graphics. The software will verify the parts, including the verification of the machining sequence of the part graphics, and check whether the part cutting line is in the first and last machining sequence. If it is not the first and last, it will automatically adjust to the first and last, and the rest of the part graphics will automatically adjust in turn.

Operating Steps:

1. In the menu bar of the **2D Edit** page, click **Common** → 

→ **Manual Sorting**.

   The cursor becomes 

, which indicates the sequential number of the object to be clicked.
2. Click the left mouse button to select the target line to be set as the first.

   The cursor becomes 

, the machining sequence on this line becomes 1. And so on.
3. To change the sequence of the last line, right-click on the area and click **Previous Order** in the context menu.
4. Right click and select **Exit** from the shortcut menu.

4.4.1.2.1. Specify Sort

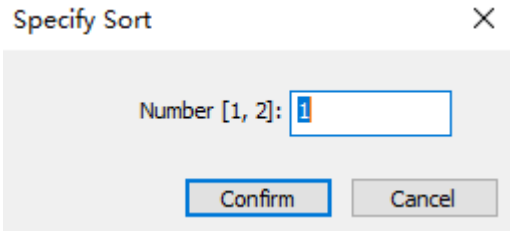
Manually specify the machining sequence of an object.

Operating Steps:

1. In **2D Edit** page, select the object.

2. In the menu bar, click **Common** → 

→ **Specify Sort** to open the **Specify Sort** dialog box:



3. Enter the specified sequence in the **Number** input box.
   N represents the maximum sequential number in the tool path file.

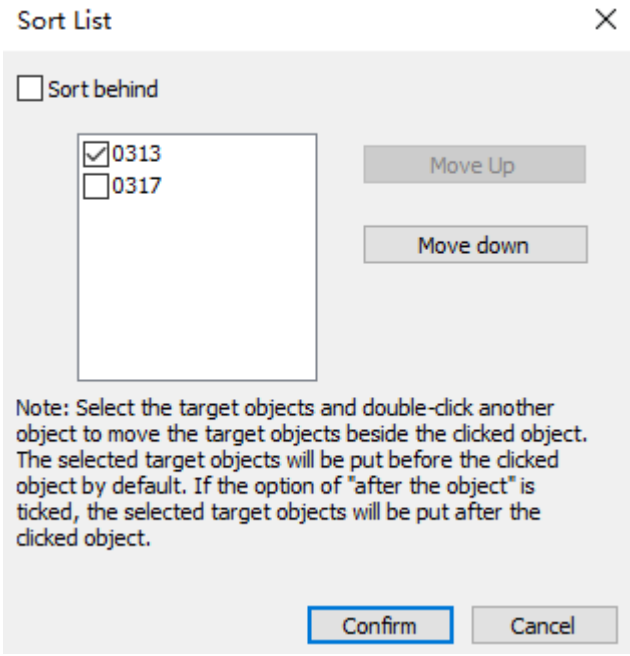
4.4.1.2.2. Sort List

The system automatically numbers each object. This operation is used to manually check the object number to change the machining sequence of its corresponding object.

Operating Steps：

1. In **2D Edit** page, select the object.
2. In the menu bar, click **Common** → 

→ **Sort List** to open the **Sort List** dialog box:



3. After checking the object, select the following method to sort:
   - Click **Move Up** and **Move Down** to move the selected object.
   - Double click an unselected object to place the selected object before it.
     If **Sort Behind** is checked, double click an unselected object to place the selected object behind it.

4.4.1.2.3. Sort to Top/Bottom

Change the machining sequence of the selected single object to the first or last.

Operating Steps：

1. In **2D Edit** page, select the single object.
2. In the menu bar, click **Common** → 

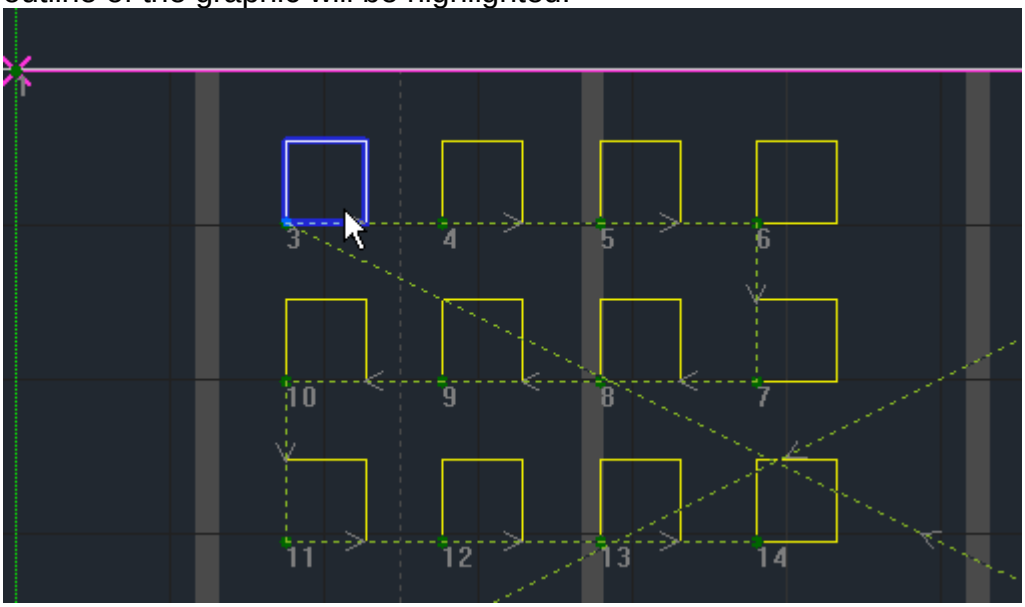
→ **Sort to Top** / **Sort to Bottom**.

4.4.1.3. Mach Order

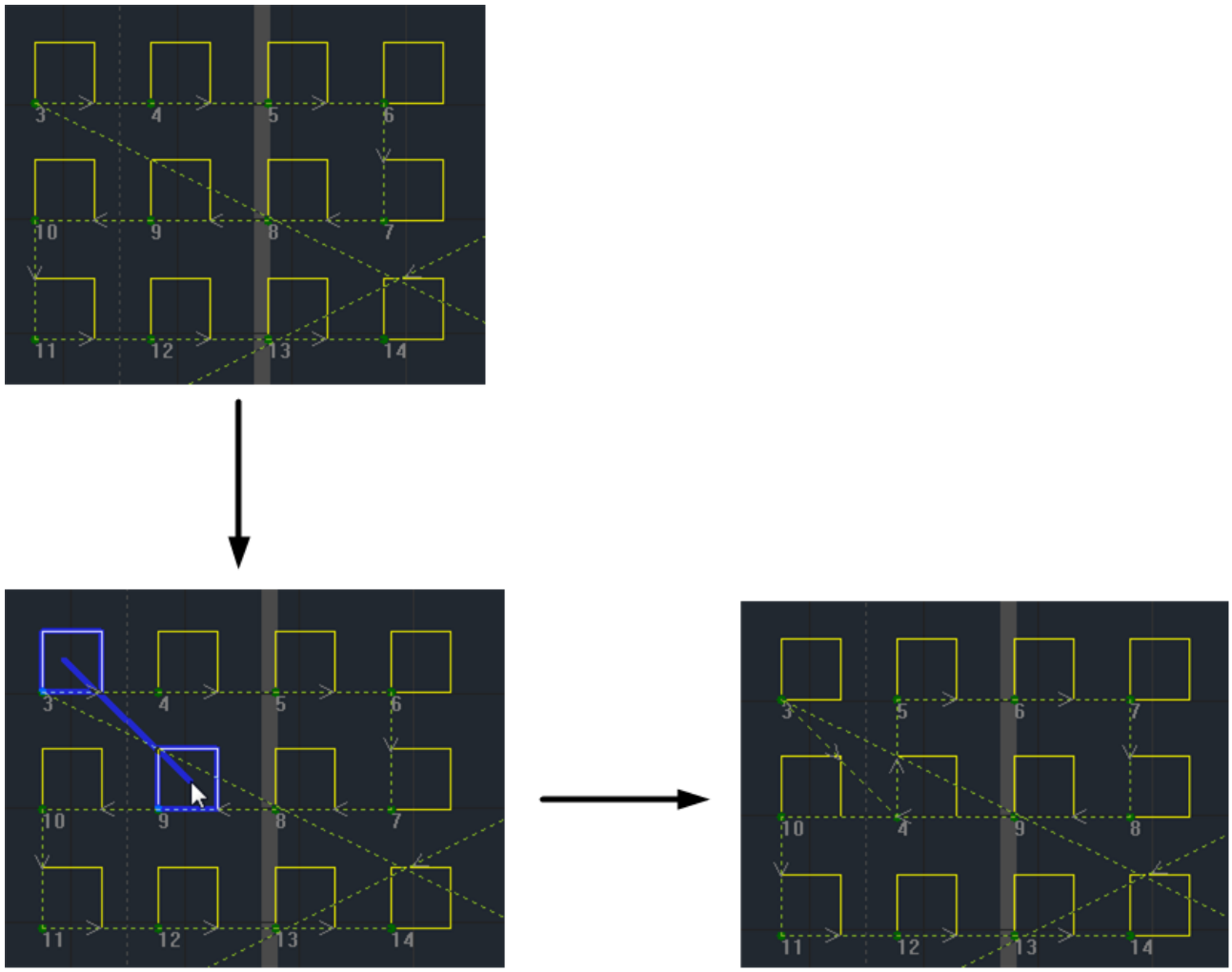
Manually connect graphics, that is, adjust the number of the next graphic according to the number of the previous graphic.

Operating Steps：

1. In the drawing area, right-click and select **Mach Order**. The empty path of the part will be displayed automatically. When the mouse passes over the graphic, the outline of the graphic will be highlighted.



2. Click to select a graphic, and click to select the next graphic. At this time, the number of the selected graphic n will be changed to n+1.
   For example, change the number 9 in the following picture to 4.



3. Right click to exit the function.

4.4.2. Scan

The scan functions allow the system to re-arrange the machining path for better efficiency and do not have the tool lifting and lowering steps required when switching between shapes during common laser cutting. During scanning machining, only laser ON or OFF operations will be performed.

If scanning parameters have already been set for the selected object(s), you need to Clean Scan.

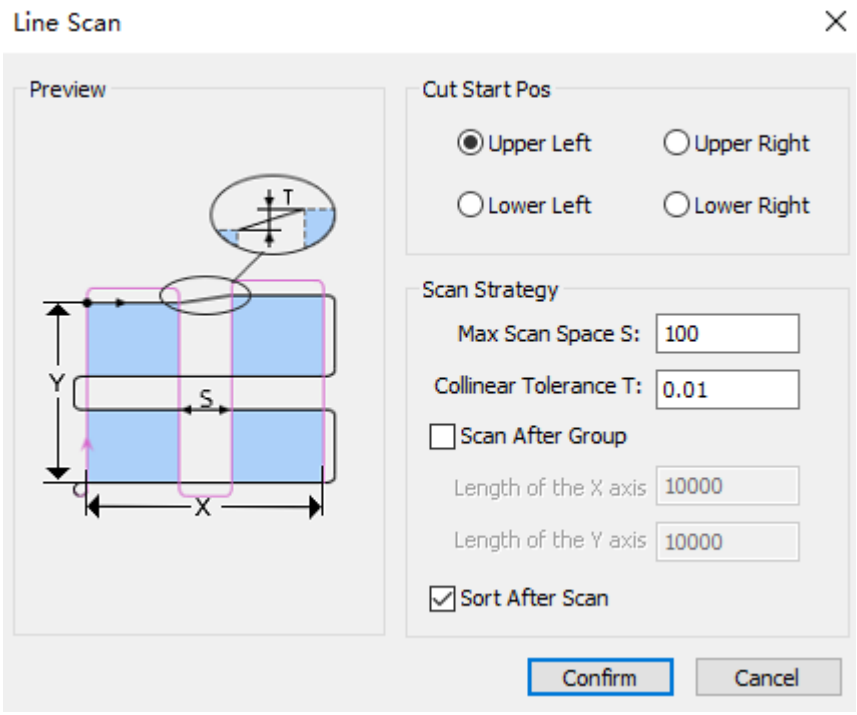
4.4.2.1. Line Scan

Straight lines can be recognized and scanned by group.

Operating Steps：

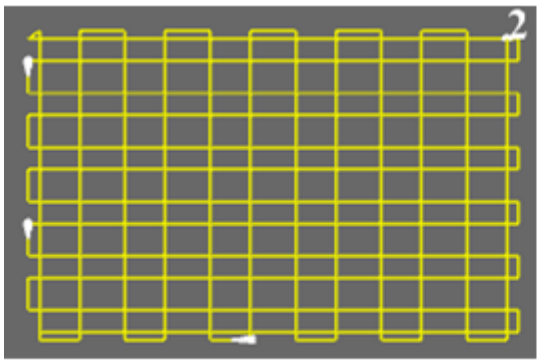
1. In **2D Edit** page, select the object.
2. In the menu bar, click **Common** → 

→ **Line Scan** to open the **Line Scan** dialog box：

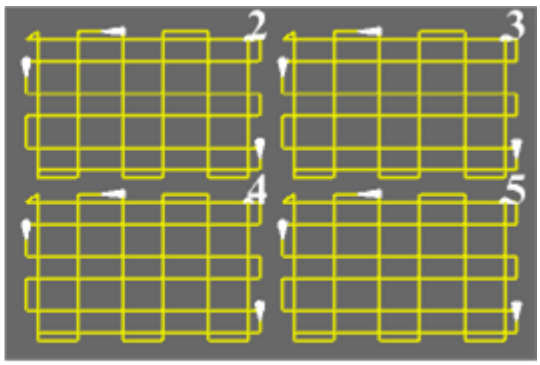


3. Select the cut start position and set the following parameters:
   - **Max Scan Space S**: If the distance between two graphics with shared edges is larger than this value, the two shapes will be scanned in two groups.
   - **Collinear Tolerance T**: If the distance between two parallel lines is smaller than this value, the two shapes are considered to have a shared edge.
4. To execute scanning by group, check **Scan After Group** and set **Length of the X axis** and **Length of the Y axis**.
   - Group: From the cut start position, graphics that within the **Length of the X axis** and **Length of the Y axis** will be counted as one group. The next group starts from the end of the first group.
   - Scan: Scan by group, that is, scan the objects in the group first, and then scan the objects in the next group.
5. **（Optional：）** To further improve machining efficiency after scanning, you can check **Sort After Scan** to re-arrange the machining path and reduce dry run distance.

Effect picture:



Ungrouped scanning



Grouped scanning

4.4.2.2. Arc Scan

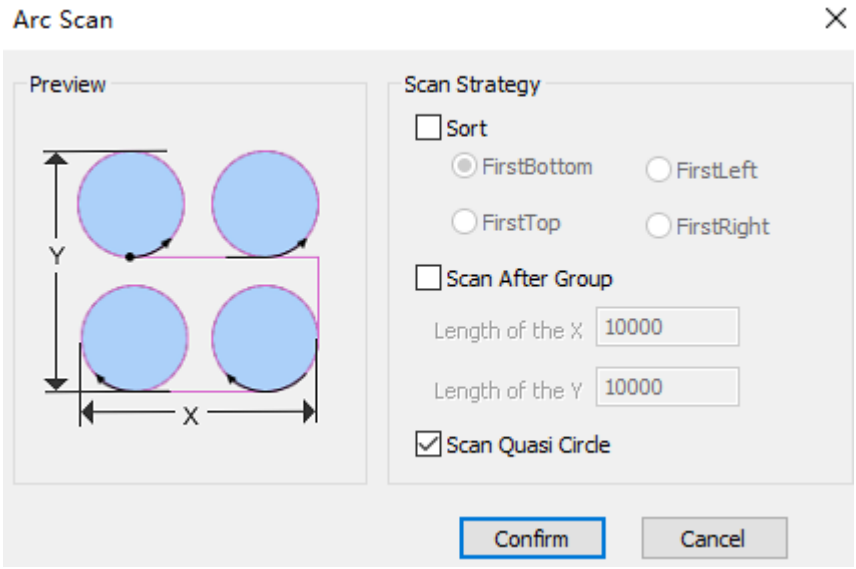
Arcs can be recognized and scanned by group, standard circles, and non-standard circles.

Non-standard circles are commonly oval or other graphics formed by arc tube intersection.

Operating Steps：

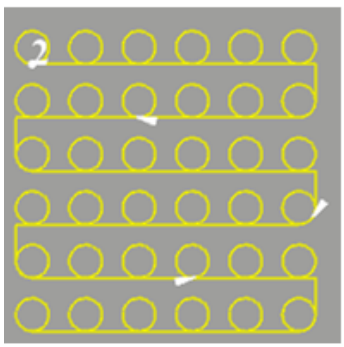
1. In **2D Edit** page, select the object.
2. In the menu bar, click **Common** → 

→ **Arc Scan** to open the **Arc Scan** dialog box：

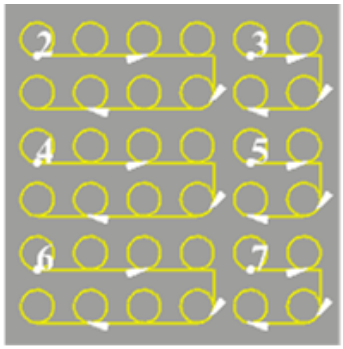


3. To execute scanning by group, check **Scan After Group** and set **Length of the X axis** and **Length of the Y axis**.
   - Group： From the start position, graphics that within the set **Length of the X axis** and **Length of the Y axis** will be counted as one group. The next group starts from the end of the first group.
   - Scan： Scan by group, that is, scan the objects in the group first, and then scan the objects in the next group.
4. If the selected objects are not standard circles, check **Scan Quasi Circle**.
5. **（Optional：）** Check **Sort** and select a sorting strategy to scan the selected objects based on the strategy.

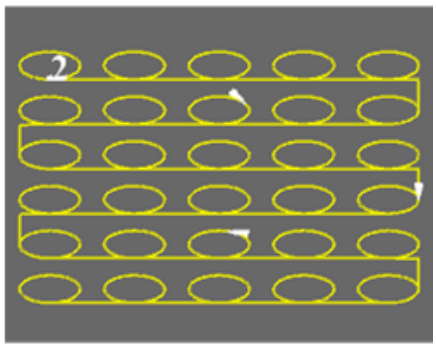
Effect picture：



Ungrouped scanning of standard circles



Grouped scanning of standard circles



Ungrouped scanning of non-standard circles

4.4.2.3. Clear Scan

Clear the tool path generated during scanning.

Operating Steps：

1. In **2D Edit** page, select the object.
2. In the menu bar, click **Common** → click 

→ **Clear Scan**.

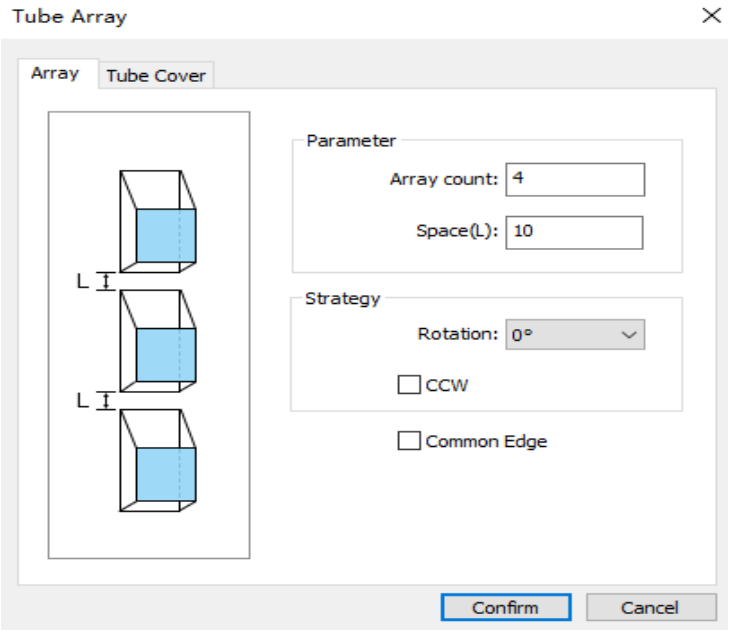
4.4.3. Tube Array

Tube array is used to copy multiple or full tubes for a single standard part according to the strategy, which is convenient for machining. Standard parts must only have two cutoff lines, one upper and one lower, and the lower cutoff line is closer to the starting surface of the tube than the upper cutoff line.

Operating Steps：

1. Select a part that contains two cutoff lines.
2. In the menu bar, click **Common** → 

to open the **Tube Array** dialog box:



1. Select array method:
   - Array: The selected single part is arrayed according to the set quantity and strategy.
   - Tube cover: Cover the selected single part in the whole tube array according to the set strategy.
2. Set the count of array or the tube length and array space.
3. Select strategy:
   When switching strategy, schematic diagrams of each strategy are visible on the left side of the dialog box.

Parameter	Description
Rotation	It refers to the rotation and translation array of tubes in the Y direction. Supported angles: 0 °, 90 °, 180 °, 270 °.
CCW	It means that the tube is turned 180 ° in the Z direction.

4. **（Optional：）** If the cutoff line that can be regarded as the same object within an allowable error, to reduce overlapped edge, check **Common Edge**.
5. After setting, click **Confirm** The system starts array nest, and the graphic is automatically generated to the drawing area.

4.5. Simulation

4.5.1. Execute Simulation

Simulation is used to check whether the machining area and tool path are proper before formal machining.

The simulation is divided into：

- **Simulation：** Used to check the toolpath in real time without running the machine tool. It provides a fast but lifelike simulation machining environment. During simulation, no actual machining occurs, and only the moving track of the cutting head at high speed shows in drawing window.
- **Dry Run：** Control the machine tool to run the program without turning on the laser and machining related ports, and check whether the machining path is correct.

Operating Premise：

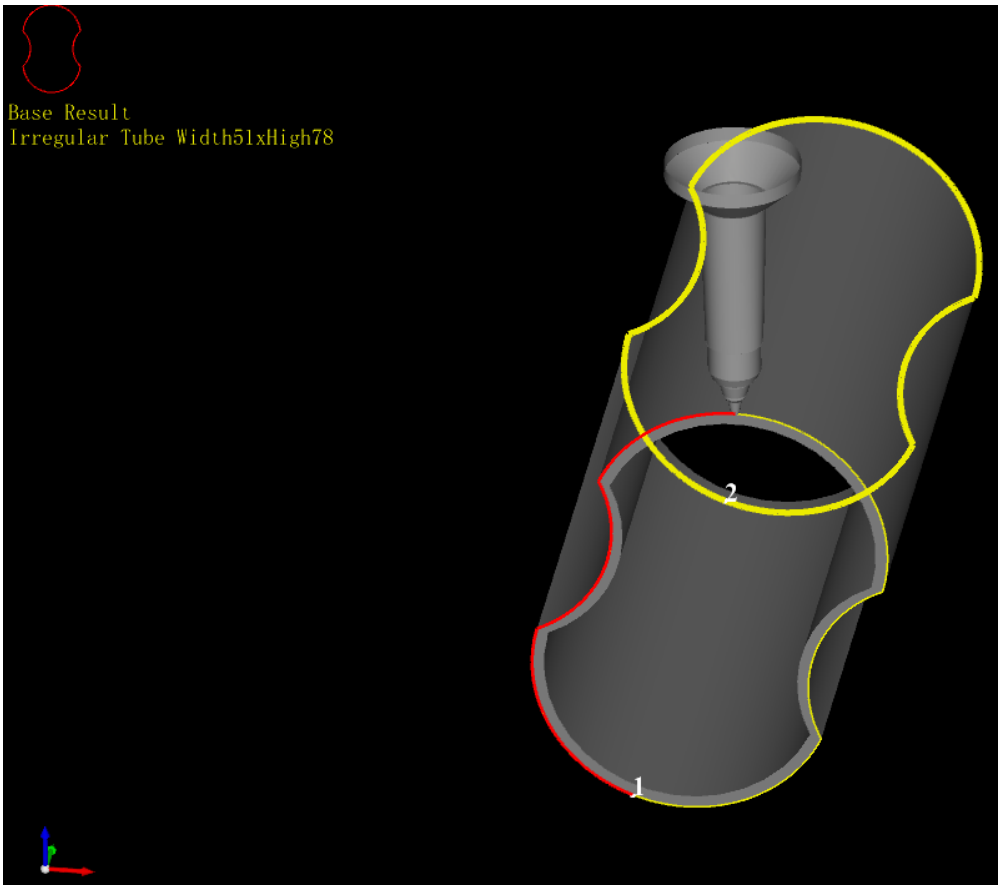
Ensure that the machining file has been saved and the current system status is **Idle**.

Operating Steps：

- **Simulation**

1. In the machine control bar, click 

. The software automatically executes high speed simulation machining from the first segment of the machining program. Effect picture：



- **Dry Run**

1. In the machine control bar, click 

.

5. Machining Related Operation

5.1. Execute Return to Mechanical Origin or Set Datum

When installing the software, if it is configured as bus, it can return to the mechanical origin or set the datum; If it is configured as non bus, it can only execute back to the mechanical origin. Take the bus as an example.

Encoder type:

- Incremental encoder: Execute Return to Mechanical Origin
- Absolute encoder: Set Datum

The encoder type can be set by finding the parameter **Encoder Type** in **System Parameter**.

5.1.1. Execute Return to Mechanical Origin

The mechanical coordinate origin of the machine tool is the mechanical origin, or called the mechanical zero point. The mechanical coordinate system of the machine tool is unique and has been determined when the machine tool leaves the factory.

Return to the mechanical origin to synchronize the mechanical coordinate system of the system with the mechanical coordinate system of the machine tool, so you must return to the mechanical origin before machining.

When axis B returns to the mechanical origin, only one signal is detected and the chuck clamping state is detected before Y/B returns to the origin.

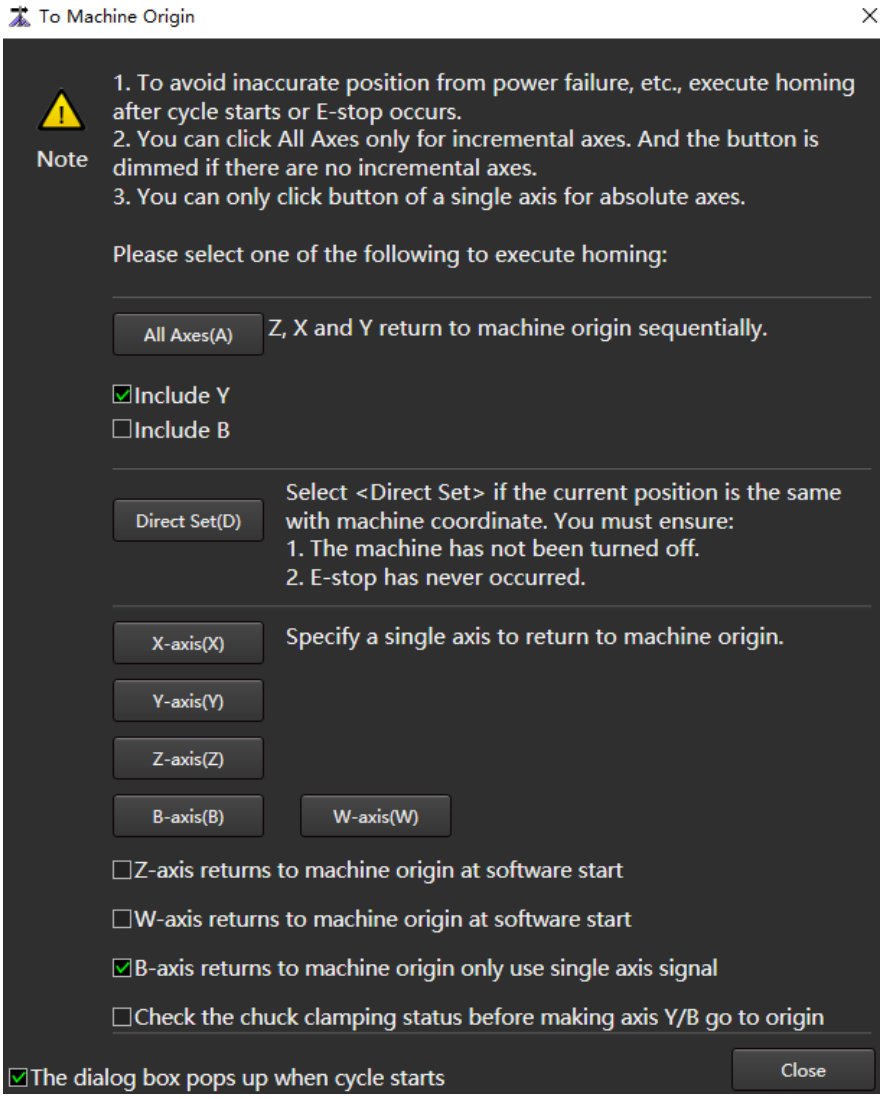
Operating Premise:

Before returning to the mechanical origin, ensure that all servo alarms have been cleared.

Operating Steps:

1. In the menu bar, click **Machining** → 

→ **Homing Setting** to open the **To Machine Origin** dialog box:



The dialog box pops up after the software starts by default. If you need to cancel the setting, uncheck **This Dialog Box Pops Up when Cycle Starts**.

2. **（Optional：）** Check the following parameters as required：
   - The B axis returns to the machine origin only use single axis signal: if checked, the B axis returns to the mechanical origin with only one position signal of the B axis mechanical origin.
   - Check the chuck clamping state before making axis Y/B go to origin: if checked, check chuck clamping state before Y/B returns to the original point, and do not return to the mechanical original point when the chuck is clamped.
3. Select one of the following methods：
   - Click **All Axes** to automatically return to the mechanical origin in the order of Z axis first, then X, Y, W axis.

     In the machine tool control bar, click 

to execute all axes return to mechanical origin.
   - Click the button corresponding to a single axis to return to the mechanical origin for each axis.

In the machine tool control bar, click 

to execute Z axis return to mechanical origin.

- If the mechanical coordinate of the current position is consistent with the actual mechanical coordinate of the machine tool, and the machine tool has not been turned off or has not experienced any abnormal conditions such as servo alarm, click **Direct Set** to set the current point as the mechanical origin.

- In the machine tool control bar, click → 

to select the method of returning to the mechanical origin.

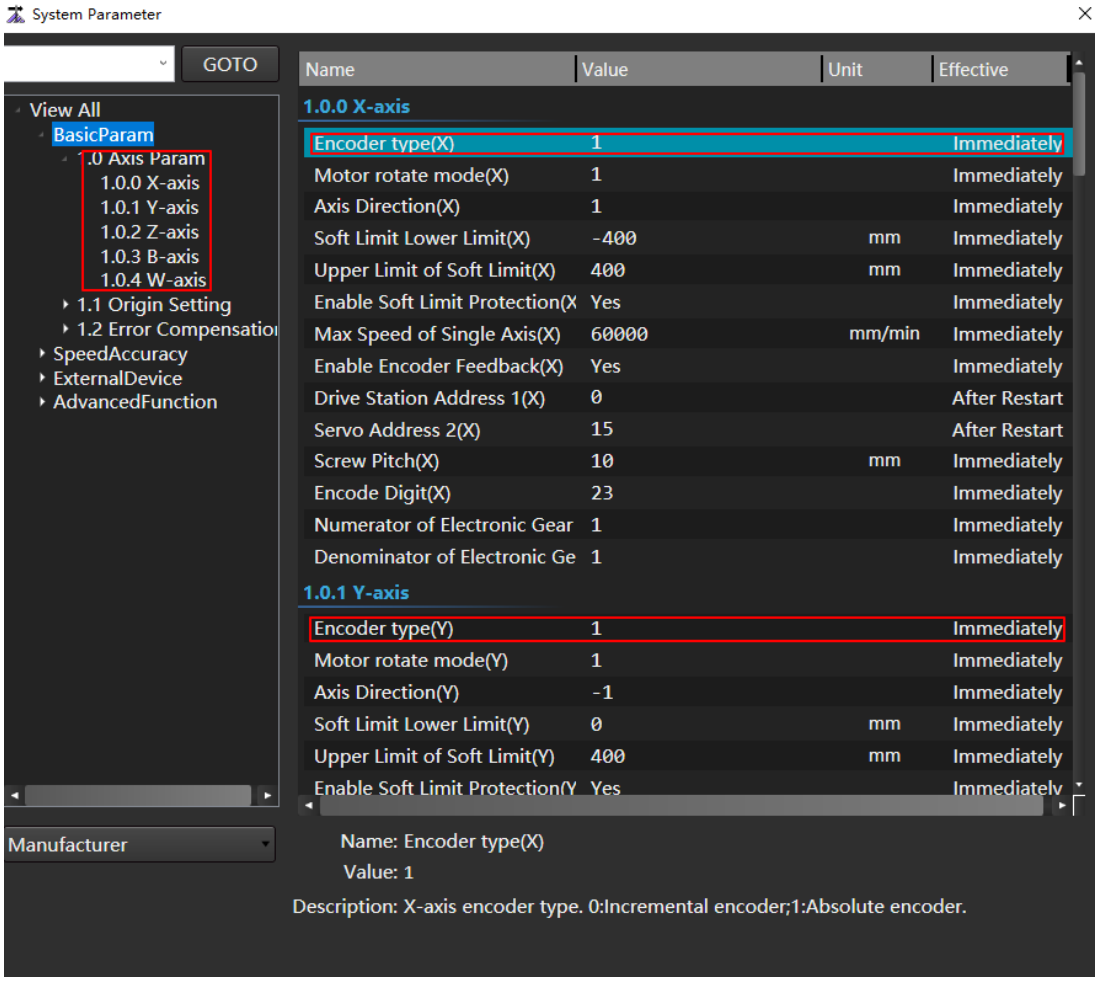
After executing returning to the machine origin, sign ▪ appears in front of each axis in the machine tool control bar.

5.1.2. Set Datum

The motor of absolute encoder can set the current cutting head position as the mechanical origin position through reference.

Operating Premise：

The encoder type of the corresponding axis in the system parameters is 1: absolute value.



Operating Steps：

1. In the menu bar, click **Machining** → Axis Datum Setting.
2. Select one of the following methods:
   - Click **All Axes** to automatically set the datum.
   - Click the button corresponding to a single axis to set the datum for each axis.

You can select **Cancel(All)** to cancel datum setting.

After setting datum, sign appears in front of each axis in the machine tool control bar. And after starting the software or releasing E-stop, the system automatically updates the machine coordinate and feedback coordinate according to feedback pulses to make the current machine coordinate match with the actual coordinate.

5.2. Axis Adjust

For absolute type encoders, after the mechanical origin mark disappears, the encoder data can be obtained through the axis adjust function, and the mechanical coordinates of the axis can be synchronized to the software.

Operating Steps:

1. In the menu bar, click **Machining** → Adjust.
2. Select one of the following methods:
   - Click **All Axes** to automatically set adjust.
   - Click the button corresponding to a single axis to adjust each axis.

5.3. To Fixed Point

Set the fixed point, and select to move the cutting head to return to the fixed point after the machining.

Operating Premise:

The fixed point position parameters in the **4.1 Fixed Point** have been set. For details, see System Parameter.

Operating Steps:

1. In the menu bar, click **Machining** → To Fixed Point.

5.4. Calibrate B-axis Center

Calculate the mechanical coordinate value of the rotation shaft center through the calibration action. This function is only needed to recalibrate the B-axis center after initial use or mechanical deviation.

Because different machining actions require different centers, the system provides two calibration methods and can use up to two B-axis centers at the same time:

- Standard: It is used to calibrate the B-axis center during ordinary machining.
- Special: It is used to calibrate the B-axis center for special machining actions. Multiple commonly used B-axis centers can be set and named. When in use, it can be configured according to requirements.

Operating Premise:

Before calibrate the B-axis center, ensure that:

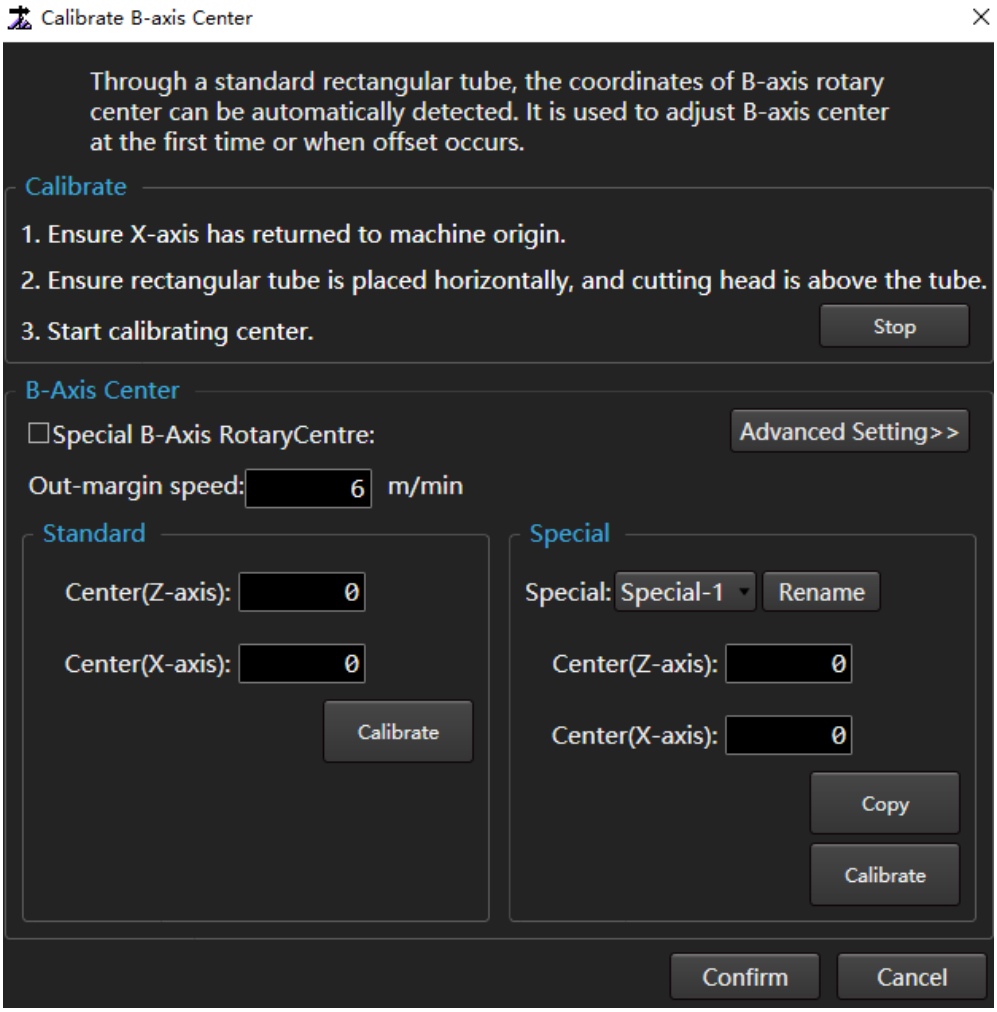
- The X axis has returned to the mechanical origin.
  For details, see Execute Return to Mechanical Origin.
- The tube size parameter settings are the same as the actual tube.

For details, see Set Tube Size.

Operating Steps：

1. Clamp a standard rectangular tube without chamfer.
2. Ensure that the clamping state of the rectangular tube is close to horizontal, and move the cutting head to the upper part of the tube.
3. In the menu bar, click **Machining** → 

to open the **Calibrate B-axis Center** dialog box:



4. In the **Standard** area, click **Calibrate**. After calibration, the calibration results will be displayed in the **Standard** area.
   If you are not satisfied with the measurement results, click **Center (Z-axis)** or **Center (X-axis)** in the **Standard** area to manually enter the coordinates.
5. **（Optional：）** If there are special B-axis center requirements, perform the following operations:
   a. In the **Special** drop-down box, select the item to be set.
   b. **（Optional：）** Click **Rename** to modify the name. Click **Rename** to rename the set item for easy identification.
   c. In the **Special** area, click **Calibrate**. After calibration, the calibration results will be displayed in the **Special** area.

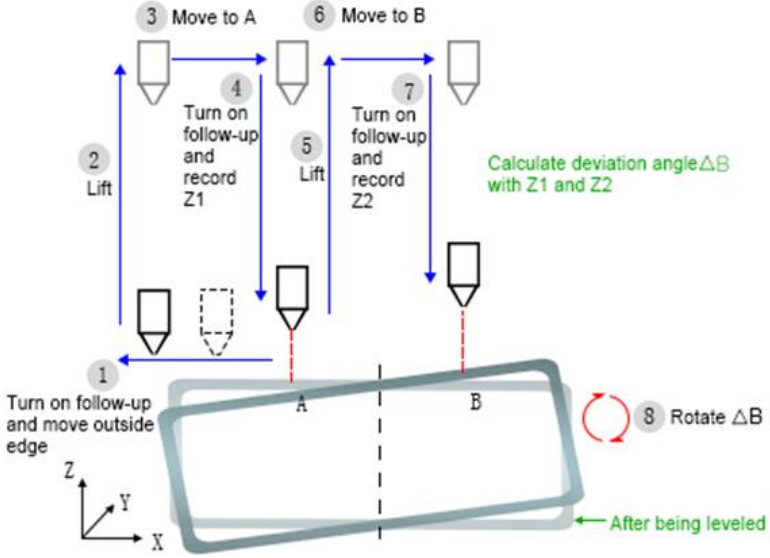
If you are not satisfied with the measurement results, click **Center (Z-axis)** or **Center (X-axis)** in the **Special** area to manually enter the coordinates.
You can also click **Copy** to copy the calibration center in the standard area to the special area, and then modify it.

d. Click **Advanced Setting** to set the rotation center of **Normal** and **Feed Cutting** in the pop-up **Advanced Setting** dialog box.
e. Check **Special B-axis Rotary Center**. During machining, the center of B-axis adopts the strategy in **Advanced Setting**.
If it is not checked, the center of Axis B uses the coordinates set in the **Standard** area during machining.

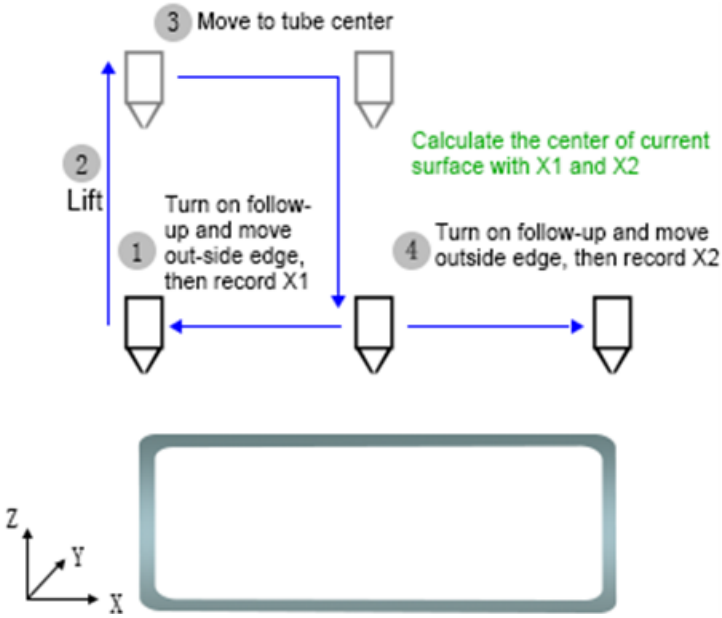
6. Input the edge exit speed followed by the cutting head in the **Out-margin Speed** input box.
7. Click **Confirm**.

The system starts to perform the calibration action:

1. Leveling tube:



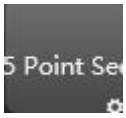
2. Find the B-axis center:
Take one side of the tube as an example:



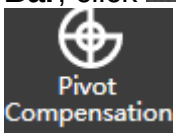
5.5. Central Compensation

There is a deviation between the tube center and the actual rotation center. With the center compensation function, the tube machining accuracy is higher.

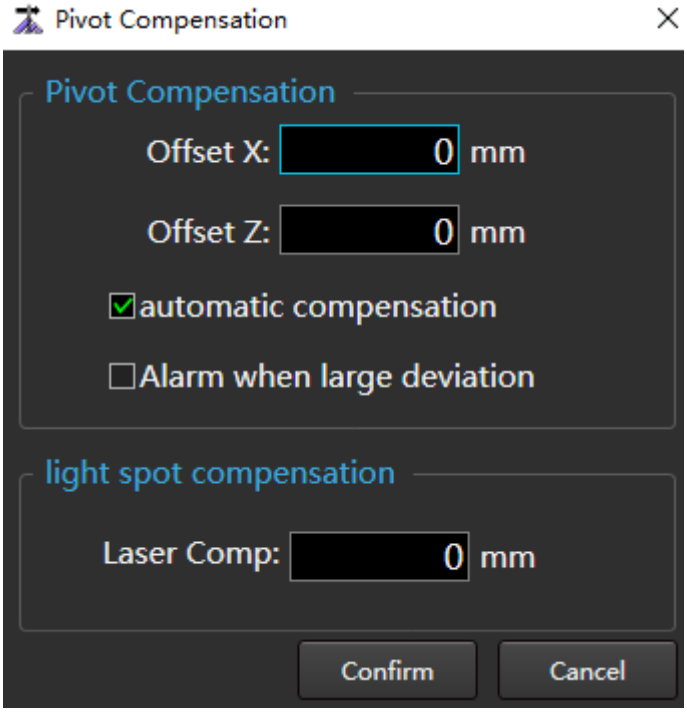
Operating Steps：

1. Perform the tube centering operation.
   a. In the **Machine Tool Control Bar**, click 

to open the **Centering Mode** dialog box.
   b. Select the centering mode and click **Confirm**.
   c. In the **Machine Tool Control Bar**, click 

.
2. In the menu bar, click **Machining** → 

to open the **Pivot Compensation** dialog box：



3. Set the following parameters in the **Pivot Compensation** area：
   - Offset X： Offset of tube center in X direction.
   - Offset Z： Offset of tube center in Z direction.
   - Automatic compensation： Turn center compensation on or off.
4. Set the following parameters in the **Light Spot Compensation** area：
   Laser comp： Used when all four face shapes are offset in the same direction. Negative values are left and positive values are right.
5. Click **Confirm**.

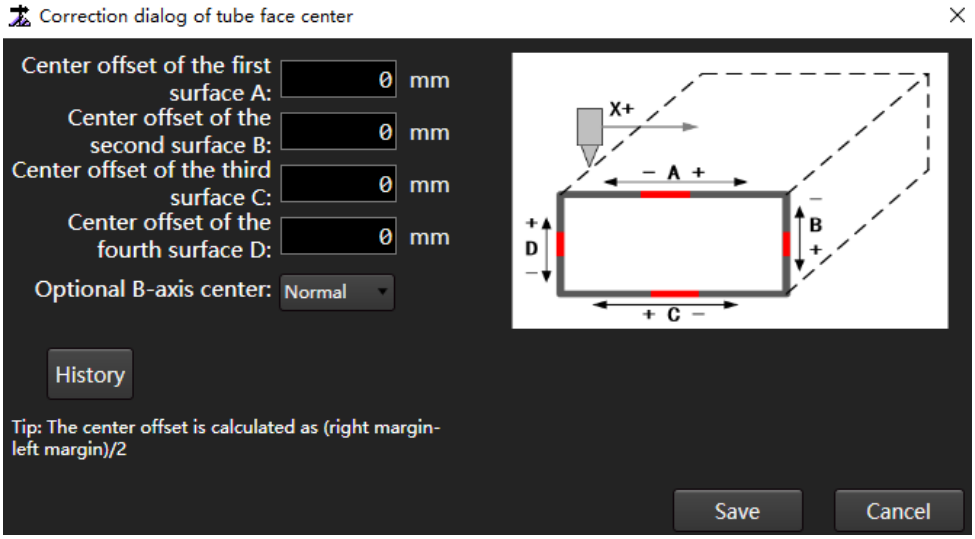
5.6. Correction Dialog of Tube Face Center

Manual correction of B-axis center and center compensation.

Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Correction Dialog of Tube Face Center** dialog box:



2. Set the following parameters:
   - Center offset of the first surface A
   - Center offset of the second surface B
   - Center offset of the third surface C
   - Center offset of the fourth surface D
   - Optional B-axis center

   Note

   For the description of the center offset of each axis, see the schematic diagram of the **Tube Circle Center Correction** dialog box. The calculation method of center offset is (right margin left margin)/2.

3. Click **Save**.

5.7. Calibrate the Tube Center

5.7.1. Level and Center

Before using the level and center function, calibrate the B-axis center. For details, see Calibrate B-axis Center.

The process of levelling and centering the tube differs in the tube type.

This section takes rectangular tubes as an example.

Operating Steps：

Click in the machine tool control bar 

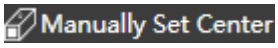
to level and center the tube by centering the tube through five points, finding the tube center and doing center compensation for the tool path:

1. Turn on follow-up separately on the left and right of the first face and level the tube according to the difference between the two results of follow-up height.
2. Rotate the tube 90°, and turn on follow-up on the center of the current face.
3. Repeat process **2** twice.
4. According to the five results of follow-up height and chuck center from calibrating B-axis, automatically calculate the center compensation for each face.

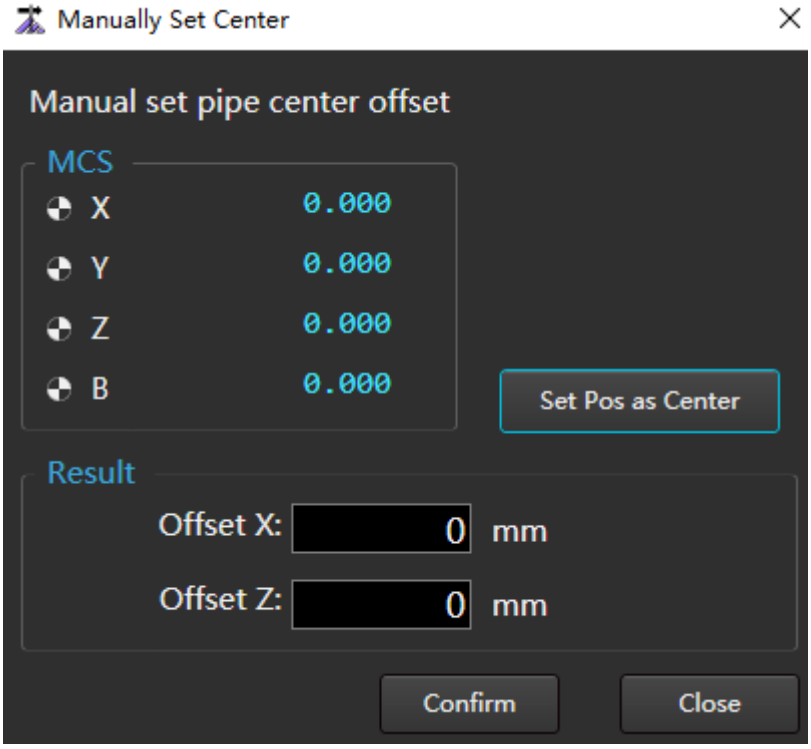
5.7.2. Manually Set Center

It is used to set the current position as the tube center for short tubes. It is mainly used when the result of levelling and centering the tube is not incorrect.

Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Manually Set Center** dialog box:



2. Rotate the tube so that it is horizontal.
3. Move the cutting head to the center of the first surface.
4. In the **Manually Set Center** dialog box, click **Set Pos As Center**.

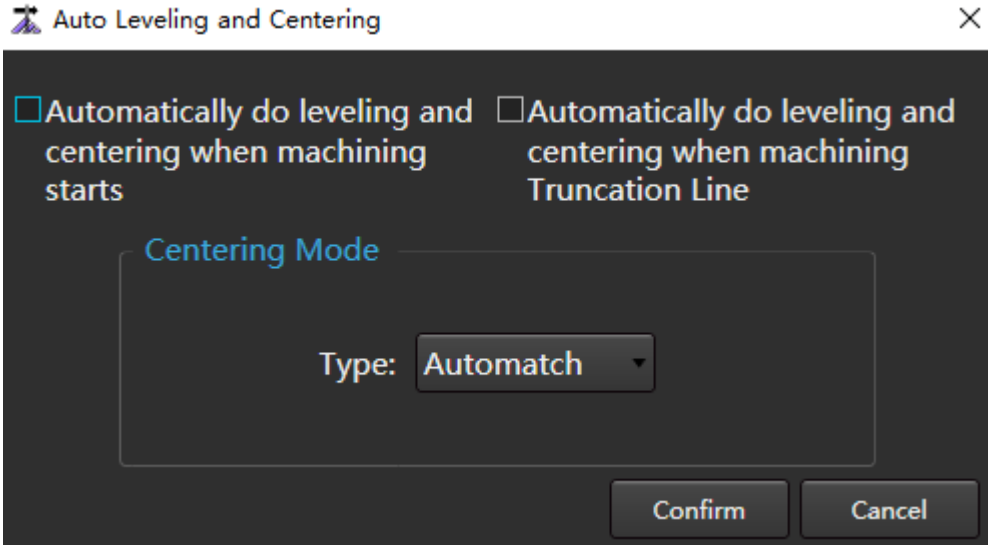
5.7.3. Auto Leveling and Centering

When the tune is long, there will be some bending deformation in the middle of the front and rear chucks, resulting in that after cutting a section of length, the centering data performed before machining cannot continue to apply. This operation can eliminate this error. After cutting a certain length, the tube will be automatically leveled and centered, and then the breakpoint will continue automatically.

If it is necessary to add the marking process of the center position in the tube. See Centering Mark.

Operating Steps：

1. In the menu bar, click **Machining** → Auto Leveling and Centering to open the **Auto Leveling and Centering** dialog box:



2. Check **Automatically Do Leveling and Centering when Machining Starts** as required, and automatically perform a leveling and centering action before each machining.
3. Check **Automatically Do Leveling and Centering when Machining Truncated Line** as required, that is, when cutting the truncated line, centering first and then cutting.
4. Select a method from the **Type** drop-down button.
   - Auto match: Automatic matching center method from machine control bar 

button. The text on the button indicates the currently selected centering method.
   - Single side leveling
   - 5 point seek center
   - 4 point seek center
   - Ellipse seek center
   - L steel seek center （clock wise）
   - L steel seek center （anticlock wise）
   - Multi edge seek center
   - Out edge seek center
   - Laser seek center
5. Click **Confirm**.

5.8. Advanced Tool

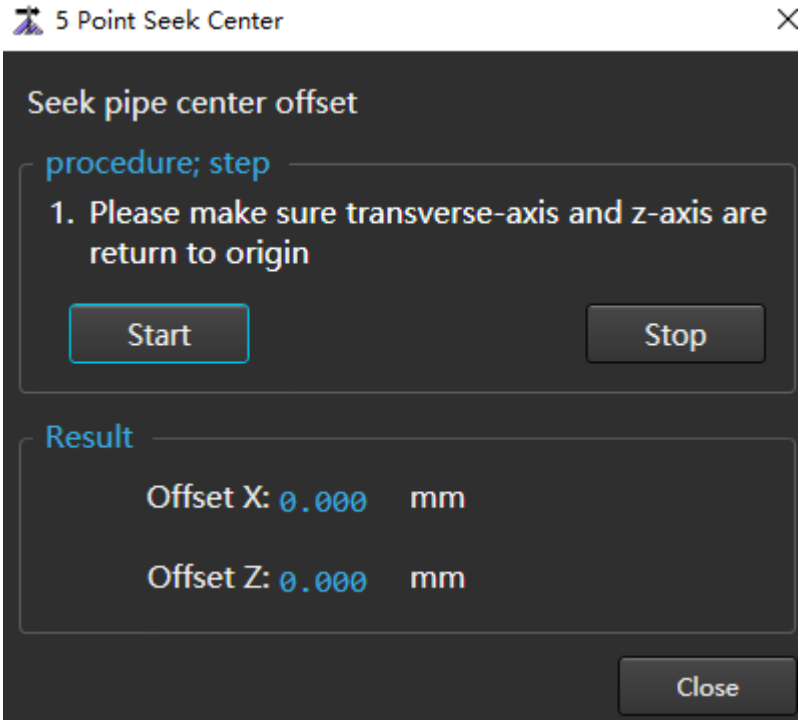
Manually debug the action of the centering method, that is, to find the deviation between the tube center and the mechanical center. The following centering methods can be debugged:

- Single side leveling

- 5 point seek center
- 4 point seek center
- Ellipse seek center
- L steel seek center
- Multi edge seek center

Operating Steps：

1. In the menu bar, click **Machining** → Advanced Tools → Select a centering method, such as **5 Point Seek Center**, to open the **5 Point Seek Center** dialog box:



2. Follow the requirements of the **Procedure; Step** area.
3. Click **Start**. After centering, the center deviation feedback is in the **Result** area. If you need to stop centering, click **Stop**.

5.9. Technic Library

5.9.1. Manage Technic Library

With the manage technic library function, you can modify, import, backup, restore and delete the parameters of existing layer technics, and add and delete material types.

If it is necessary to add a layer technic. For details, see Set Layer Technic.

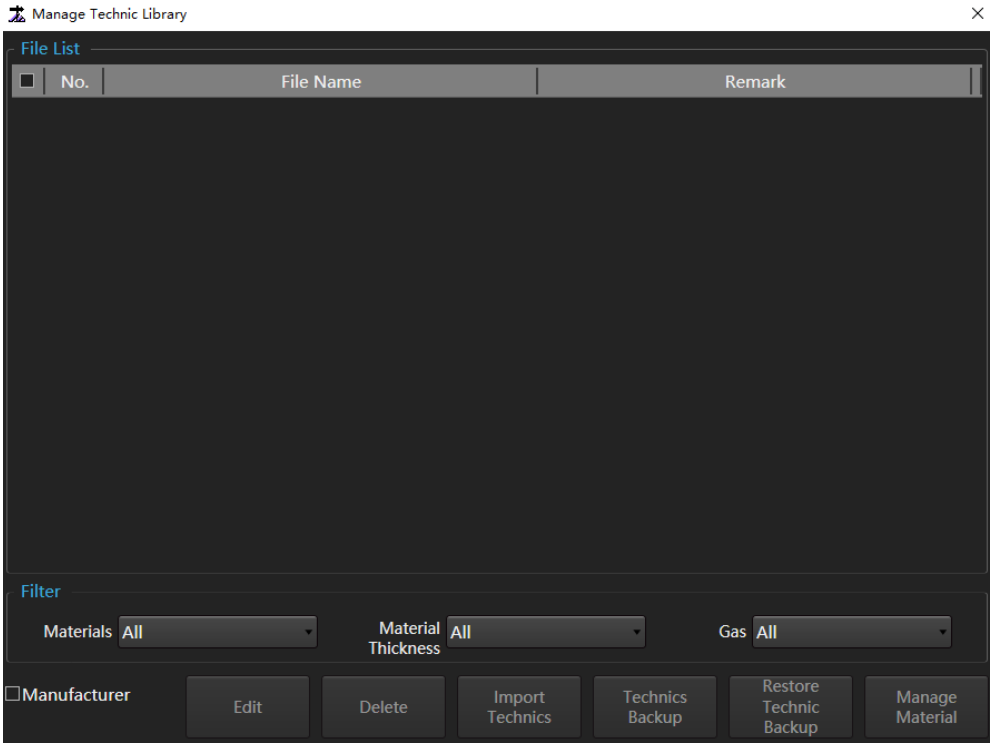
5.9.1.1. Modify Technic

Reset layer parameters.

Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Manage Technic Library** dialog box:



2. Check **Manufacturer** to activate the button.
3. Click a technic and it becomes highlighted. Click **Edit** to open the **Technic Info.** dialog box.

Note

If there are too many documents in the technic document list, you can filter through the condition drop-down boxes in the **Filter** area below.

4. Modify layer parameters. For details, see Set Operation.
5. After modification, click **Confirm**, and the modified technic will be automatically updated to the manage technic library list.
6. **（Optional：）** If need to delete a technic, select the technic and click **Delete**.
7. **（Optional：）** If need to view all technic files, click **Show All**.

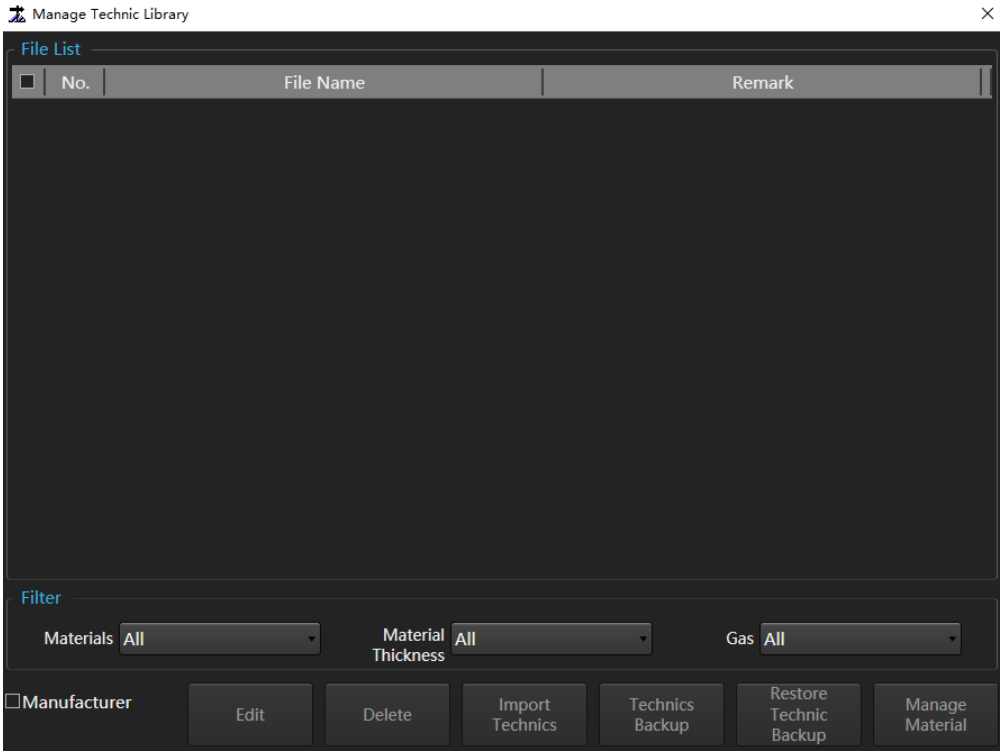
5.9.1.2. Import Technic

Import the local technic to the technic library.

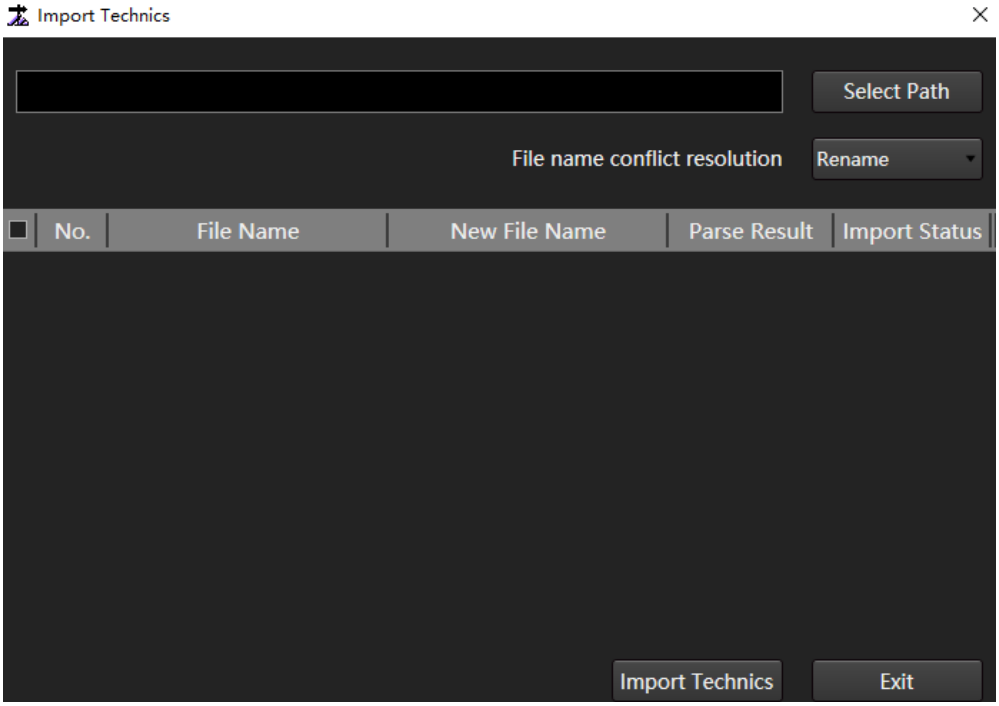
Operating Steps：

1. In the menu bar, click **Machining** → 

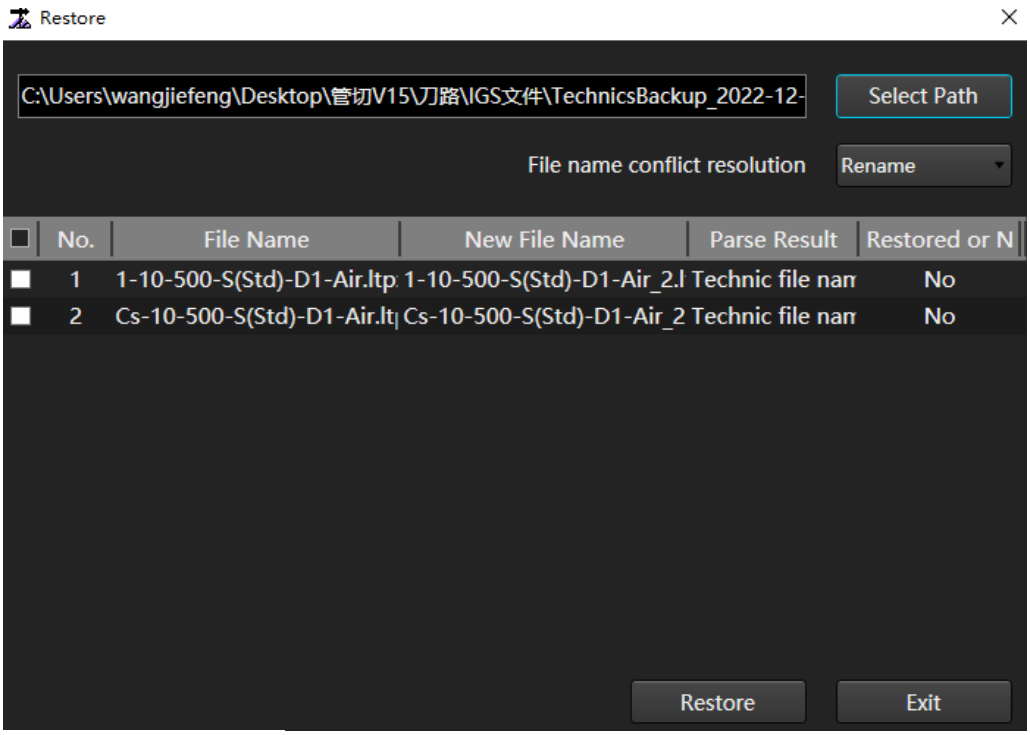
to open the **Manage Technic Library** dialog box:



2. Check **Manufacturer** to activate the button.
3. Click **Import Technics** to open the **Import Technics** dialog box:



4. Click **Select Path**, select the source file, and the **Import Technics** dialog box displays the technic information, analysis results, and import status:



Note

There are two kinds of parse results:

- Success: The technic information of the file to be imported is different from that of the file in the technic library.
- Technic film names conflict: The technic information of the file to be imported is identical to that of the file in the technic library.

5. **（Optional：）** If the analysis result is a technic conflict, select the following methods as needed to resolve the conflict:
   - Rename: Rename the file to be imported and save it in the technic library.
   - Cover: Replace the technic information of the file in the technic library.
   - Skip: Keep the files to be restored and the files in the technic library respectively. That is, the restore operation is not performed.
6. Check the technic file to be restored and click **Import Technics** to view the import status list.
7. Click **Exit**.

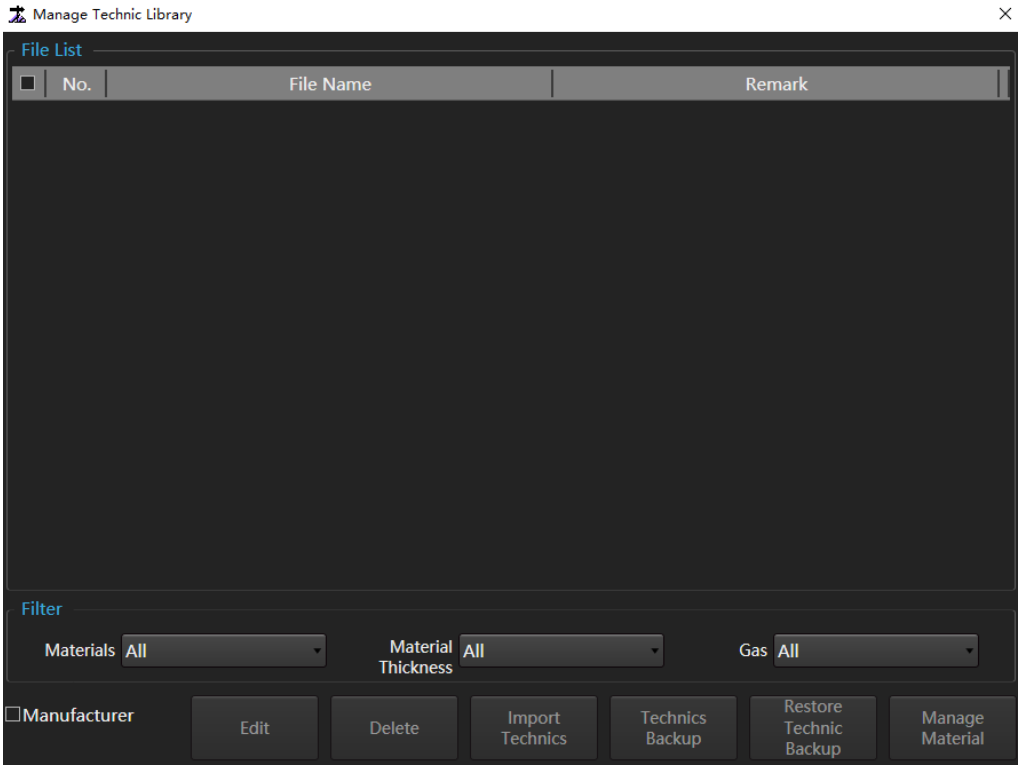
5.9.1.3. Backup Technic

Save the selected technic in the technic list to the specified storage path.

Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Manage Technic Library** dialog box:



2. Check **Manufacturer** to activate the button.
3. Check one or more processes and click **Technics Backup** to open the **Technics Backup** dialog box.
4. Modify the file name and select the saved path.
5. Click **Save**.

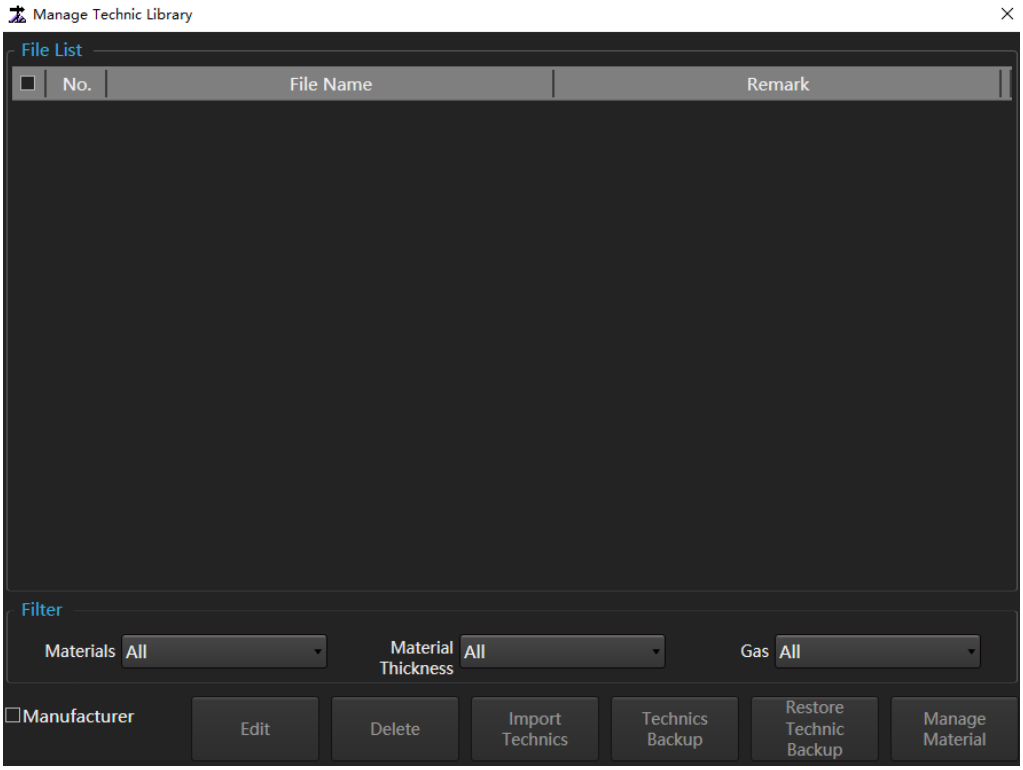
5.9.1.4. Restore Technic

Restore the stored backup technic to the technic library.

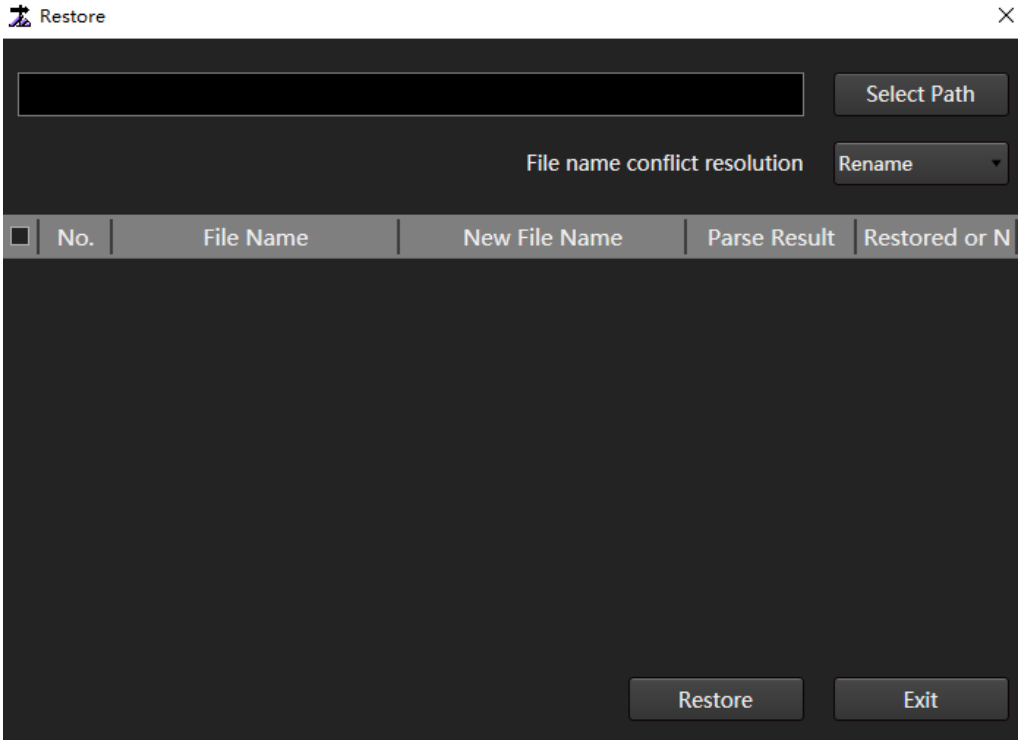
Operating Steps：

1. In the menu bar, click **Machining** → 

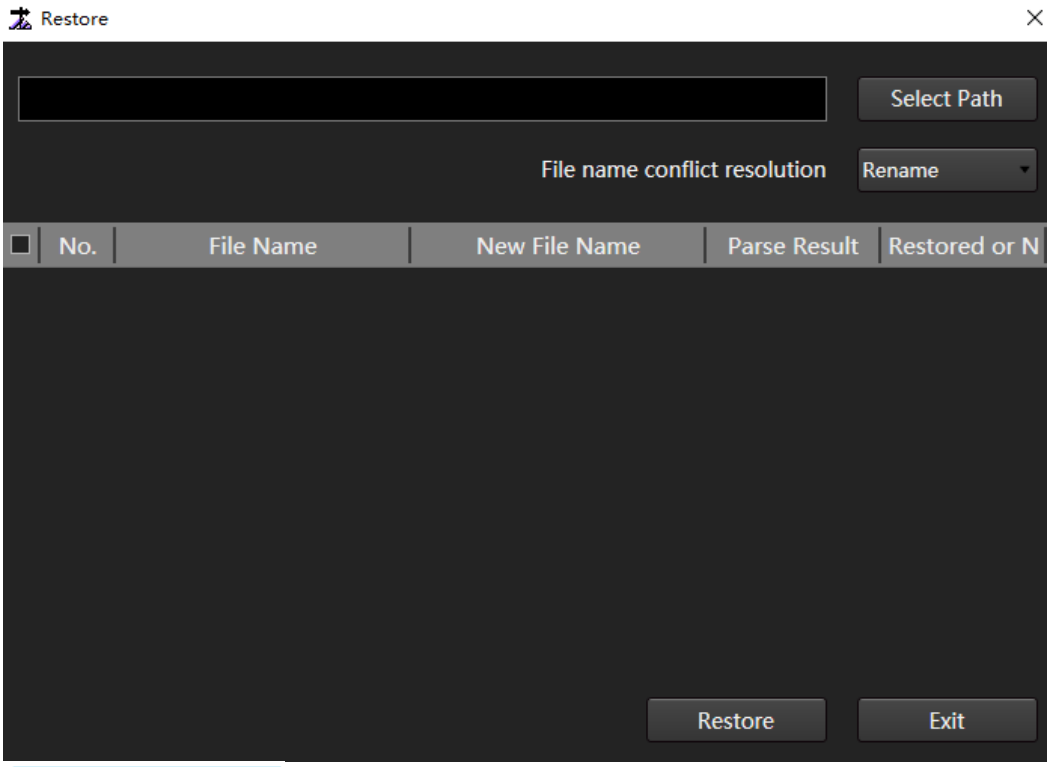
to open the **Manage Technic Library** dialog box：



2. Check **Manufacturer** to activate the button.
3. Click **Restore Technic Backup** to open the **Restore** dialog box:



4. Click **Select Path**, select the source file, and the **Select Backup Technic** dialog box displays the technic information, parse results, and restore status:



Note

There are two kinds of parse results:

- Success: The technic information of the file to be imported is different from that of the file in the technic library.
- Technic film names conflict: The technic information of the file to be imported is identical to that of the file in the technic library.

5. **（Optional：）** If the analysis result is a technic conflict, select the following methods as needed to resolve the conflict:
   - Rename: Rename the file to be imported and save it in the technic library.
   - Cover: Replace the technic information of the file in the technic library.
   - Skip: Keep the files to be restored and the files in the technic library respectively. That is, the restore operation is not performed.
6. Check the technic file to be restored and click **Restore** to view the restore status list.
7. Click **Exit**.

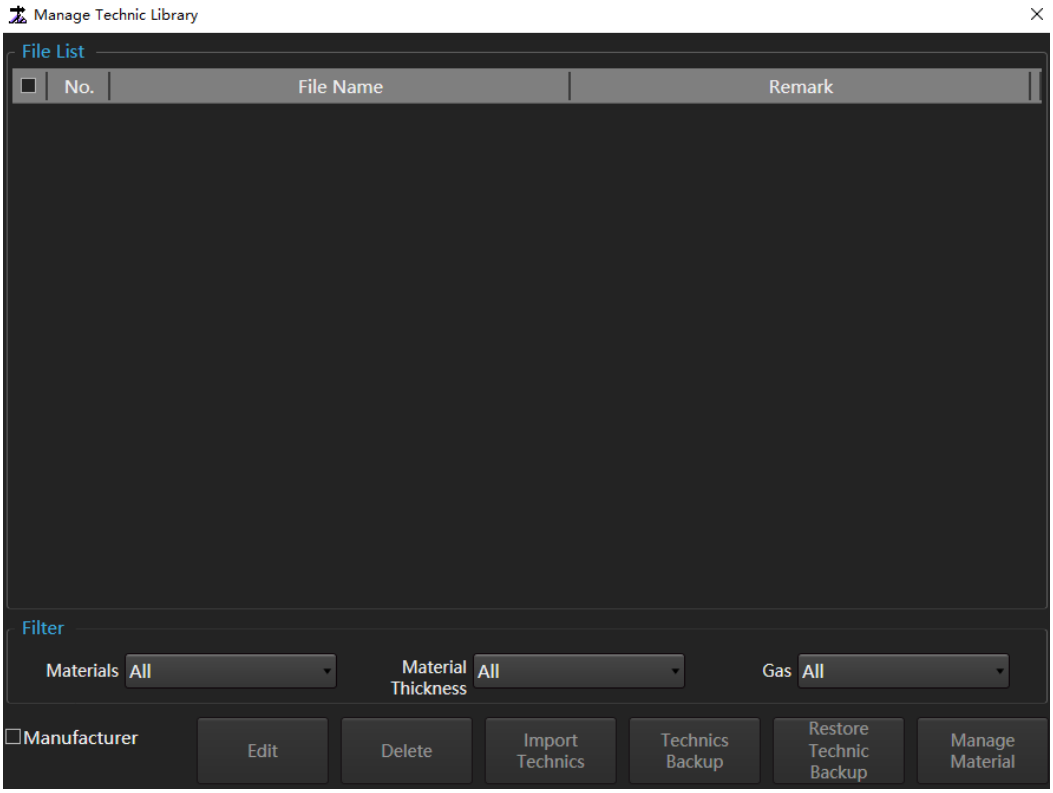
5.9.1.5. Manage Material

You can add or delete material types of tubs.

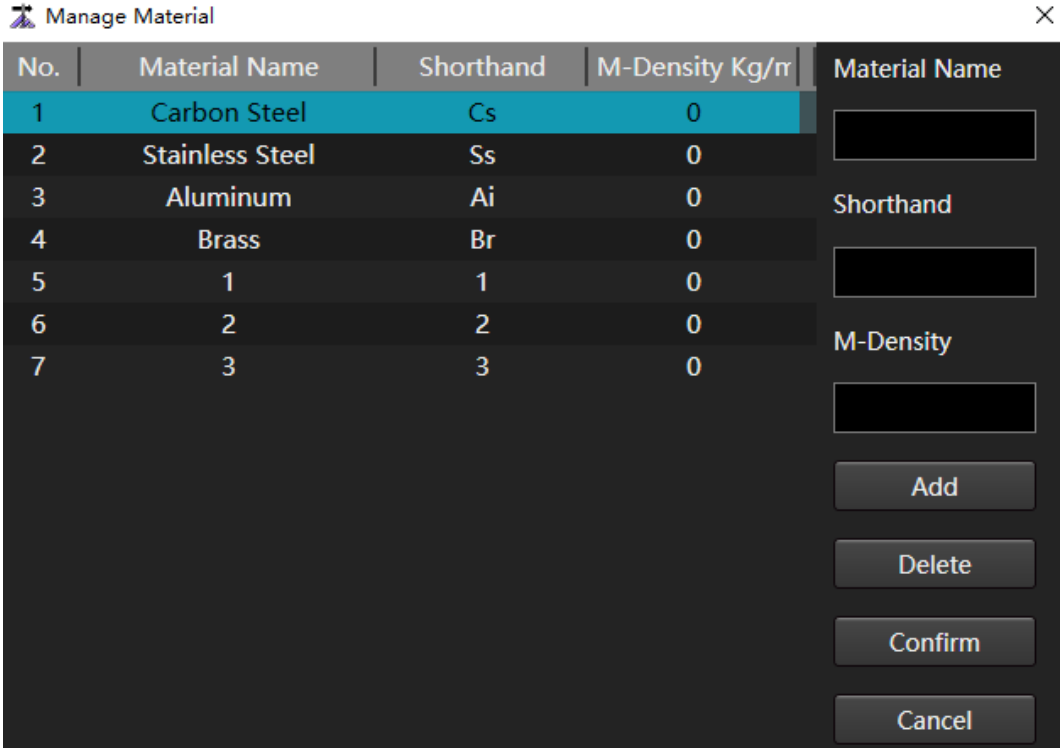
Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Manage Technic Library** dialog box:



2. Check **Manufacturer** to activate the button.
3. Click **Manage Material** to open the **Manage Material** dialog box:



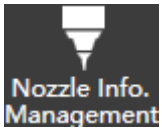
4. Enter the corresponding information in the **Material Name**, **Shorthand**, **M-Density** input boxes.
5. Click **Add**, and the material information will be displayed in the left list.
6. **（Optional：）** To delete a material, click the target material to highlight it, and then click **Delete**.
7. Click **Confirm** to close the **Manage Material** dialog box.

5.9.2. Nozzle Info. Management

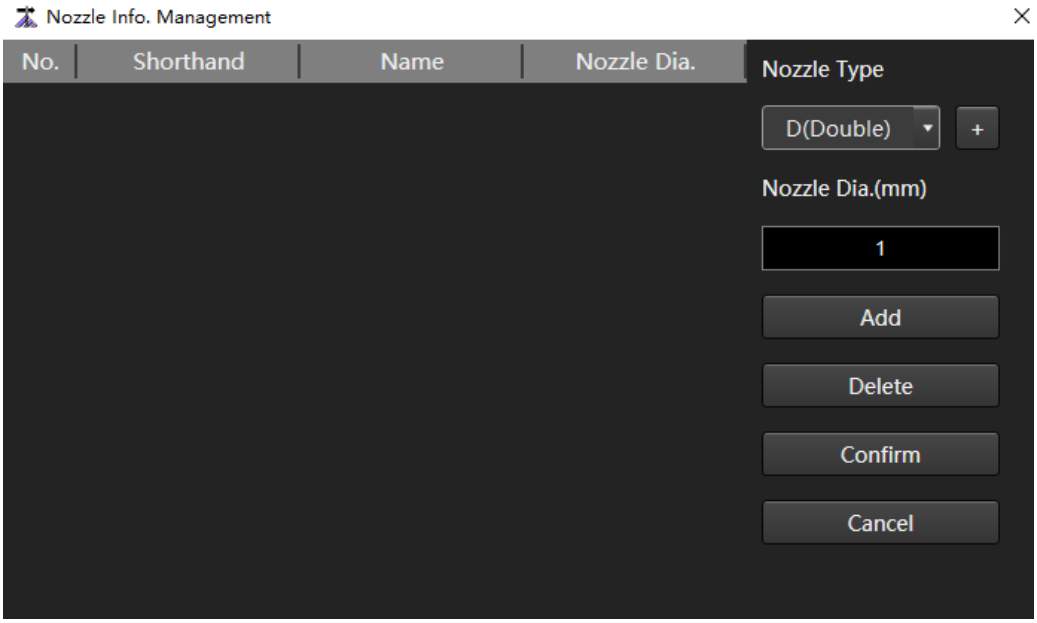
You can view, add, and delete nozzle information. After the nozzle information is updated, the corresponding nozzle type will be added in Layer Technic.

Operating Steps：

- View nozzle information

1. In the menu bar, click **Machining** → 

to open the **Nozzle Info. Management** dialog box:



2. Select the nozzle type you want to view from the drop-down key in the **Nozzle Type** area, and it will be displayed after filtering according to the nozzle type on the left.

- Add nozzle

1. Select the nozzle type to add from the drop-down key in the **Nozzle Type** area.

   If there is no suitable nozzle type, you need to add a new nozzle type: click 

, in the pop-up **Nozzle Information** dialog box, fill in the nozzle shorthand and name, and click **Confirm**.
2. Fill in the information of **Nozzle Dia.**.
3. Click **Add**.

- Delete nozzle information

1. Select the nozzle information to highlight it.
2. Click **Delete**.

- Close the **Nozzle Info. Management** dialog box

1. Click **Confirm**.

5.9.3. Chuck Technic

The technic library is established, according to the tube shape, tube material, thickness, and diameter (or width and height) of the held tube, the torque and pressure of the front/back chuck, and the clamping/unclamping time of the chuck, so as to achieve the precise torque holding control of the chuck over tubes according to the technic library.

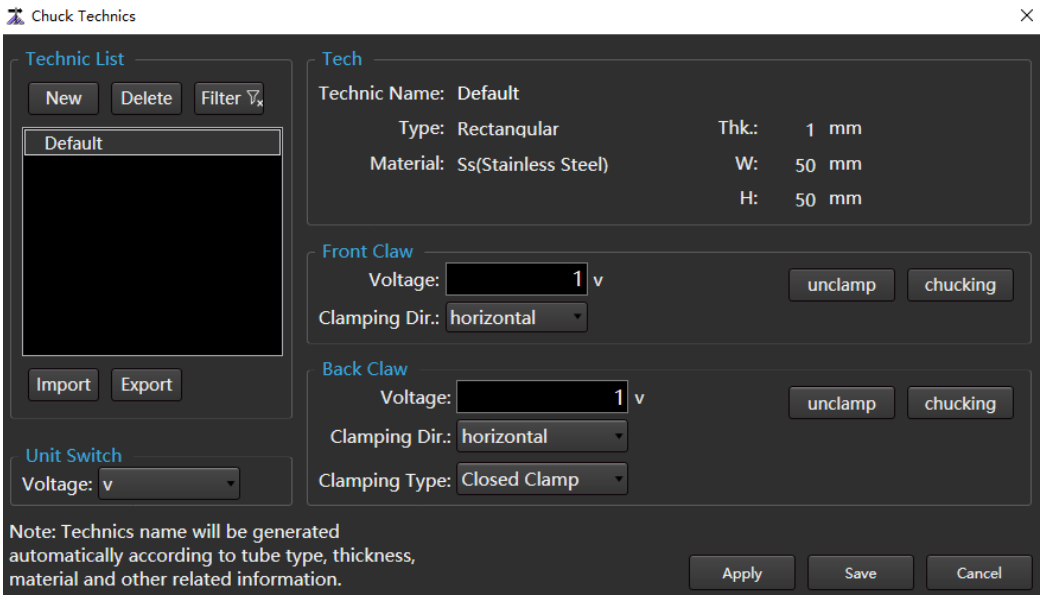
Tubes with the same shape, material, thickness and diameter shares the same torque holding solution.

5.9.3.1. New Chuck Technic

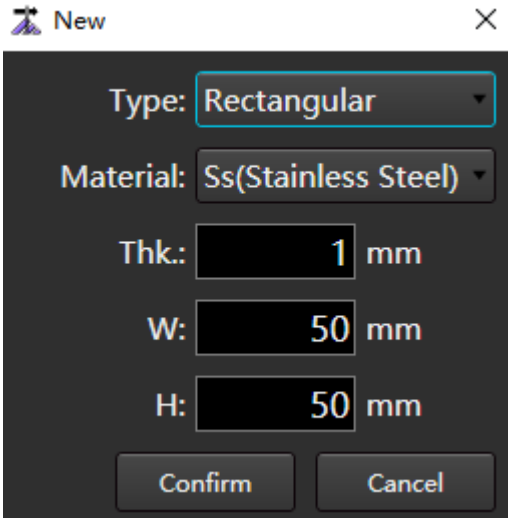
Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Chuck Technics** dialog box:



2. Click **New** to open the **New** dialog box:



3. Set the type, material, thickness, and diameter of the tube, or width and height.
4. Click **Confirm** to add the chuck technic to the **Technic List**.

Note

The technic name will be automatically generated according to the set tube type, material, thickness and other related information.

If you need to rename the technic name, select the technic in the **Technic List**, right click to select the **Rename**, modify the name, and click **Confirm**.

Related Tasks：

For the **Technic List** area, you can also perform the following operations：

- Delete： Select one of the following methods to delete the chuck technic.
  - Select a process in the **Technic List** and right click to select **Delete**.
  - Select a technic in the **Technic List** and click **Delete**.
- Filter：
  a. In the **Technic List**, click **Filter** to open the **Filter** dialog box.
  b. Set the type, material, thickness, and diameter of the tube, or width and height.
  c. Click **Filter**.
- Import：
  a. In the **Technic List**, click **Import** to open the **Open** dialog box.
  b. Select the imported file. The file format is .ctp.
  c. Click **Open** to pop up *Imported parameters will overwrite current parameters. Continue or not?*.
  d. Click **Yes**.
- Export：
  In the **Technic List**, click **Export** to open the **Save As** dialog box.
  Select the path to save.
  Click **Save**.

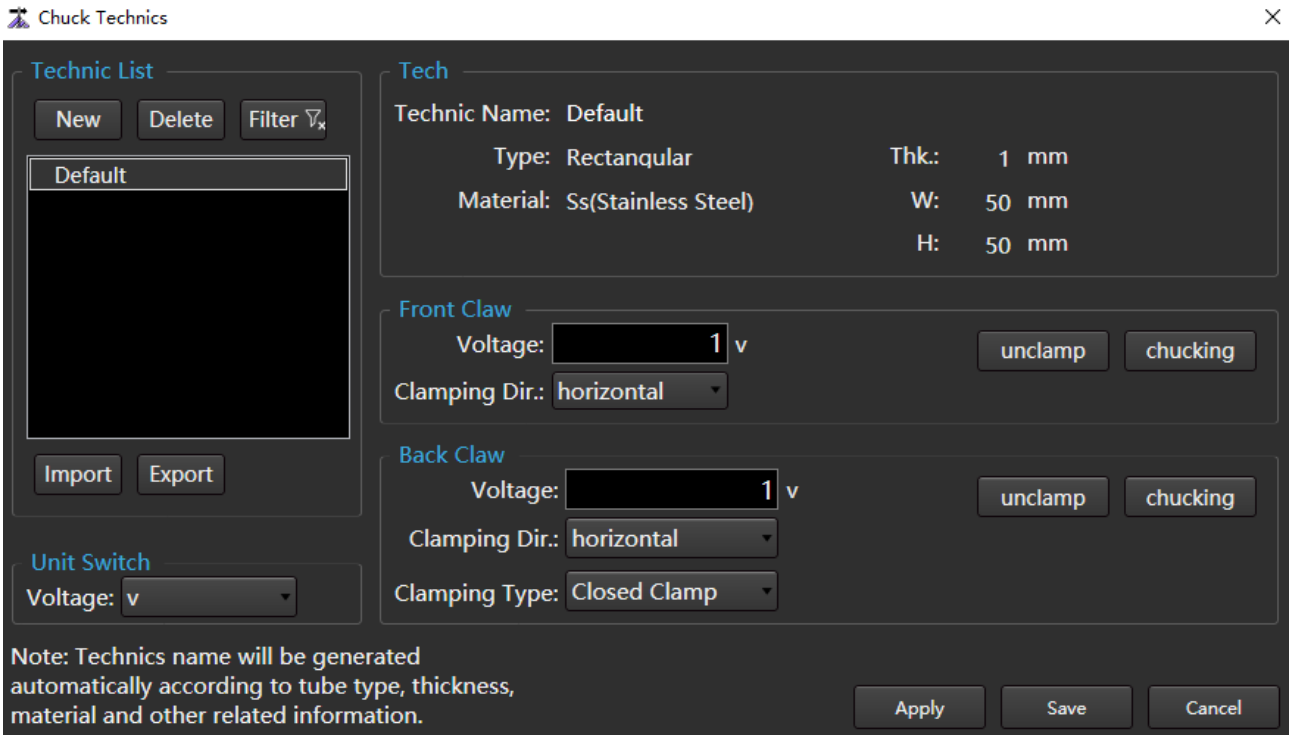
5.9.3.2. Apply Chuck Technic

Set the chuck technic information as the default chuck technic.

Operating Steps：

1. In the menu bar, click **Machining** → 

to open the **Chuck Technics** dialog box：



2. Select the clamped tube technic in the **Technic List** area, and then view the information in the **Tech** area.
3. Select the unit by pressing the **Voltage** drop-down button in the **Unit Switch** area.
4. In the **Front Claw** area, debug the front claw.
    a. Input voltage.
    b. Select the clamping direction.
    c. Click **Unclamp/Chucking** to debug the front claw.
    d. If the debugging is not satisfactory, repeat the above actions and continue debugging.
5. In the **Back Claw** area, debug the back claw.
    a. Input voltage.
    b. Select the clamping direction.
    c. Select the clamping type.
    d. Click **Unclamp/Chucking** to debug the back claw.
    e. If the debugging is not satisfactory, repeat the above actions and continue debugging.
6. Click **Apply** and *The default chuck technic is set successfully* will pop up.
7. Click **Confirm**.
8. Click **Save**.

5.10. Machining Mode

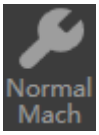
5.10.1. Normal Machining

The formal machining of a single document controls the start of machining. The system is in normal machining mode by default.

Operating Premise：

- Make sure that the machining file has been saved.
- Ensure no emergency stop and alarm.

Operating Steps：

1. **（Optional：）** If it is in the **Batch** status, click **Machining** in the menu bar → 

to switch to the **Normal Mach** status.
2. In the machine tool control bar, click 

.
The system automatically starts machining from the first line of the machining file.
3. After start machining, the following operations can be performed：
   - Stop machining： In the machine tool control bar, click 

, make the machine stop machining and terminate the entire machining task, and the system enters the **Idle** state.
   - Resume： In the machine tool control bar, click 

to enable the system to automatically control the machine tool to continue machining from the position where the last machining stopped.

     Before resuming, make sure that the mechanical coordinates are accurate. If not, return to the mechanical origin first.

5.10.2. Batch Machining

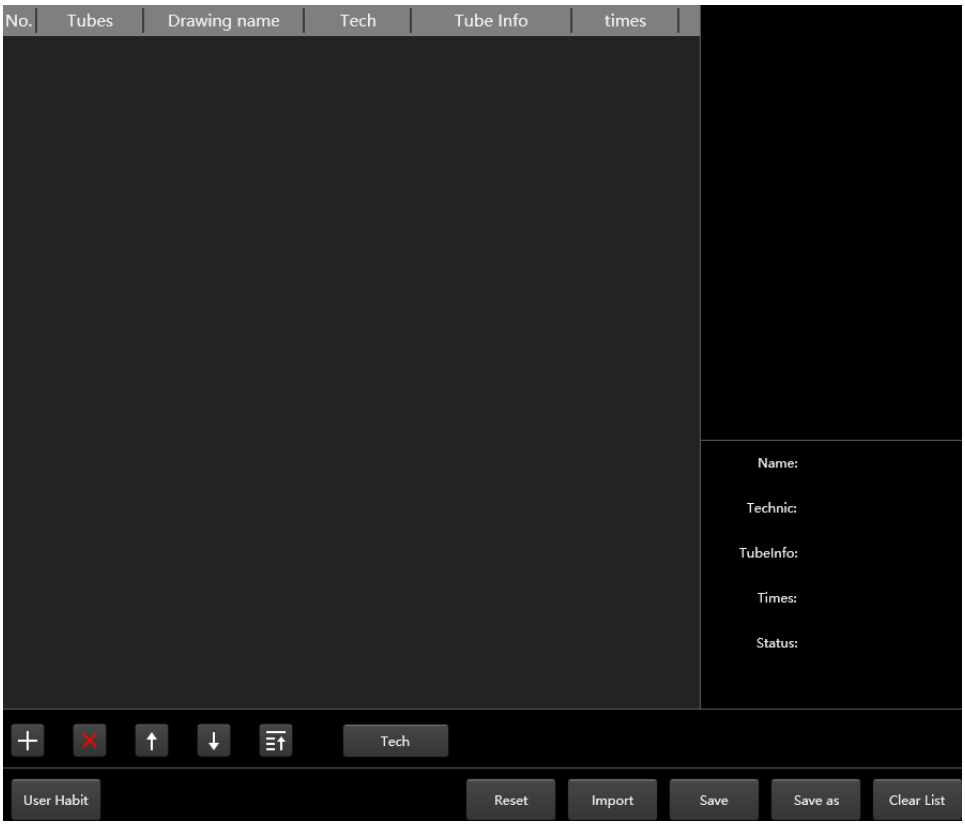
By adding drawings with technic parameters, set the number of machining times for each drawing to form a machining queue. In the technic of machining, each drawing will be automatically loaded after machining, realizing automatic batch machining.

The drawing format support is .ncex， .ncexa， .nc.

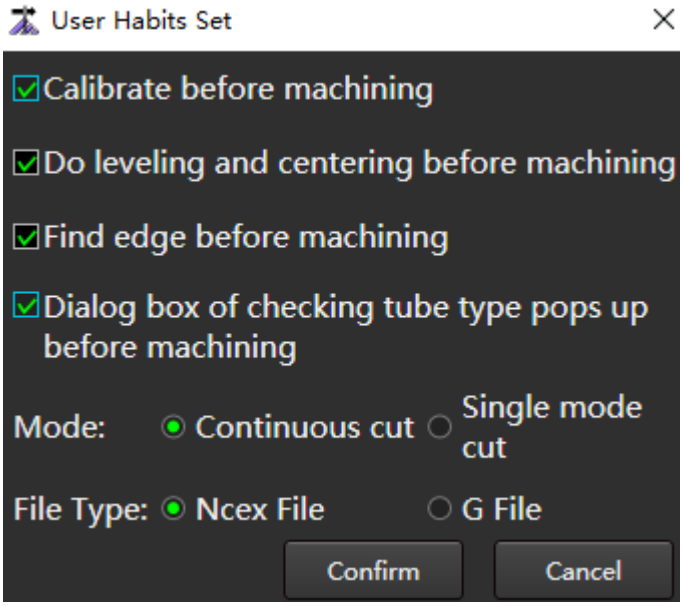
Operation Steps:

1. In the menu bar, click **Machining**→ 

. The drawing area of the main interface of the software becomes the **Batch Machining** interface：



2. Set **User Habit**.
   a. Click **User Habit** in the lower left corner to open the **User Habits Set** dialog box:



   a. Check common operations before machining:
      - Calibration before machining
      - Do leveling and centering before machining
      - Find edge before machining
      - Dialog box of checking tube type pops up before machining
   b. Check **Mode**:
      - Continuous cut: The system starts batch machining according to the sequence.

- Single mode cut: After machining the technic drawing once, the machining stops. To continue machining, click Start.

c. Check the imported drawing file type.
   - Ncex film: .ncex, .ncexa format.
   - G film: .nc format.

Note

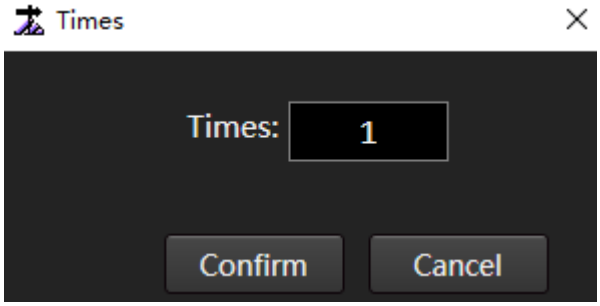
When adding drawings, different file formats can be added according to the selected file types.

d. Click **Confirm**.

3. Select the following method to add a drawing.
   - Click + to select the drawing in the specified path and add it to the machining queue.
   - In the machining queue, right click to select **Add File**, select the drawing in the specified path, and add it to the machining queue.
   - Click **Open**, select the machining queue drawing file (.TLD format file) in the specified path, and add the drawing to the machining queue.
4. Select the drawing, and use the following buttons to adjust the order of machining drawings.
   You can also click the right mouse button to select the corresponding shortcut menu.

Button	Description
↑	Move the drawing up.
↓	Move the drawing down.
⤒	Top the drawing.
✕	Delete the drawing.

5. Set the number of drawing machining in turn.
   a. In the processing queue, double-click the **Times** column of the corresponding drawing to open the **Times** dialog box:



   b. Set the number of machining.
   c. Click **Confirm**.
6. Set the layer technic in turn.

a. Select a drawing, double-click **Tech**, and the **Layer Setting** dialog box pops up.
b. Set layer technic. For details, see Set Layer Technic.

7. Click **Save**. Save the current processing queue in the file name of TaskListInfo.tld to the software installation directory \Tocs\MultiTask.

Note

This file is unique. Saving it again will overwrite and replace the last queue file.

8. In the machine tool control bar, click Start, the system starts machining from the first drawing in the machining queue.

Related Tasks：

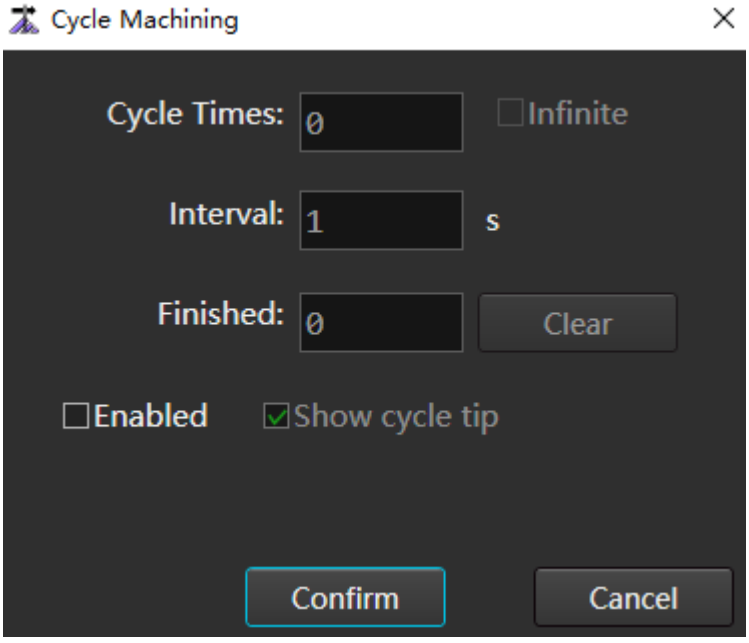
- Click **Reset** to clear all the status in the current machining queue and reset it to the wait machining status.
- Click **Save As** to export the file of the current machining queue to the local.
- Click **Clear List** to delete all drawings in the current queue.

5.10.3. Cycle Machining

Set the number and interval of cycle machining, and view the current number of machined times. Cycle machining is applicable to formal machining and dry run.

Operating Steps：

1. In the menu bar, click **Machining** → Cycle Machining to open the **Cycle Machining** dialog box:



2. Check **Enabled** to set **Cycle Times** and **Interval**. Check **Infinite** to set the time of cycles to infinite.

3. Check **Show Cycle Tip** as required, that is, when starting cycle machining, a prompt box will pop up, prompting the current number of cycles and the total number of cycles, and whether to continue machining.
4. Click **Confirm**.
5. In the machine control bar, click 

, The system automatically counts according to the rules set above:
   - If the machining is suspended or stopped before the set number of cycles is reached, it indicates the current actual number of cycles.
   - The program completely executes one cycle at a time.
6. In the **Finished** parameter of the **Cycle Machining** dialog box, view the current completion times.
7. After the set number of times of machining, if you want to reset the number of times of completion, click **Clear**.

5.11. Machining Report

It is divided into production report and operation report. The generated report can be exported to a file in PDF format or printed directly.

5.11.1. View Production Report

Before machining, the statistical information or machining is charged. The number of perforations, machining length, machining time and machining cost can be estimated before actual machining. At the same time, it supports the setting and display of vendor name, client name, task number, vendor logo, QR code and other information.

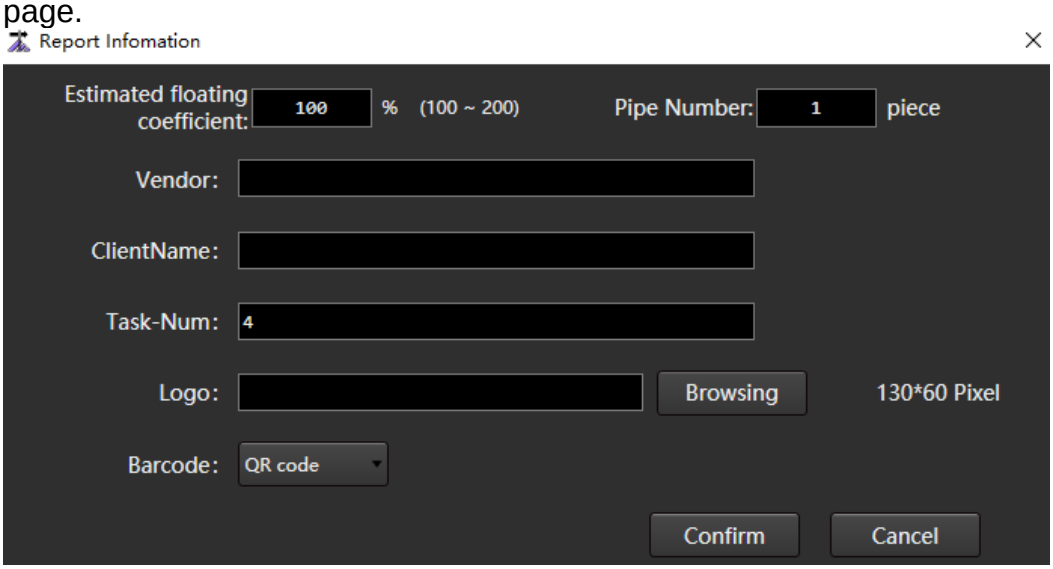
Operating Premise:

Ensure that the target tool path file has been machined or simulated at least once.

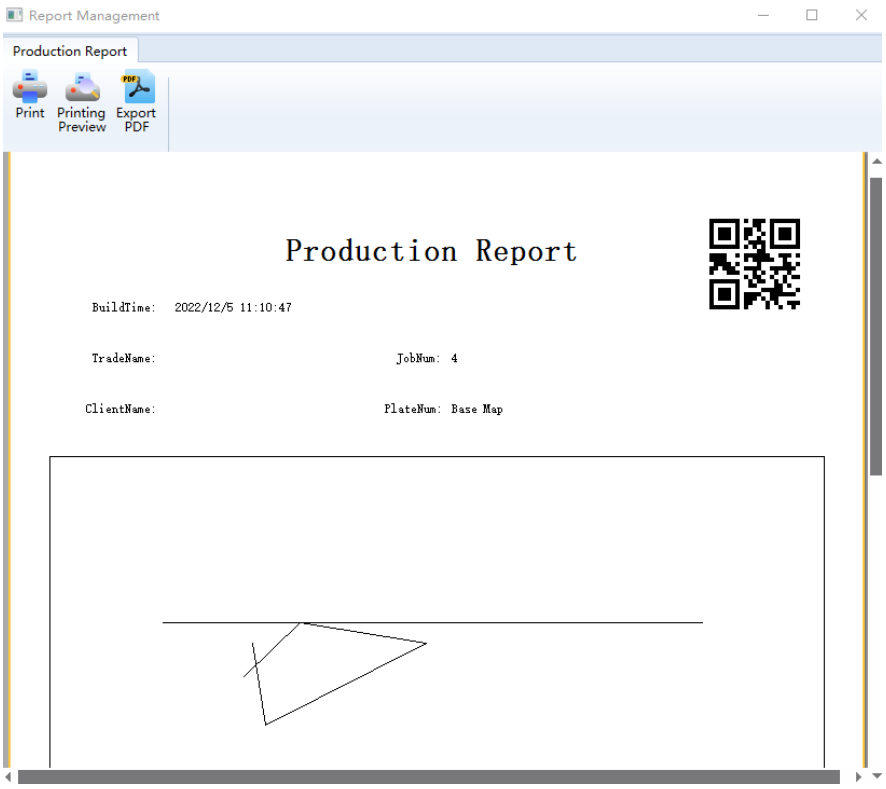
Operating Steps:

1. In the menu bar, click **Machining** → 

to open the **Report Information** page.



2. According to the actual material information and display information, click **Confirm** to generate **Production Report**.



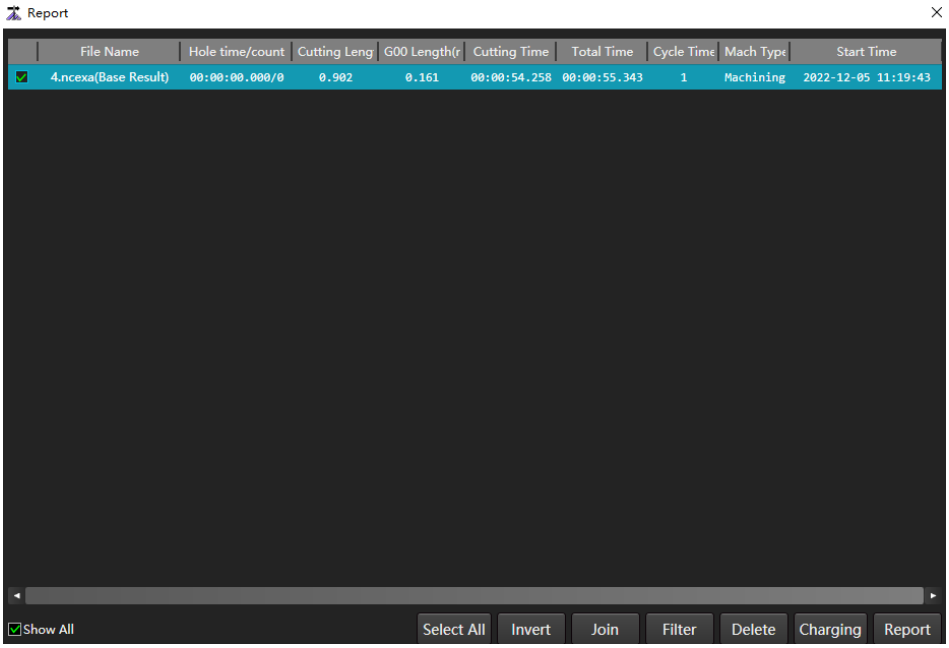
5.11.2. View Run Report

After the target files (single or multiple) are machined, check the count of hole, hole time, cutting length, G00 length, cutting time, total time, and cycle time. At the same time, it supports the setting and display of vendor name, client name, task number, vendor logo, QR code and other information.

Operating Steps：

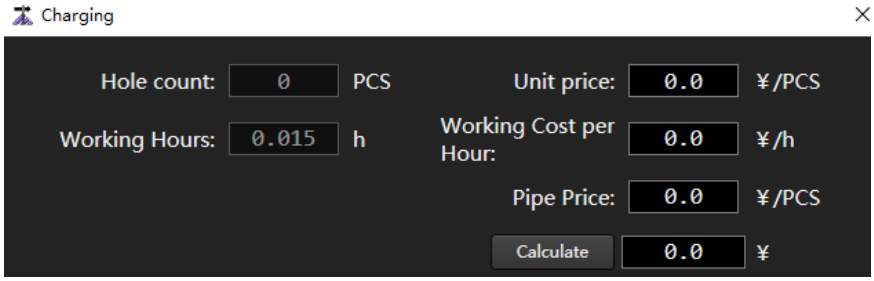
1. Select any of the following methods to open the **Report** dialog box:
   - In the report bar, click **Report**.
   - In the menu bar, click **Machining** → 

.

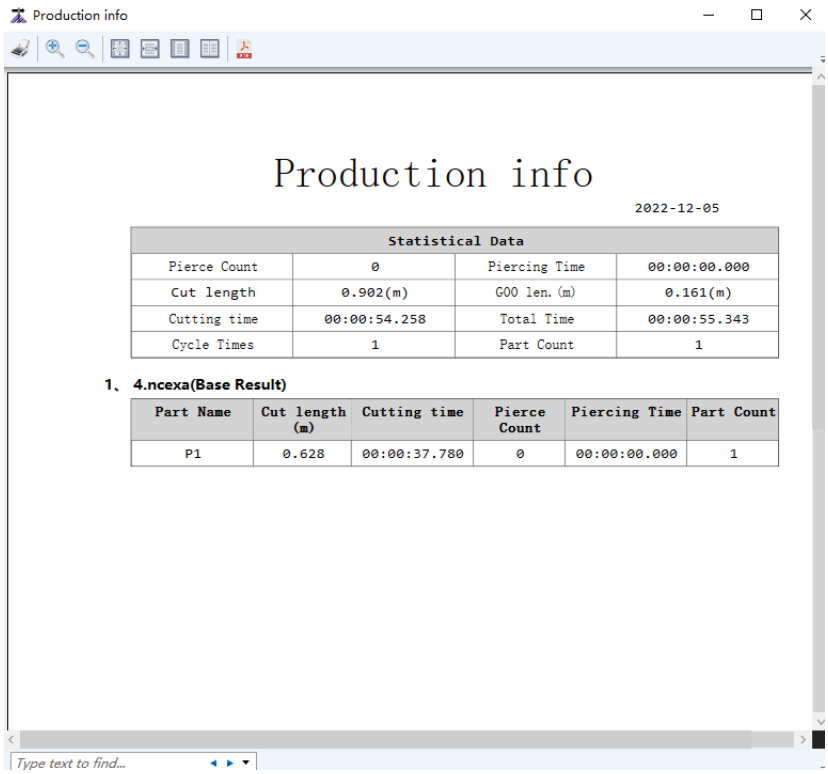


2. View statistics.
3. As needed, do the following:

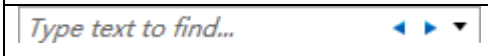
- Charging
  a. After checking the machining item to be charged, click **Charging** to open the **Charging** dialog box:



  b. Enter each unit price, click **Calculate**, and the system will automatically calculate the total cutting cost.
- Output report
  a. Check the target machining item and click **Report** to generate a report of each statistical information:



b. Click the corresponding button as required. The button description is as follows:

Button	Description
	Click this button to print the report.
	Click this button to zoom in the report.
	Click this button to zoom out the report.
	Click this button to display the report in 100% proportion.
	Click this button to display the report width according to the page width.
	Click this button, and the report will be completely displayed in the page according to the whole page.
	Click this button, and the report will be completely displayed in the page according to two pages.
	Click this button to save the report locally in the format of .pdf.
	In this input box, enter keywords to find text.

5.11.3. View Log

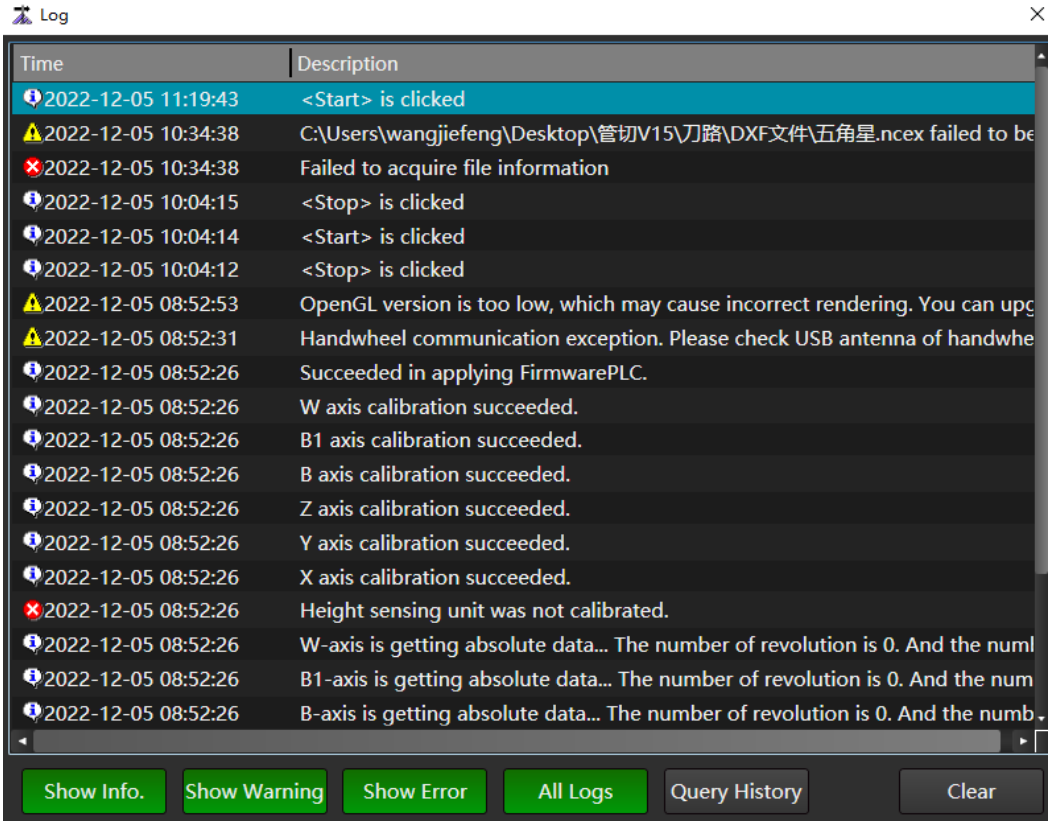
The log records important user operations, system events and time, including information and historical information after the system is started.

When the log file is greater than 20M, a backup log file （NcStudio_xxxx.log） is automatically generated in the **User** folder (path), and the original log content is cleared.

Operating Steps：

1. Select any of the following methods to open the **Log** dialog box and view the log:
   - In the operation information bar, double-click **Alarm** / **Log**.
   - In the menu bar, click **Machining** → 

.



2. Select the log type to view:
   - Turn on the **Show Info.** button to display software operation information with the icon .
   - Turn on the **Show Warning** button to display the warning information with the icon .
   - Turn on the **Show Error** button to display the error information with the icon .
   - Turn on the **All Logs** button to display all the corresponding log information since the system was started.

   All buttons are on by default.
3. To view more log information, click **Query History** and select the viewing date. You can view log information within 1 year at most.
4. To delete all log information, click the **Clear** button.

Caution

Please clear the system log regularly! Otherwise, when the system log file is too large, the system performance and response time will be affected.

6. Set Parameter

6.1. Parameter

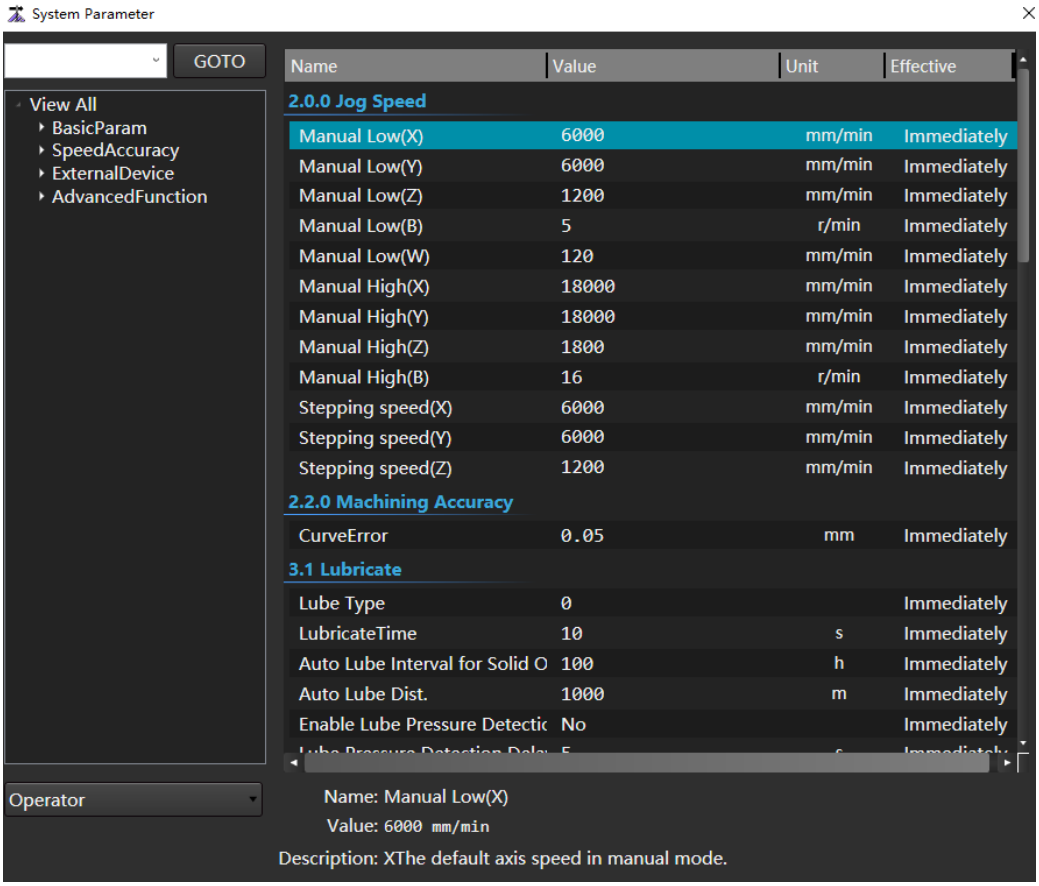
6.1.1. System Parameter

This section describes how to find and modify system parameters by taking the modification of parameter **Fixed Point Position (X)** as an example.

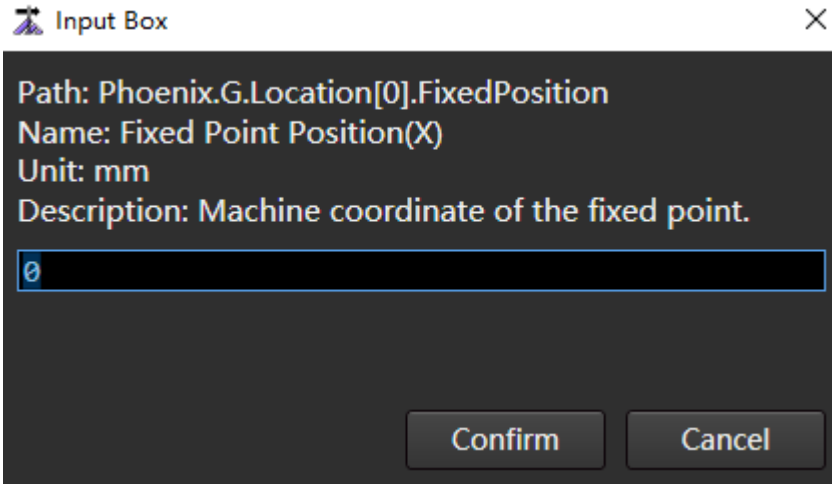
Operating Steps：

1. In the menu bar, click **Set** → 

to open the **System Parameter** dialog box:



2. Use any of the following methods to find the **Fixed Point Position (X)**:
   - Search: In the search box, enter **Fixed Point Position (X)**, click **Go To**, and the search results will be displayed on the right.
   - Find through the node tree on the left: In the node tree, click **View All** → **Advanced Function** → **4.1 Fixed Point**, and the parameters in the **4.1 Fixed Point** node will be displayed on the right. Find the **Fixed Point Position (X)**.
3. Double click the parameter **Fixed Point Position (X)** to pop up the **Input Box**:



4. Enter value.
5. Click **Confirm**.

6.1.2. Port Setting

Monitor the condition of the machine tool by controlling the input and output ports, including conducting simulation tests, modifying port polarity, and setting port attributes.

General port settings are used to detect whether each port is effective during debugging, and change the port polarity according to actual needs.

The relationship between the machine tool condition and the input and output ports is as follows:

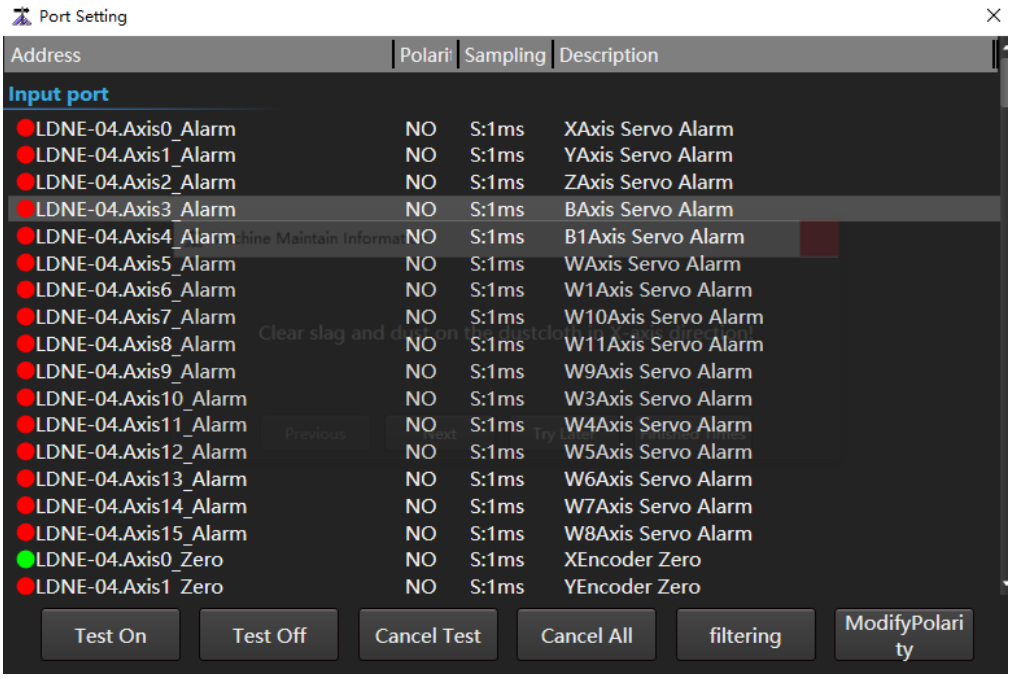
- Input port: With single; No single.
- Output port: With single; No single.

When the port is in the test state, there is a T in the lower left corner of the signal, such as:

Operating Steps:

1. In the menu bar, click **Set** → 

to open the **Port Setting** dialog box:



2. Select the port, and select the following operations according to the actual situation:

Operation	Description
Click **Test On** or **Test Off**	Carry out simulation test, simulate opening or closing the port, and judge whether there is output by testing the port signal. The mark T in front of the port indicates that the port is testing.
Click **Cancel Test**	Cancel the test of the port.
Click **Cancel All**	Cancel the test of all ports.
Click **Filtering**	Set the filtering duration, and the system will exclude the signals whose occurrence time is less than this duration.
Click **Modify Polarity**	The polarity of the port changes to the opposite polarity.

6.1.3. Lead Screw Compensation

When the machine tool itself has errors and cannot reach the expected accuracy, compensate the lead screw error to improve the machining accuracy.

Before using the lead screw compensation, click **Set** in the menu bar → 

, to find and set the manufacturer's parameter **Lead Screw Compensation Mode** as not 0:

- 1: Backlash compensation only
- 2: Backlash compensation and unidirectional compensation
  The compensation is made by printing the one-way data of the laser interferometer and the backlash value (printed by the dial indicator) set in the system parameters. Its operation is consistent with that of using bidirectional compensation.

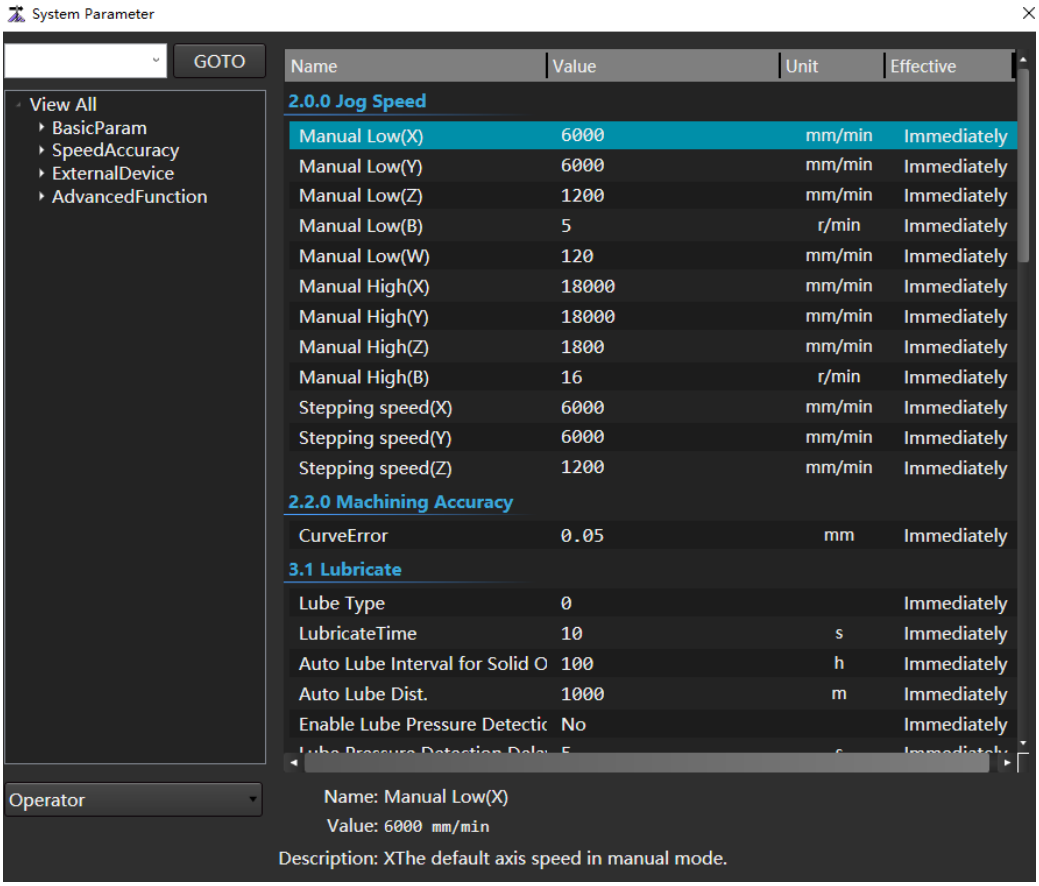
- 3：Bidirectional compensation
  Compensate according to the bidirectional data of laser interferometer.

6.1.3.1. Use Backlash Compensation

Operating Steps：

1. In the menu bar, click **Set** → 

to open the **System Parameter** dialog box:



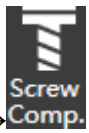
2. Select **Manufacturer** permission.
3. Click **View All** → **Basic Param** → **1.2 Error Compensation** on the right to find the parameter **Lead Screw Compensation Mode**.
4. Double click **Lead Screw Compensation Mode** and enter 1 in the input box.
5. Click **Confirm**.

6.1.3.2. Use Bidirectional Compensation

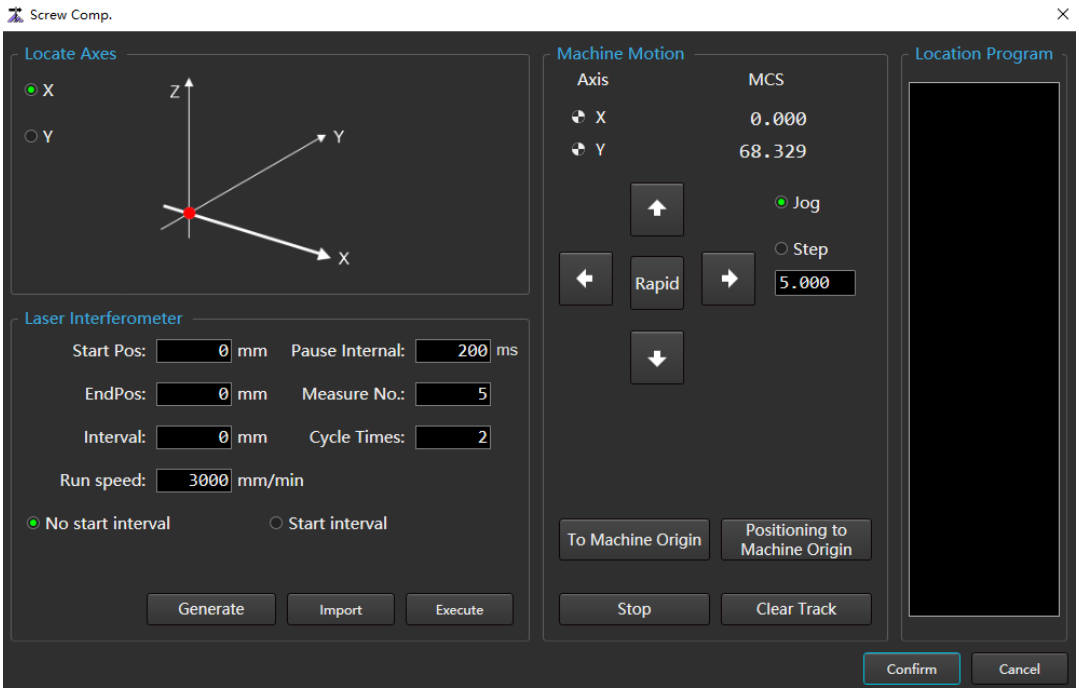
Operating Premise：

The value of parameter **Lead Screw Compensation Mode** is 3.

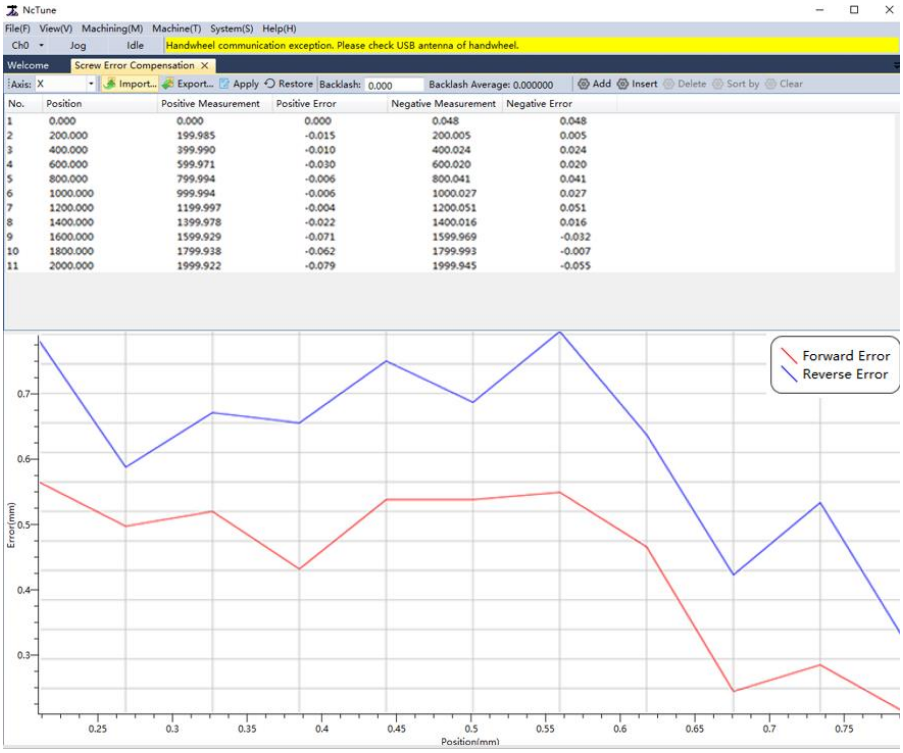
Operating Steps：

1. In the menu bar, click **Set** → 

to open the **Screw Comp.** dialog box:



2. （**Optional：**） If X axis and Y axis need to return to mechanical origin, click **To Mechanical Origin** in the **Machine Motion** area.
3. Follow the steps below to get the actual measurement data of the machine tool：
    a. In the **Locate Axes** area, select the locate axis.
    b. Set the relevant parameters of the positioning program in the **Laser Interferometer** area, and click **Generate**. The results are automatically written into the **Location Program** area.
    c. Click **Execute**, the machine tool starts to move according to the generated positioning program, and records the position data at the measuring point.
    d. On the laser interferometer side, save the recorded position data as a RTL or LIN format lead screw error compensation file.
4. Close the software and double-click **NcTune** under the file installation directory C:\Program Files\Weihong\NcStudio\Bin to enter the **NcTune** software.
5. Click **Screw Error** to enter the **Screw Error Compensation** page.
6. Click **Import** to import the lead screw error compensation file **NcTune** generates curves from files：



Red curve: Positive error; Blue curve: Reverse error.

7. Click **Apply** to automatically save the compensation data to the corresponding configuration file.

Restart the **NcStudio** software, and the compensation takes effect. During the system machining, the lead screw error is automatically compensated according to the compensation data.

6.1.4. Drive Parameter

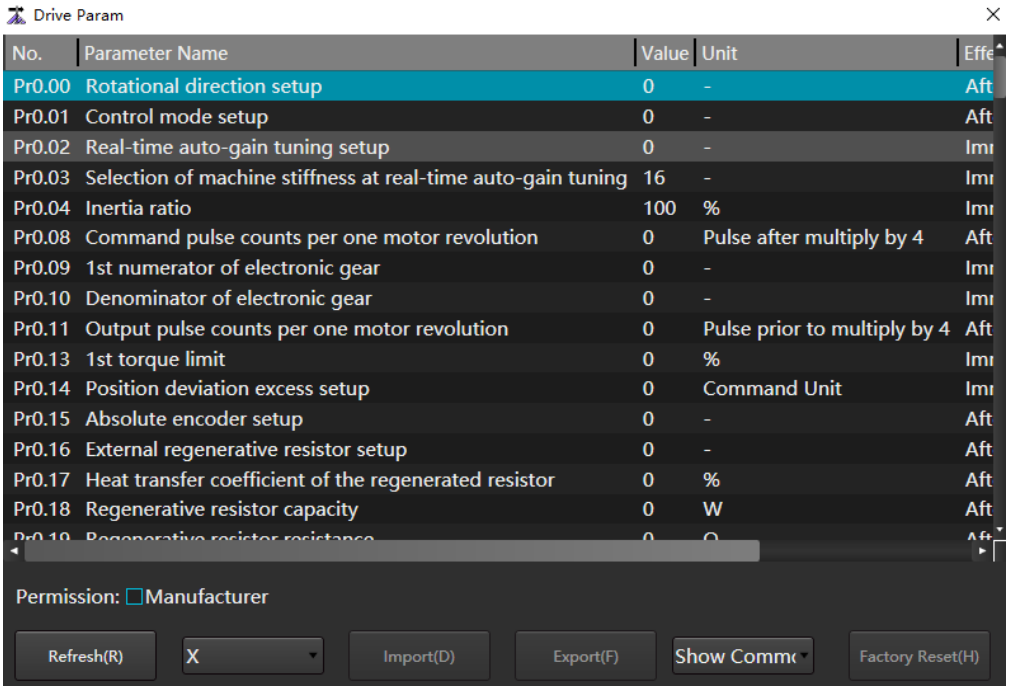
Display the parameter value, unit, effective time and value range of driver parameters according to each axis, and support import, export and restore initial value settings.

Generally, during debugging, it is necessary to set basic driver parameters to drive the machine tool.

Operating Steps:

1. In the menu bar, click **Set** → 

to open the **Drive Param** dialog box:



2. Perform the following as needed.

If…	So…
View drive parameters	1.In the drop-down key of axis, select the axis to view.2. In the **Show All** drop-down button, select **Show All** or **Show Common**.
Import drive parameters	1. Check **Manufacturer** permission.2. Click **Import** to import a file in the format of .dat.
Export drive parameters	1. Check **Manufacturer** permission.2. Click **Export** to save all drive parameters locally in the file format of .dat.
Factory reset	1. Check **Manufacturer** permission.2. Click **Factory Reset** .
Refresh	Click **Refresh**.

6.1.5. Laser Device Setting

The laser device settings are divided into:

- Basic Setting: According to the type of laser driver, set the basic parameters, spot parameters and communication parameters of the laser driver.
- QCW Mode Setting: If the peak power is much greater than the average output power of the laser driver, set the QCW mode to pulse mode.

Note

  - If only basic settings are made, the QCW mode is continuous.
  - When the QCW mode is pulse mode, the cutting parameter speed power and speed frequency curve functions will be shielded.

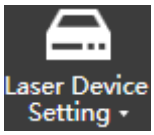
- Adjust Laser AVC: Make the analog target output value of the laser consistent with the actual voltage.

The communication methods supported by each laser driver type include:

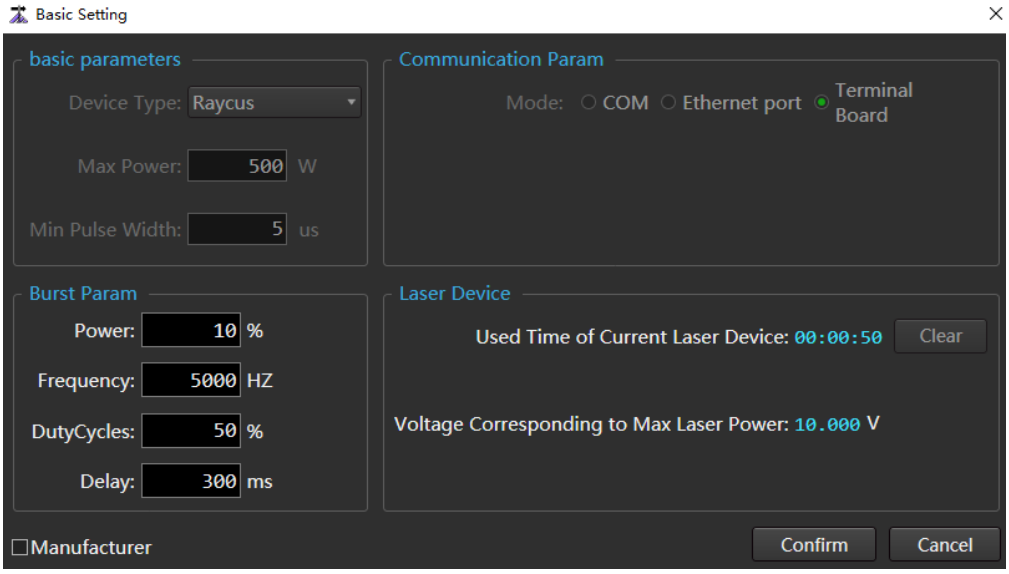
Communication Mode	Raycus	IPG-YLR	YLR-K	Maxphotonics	SPI	IPG(US)	IPG(DE)	SUPER	Feibo	GW	JPT	Trumpf
Serial port	√	√	√	×	√	×	×	×	×	×	×	×
Internet port	×	√	√	×	×	√	√	×	×	×	×	×
Terminal board IO	√	√	√	√	√	√	√	√	√	√	√	√

6.1.5.1. Basic Setting

Operating Steps：

1. In the menu bar, click **Set** → 

→ **Basic Setting** to open the **Basic Setting** dialog box:



2. Check **Manufacturer** to activate the **Basic Parameters** area and **Communication Param** area.
3. In the **Basic Parameters** area, set the following basic parameters:
   - Device type
   - Max power
   - Min pulse width
4. In the **Communication Param** area, select the communication mode.
5. In the **Burst Param** area, set the following click parameters:
   - Power
   - Frequency
   - Duty cycle
   - Delay
6. **（Optional：）** To reset the current laser driver service time, click **Clear** in the **Laser Device** area.
7. Click **Confirm**.

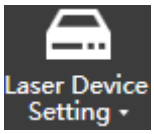
6.1.5.2. QCW Mode Setting

The continuous mode is the default mode. The QCW laser driver can switch the working mode of the laser driver to the pulse mode on this page.

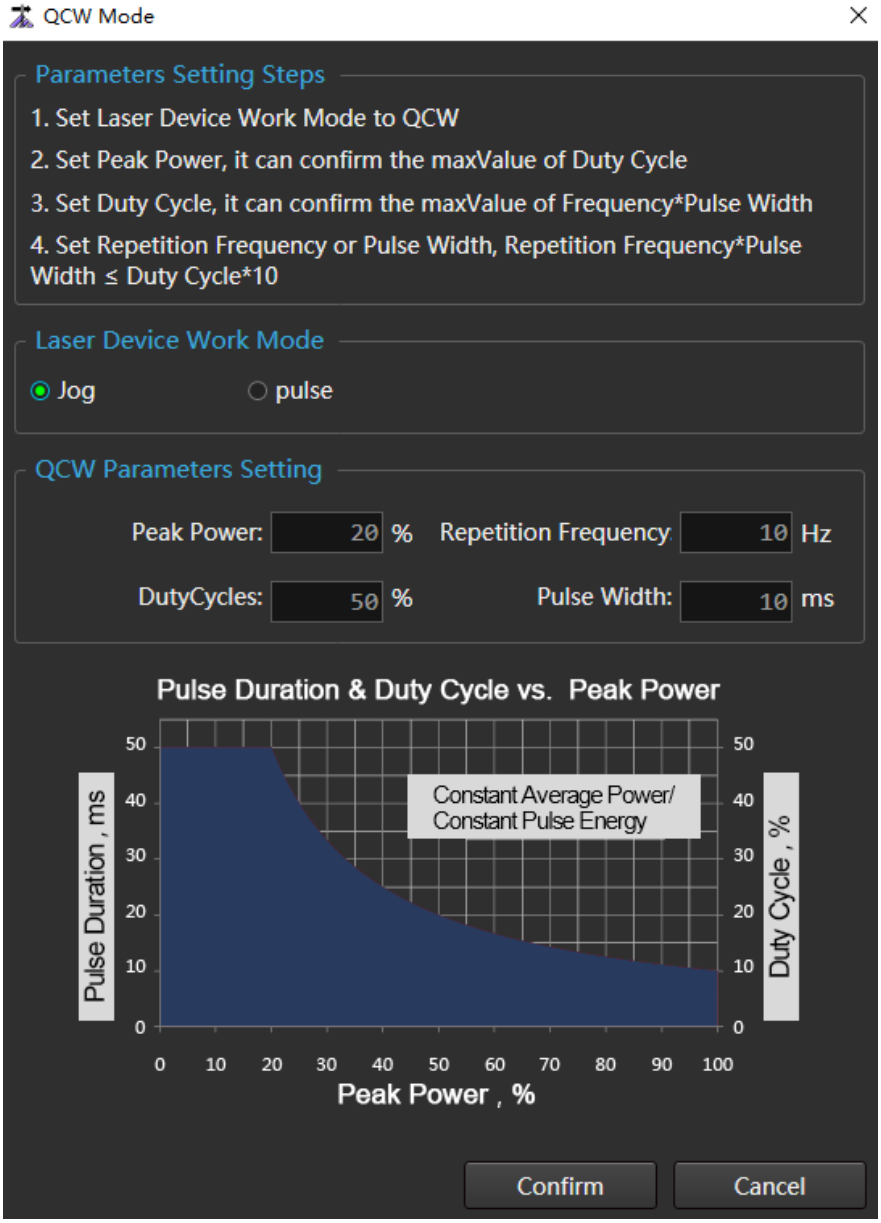
Operating Premise：

Make sure that the relevant parameters have been set on the **Basic Setting** page.

Operating Steps：

1. In the menu bar, click **Set** → 

→ **QCW Mode** to open the **QCW Mode** dialog box:



2. In the **Laser Device Work Mode** area, check **Pulse**.
3. Set parameters in the **QCW Parameters Setting** area:
   - **Peak Power**
   - **Duty Cycle**

- **Repetition Frequency**
- **Pulse Width**

Among:

- Maximum input duty cycle=1000 ÷ peak power
- Repetition frequency × pulse width ≤ duty cycle * 10

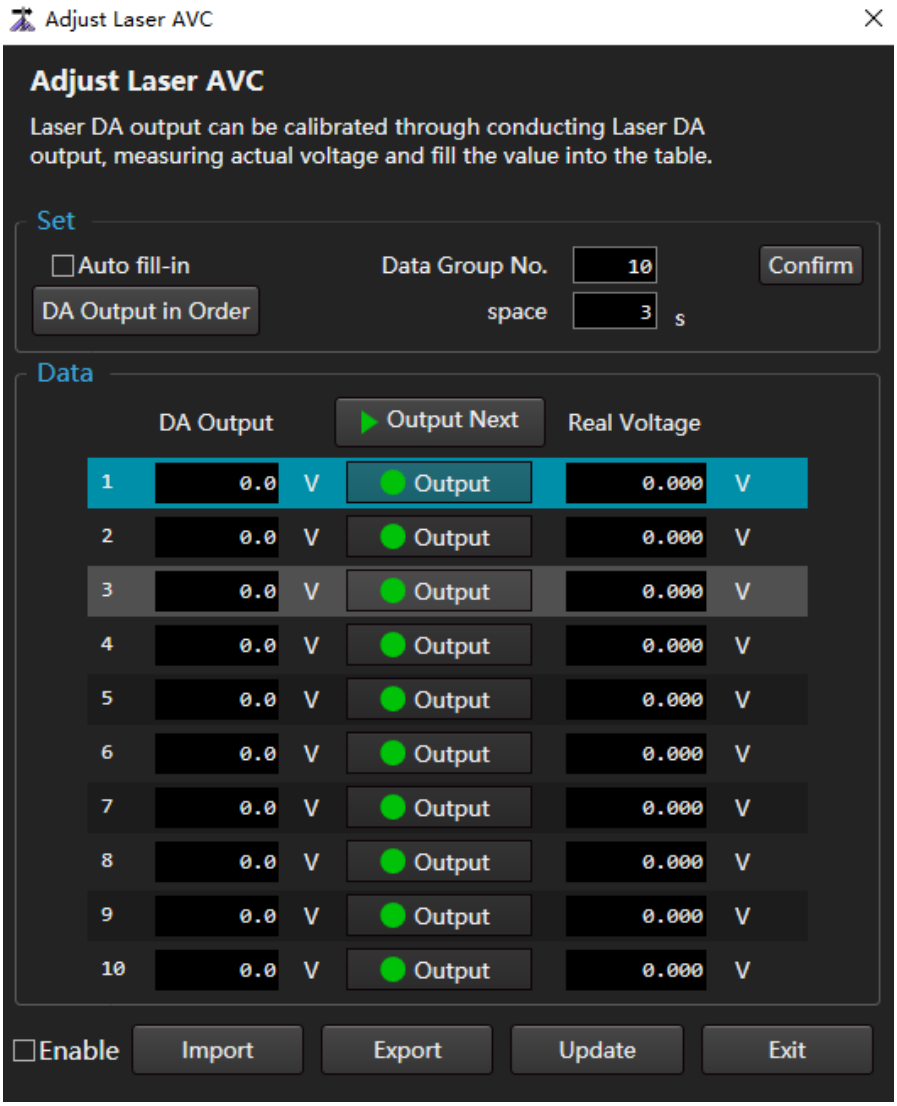
4. Click **Confirm**.

6.1.5.3. Adjust Laser AVC

Operating Steps:

1. In the menu bar, click **Set** → 

→ **Adjust Laser AVC** to open the **Adjust Laser AVC** dialog box:



2. In the **Set** area, set the number of data groups and click **Confirm**.
   By default, the number of data groups is 10. In the **Data** area, there are 0 to 9 rows of data.
3. Perform different operations according to different ways of filling **DA Output** column data.
   - Auto fill-in: Check **Auto Fill-in** and click **Confirm**.

- Fill in manually: In the **DA Output** column, fill in the values in turn.

1.Perform different operations according to the way to fill in the **Real Voltage** column data.

- Automatically output analog quantity according to the set time interval: Click the **Space** input box to input the set value, and click **DA Output in Order** in turn.
- Fill in manually:
  a. In the **Data** area, select the target data and turn on Output.
  b. Fill the actual measured voltage into the corresponding **Real Voltage** column.

1. **Enable**.

- Check: Perform voltage correction. When the DA output data is inconsistent with the actual voltage data, it is recommended to check.
  - Uncheck: No voltage correction is performed.

1. Click **Update**.

Related Tasks:

- Export: Click **Export** to save the current data locally.
- Import: Click **Import** to import the locally saved data into the current **Adjust Laser AVC** dialog box.

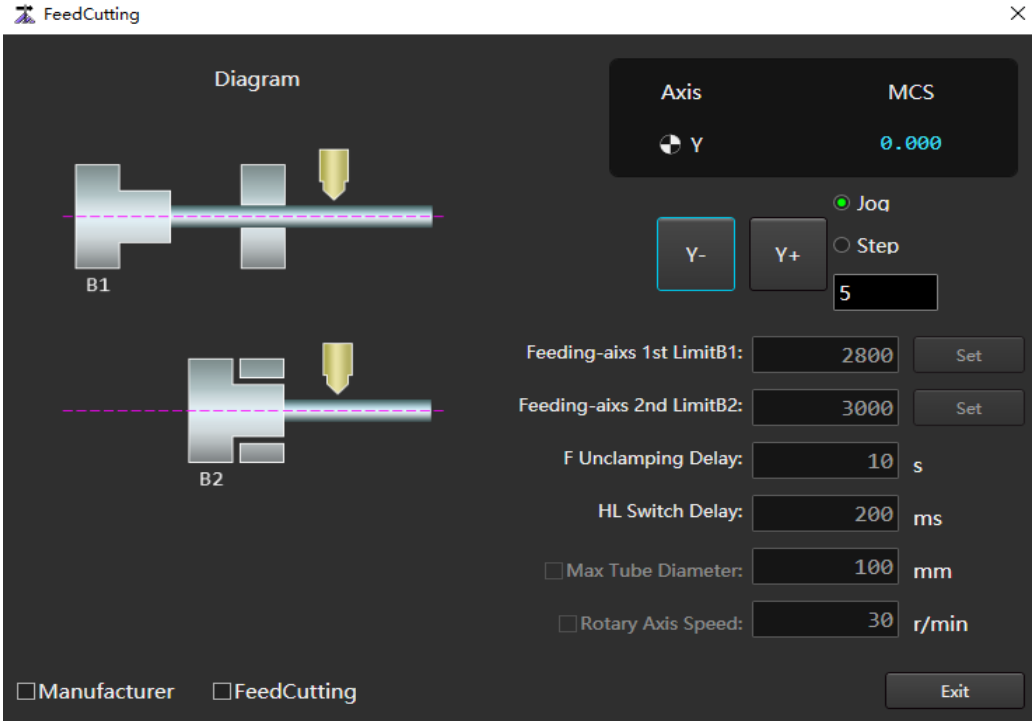
6.2. Advanced

6.2.1. Feed Cutting

When machining to the end of the tube, the front chuck opens, and the rear chuck extends into the inside of the front chuck, so that the tube clamped by the rear chuck can be cut to save the tail length.

Operating Steps:

1. In the menu bar, click **Set** → FeedCutting to open the **Feed Cutting** dialog box:



2. Check **Manufacturer** to activate parameters.
3. Sets the movement mode of the Y axis.
4. According to the different methods of setting parameters **Feeding-axis 1st Limit B1** and **Feeding-axis 2nd Limit B2**, select the following operations:

Method	Step
Manual input	1.Click the corresponding input box to open the **Input Box** dialog box.2. Enter value.3.Click **Confirm**.
Automatic input	1.Set the movement mode of Y axis: jog or step and the distance of one movement.2.Click **Y -/Y+** to move the Y axis to the mechanical coordinate position to be set. 3. Click **Set** corresponding to the parameter to automatically fill the mechanical coordinate of Axis Y into the input box.

5. Enter values in the input boxes of parameters **F Unclamping Delay** and **HL Switch Delay**.
6. **（Optional：）** Check **Maximum Tube Diameter** / **Rotary Axis Speed** as required, and set parameter values in the input box.
7. Check **Feed Cutting** to enable this function.
8. Click **Exit** to complete the setting.

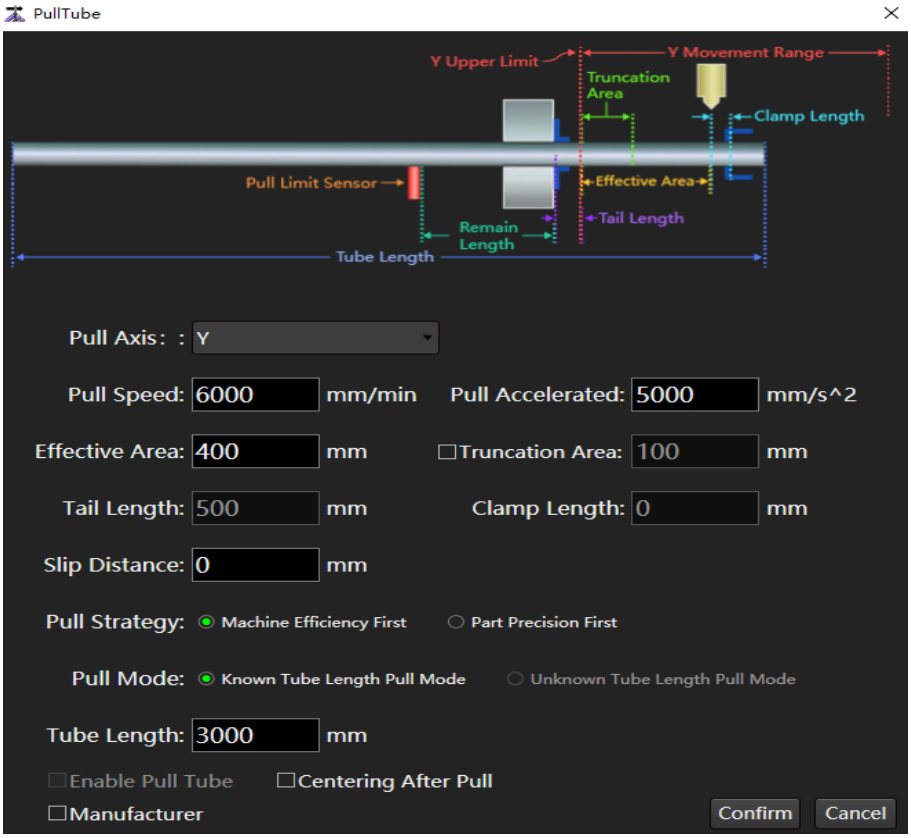
6.2.2. Pull Tube

The pull tube function is applied to the machine tool with short Y axis stroke. Through multiple pull tubes, the purpose of machining long tubes can be achieved

Operating Steps：

1. In the menu bar, click **Set** → 

to open the **Pull Tube** dialog box:



2. Check **Manufacturer**.
   If you do not use the pull tube function after modifying the parameters, uncheck **Enable Pull Tube**.
3. Set parameters. The parameters are described as follows:
   Please refer to the schematic diagram of the **Pull Tube** dialog box.

Parameter	Description
Pull axis	Select different pull axis according to different machine tool designs: ▪ Y axis: The rear chuck pusher, that is, the model with the cutting head stationary and the rear chuck driving the tube movement.▪ Extended axis: The front chuck pull tube machine tool, that is, the machine with cutting head moving and tube not moving.
Pull speed	The speed at which the tube is pulled on the Y axis.
Pull accelerated	The acceleration of tube pulling on the Y axis.
Effective area	▪ When the rear chuck is clamped at one end, the machining accuracy of the front end of the tube is poor, ensuring that all graphics are cut in this area. ▪ The length of a single part to be machined cannot exceed this parameter. If it exceeds this parameter, it cannot be machined.▪ Generally about 100 mm.▪ The actual position is, For example, if the upper limit is 3000, the actual machining position is 2900-3000.
Truncation area	Under the single end clamping state of the rear chuck, the machining accuracy of the front end of the tube is poor, so that all truncation lines are cut in this area.
Tail length	When the Y axis is at the upper limit, the distance from the cutting head to the end of the tube when the tube can be clamped.
Clamp length	It shall be set according to the position of the collet to prevent clamping.▪ The collet extends beyond the left end of the cutting head and is set to 0.▪ The collet is located flush with the cutting head and set to a positive value, generally 10-20 mm.
Slip distance	Supplementary distance given to prevent skidding during pull tube.
Pull strategy	▪ Machining efficiency first: Ensure the machining efficiency, and reduce the number of pull tube as much as possible.▪ Part precision first: Try to ensure that the cutting of a single part is completed after once pull tube.
Pull mode	▪ Know tube length pull mode: This mode is selected when the length of the tube to be cut is known. ▪ Unknown tube length pull mode: This mode is selected when the length of the tube to be cut is unknown. **Note:** This option can be selected only after the port **Pull limit** is configured in the ncConfig application.
Tube length	The length of the tube to be machined. It takes effect when the pull mode is **Know Tube Length Pull Mode**.

1. Check as needed:

- **Enable Pull Tube**: If you do not use the pull tube function after modifying the parameters, uncheck **Enable Pull Tube**.
- **Centering after Pull**.

2. Click **Confirm** to complete the setting.

7. System Maintenance

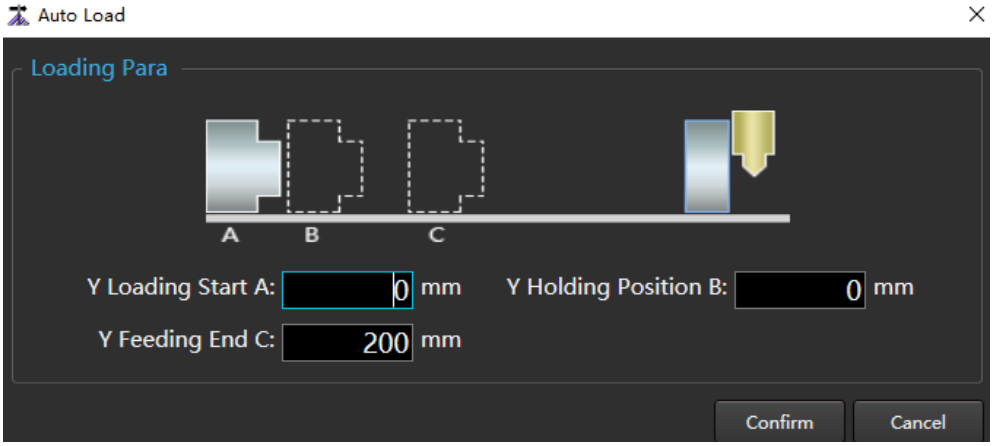
7.1. Automation

7.1.1. Set Auto Load Parameter

When auto loading is executed, the system executes according to the set parameters.

Operating Steps：

1. In the menu bar, click **Maintain** → Auto Load to open the **Auto Load** dialog box:



2. Set the load parameter according to the meaning of the schematic diagram and the actual situation.
3. Click **Confirm**.

Related Tasks：

To enable the automatic loading function, configure it in **NcConfig** → **Configuration** → **Process Edit**.

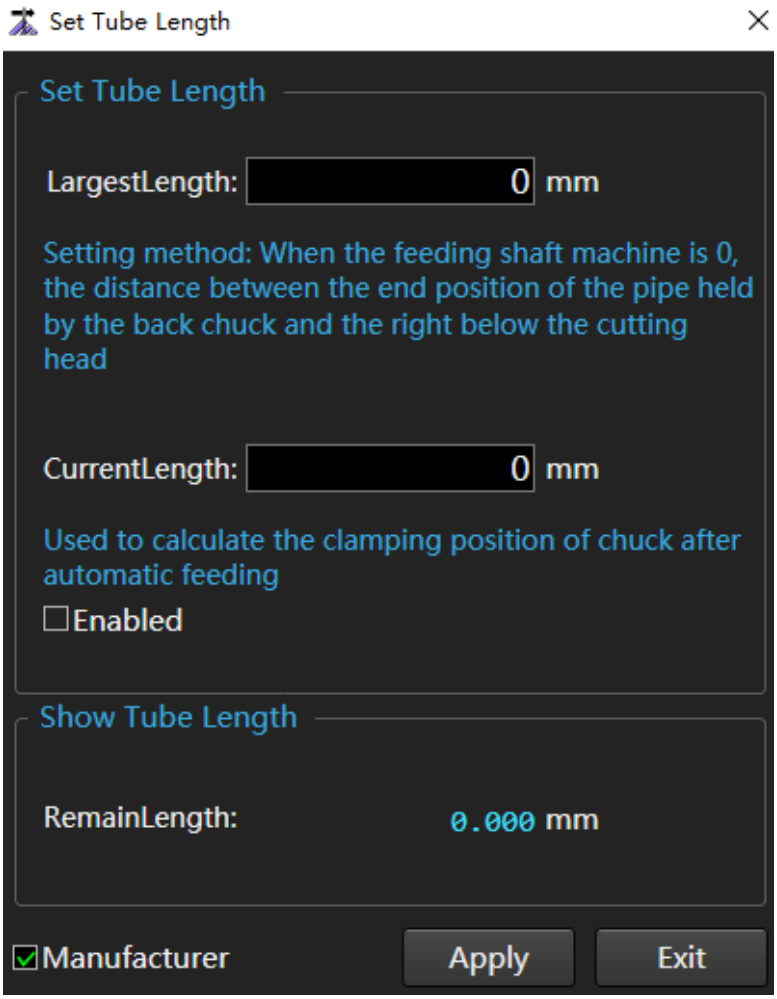
7.1.2. Set Tube Length

Set the current tube length for auto load, calculate the clamping position of the rear chuck, and set the maximum tube length, that is, the distance from the end position of the tube clamped by the rear chuck to the lower part of the cutting head when the pull axis machine is 0.

The tube length setting requires the manufacturer's password.

Operating Steps：

1. In the menu bar, click **Maintain** → Set Tube Length to open the **Set Tube Length** dialog box:



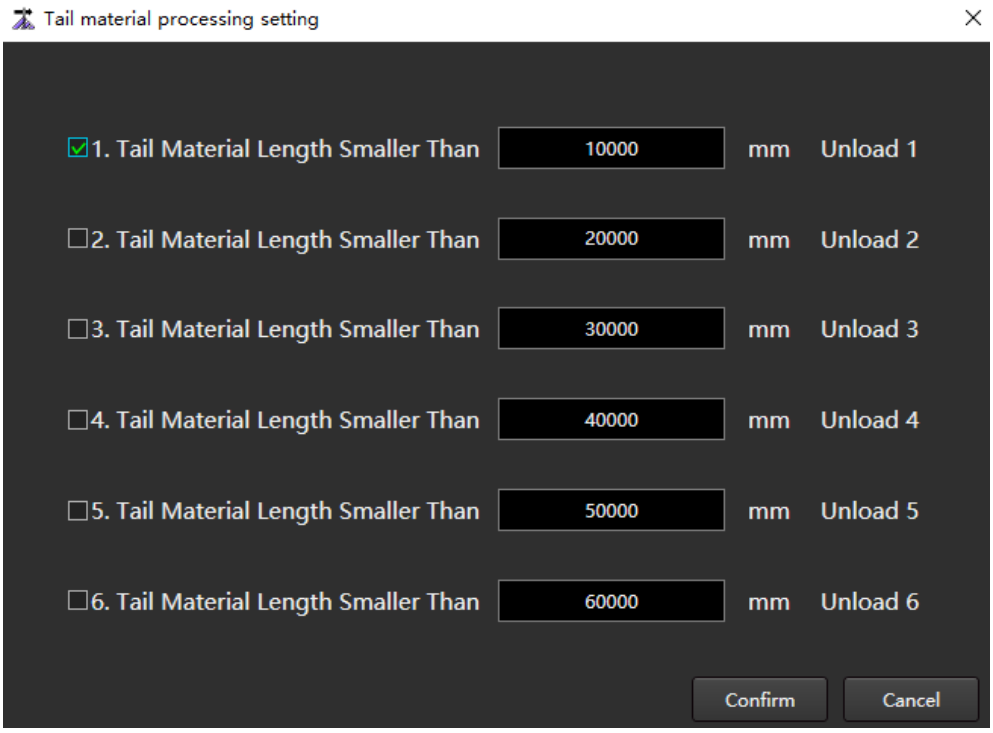
2. Check **Manufacturer**.
3. Set the parameters **Largest Length** and **Current Length**.
4. Check **Enabled**.
5. Click **Apply**.

7.1.3. Tail Material Processing Setting

The system provides 6 unloading actions. The user can set the length of different tailings and execute different unloading actions. Up to 6 tailing lengths can be set.

Operating Steps：

1. In the menu bar, click **Maintain** → Tail material processing setting to open the **Tail Material Processing Setting** dialog box:



2. Set the tailing length.
3. Check the items that need to be enabled.
4. Click **Confirm**.

Related Tasks：

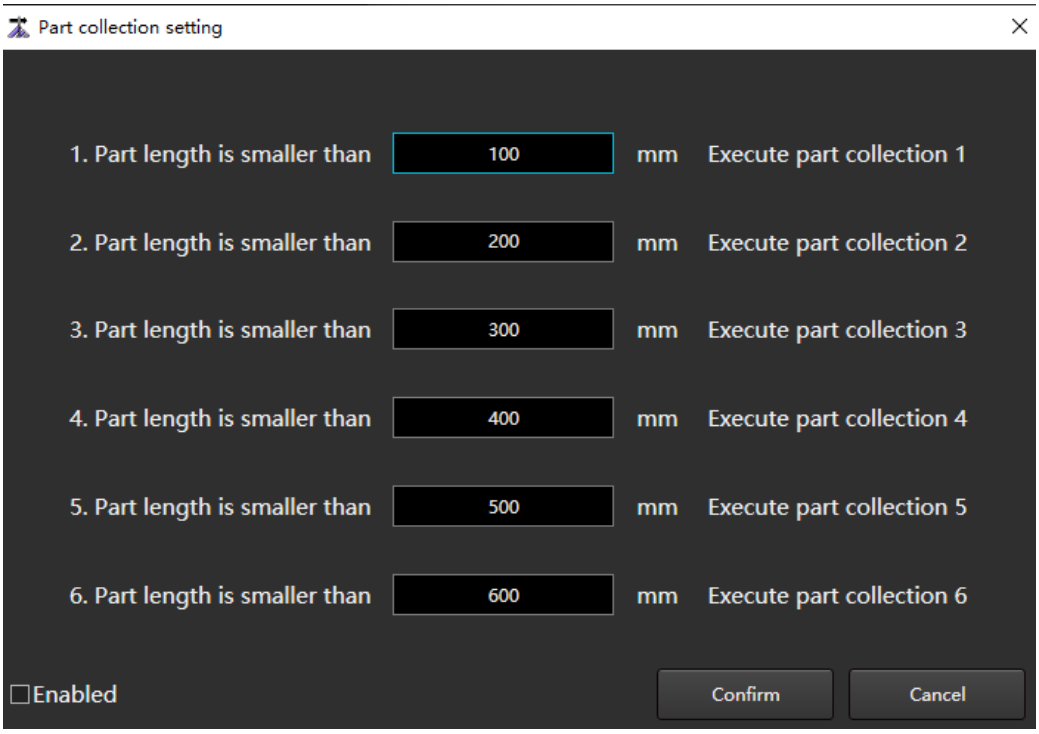
To enable the tail material processing function, you need to configure it in **NcConfig** → **Configuration** → **Process Edit**.

7.1.4. Part Collection Setting

The system provides 6 part collection actions. You can set the length of different parts and perform different parts collection actions.

Operating Steps：

1. In the menu bar, click **Maintain** → Part collection setting to open the **Part Collection Setting** dialog box:



2. Sets the part length.
3. Check **Enabled**.
4. Click **Confirm**.

Related Tasks：

To enable the part collection function, you need to configure it in **NcConfig** → **Configuration** → **Process Edit**.

7.2. External Device

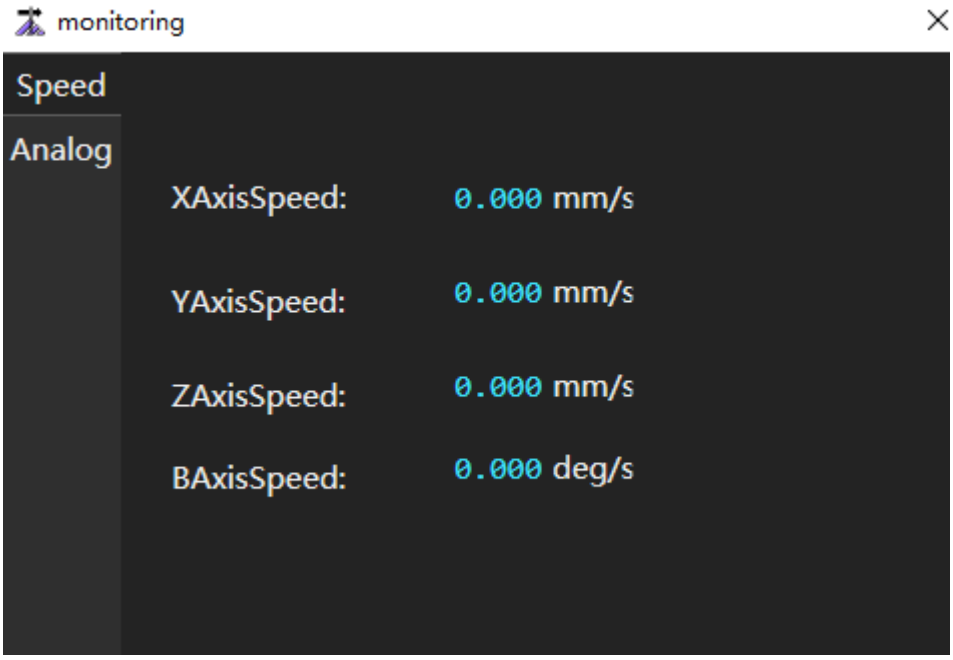
7.2.1. Monitoring

Real time monitoring of the following information：

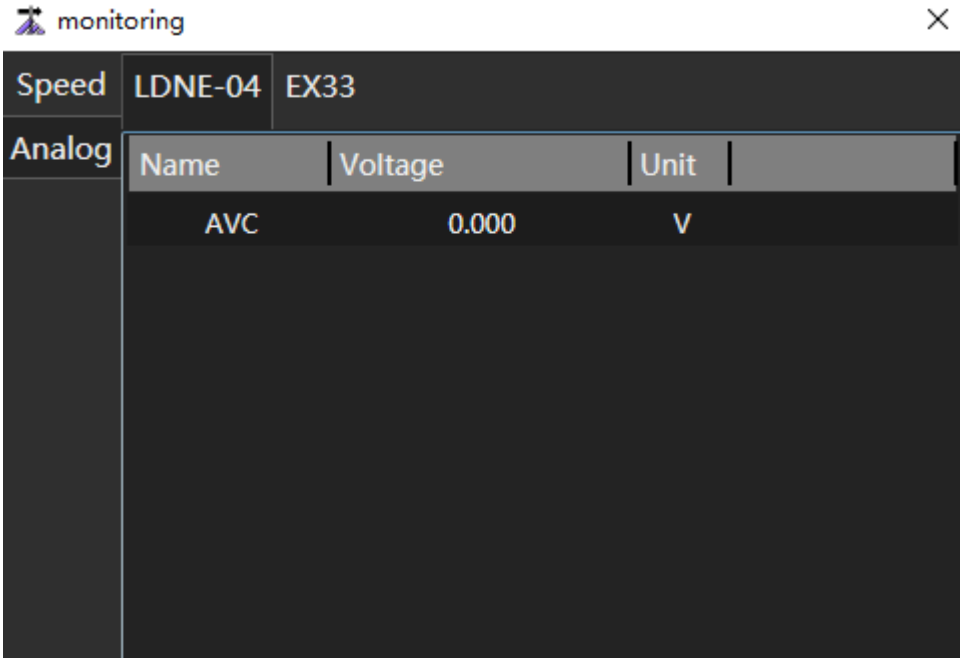
- Motion speed of each axis
- Port voltage value of Lambda controller and expansion board

Operating Steps：

1. In the menu bar, click **Maintain** → monitoring to open the **Monitoring** dialog box：



2. Click **Analog** to switch pages to view the Lambda controller and expansion board.



7.2.2. Laser Monitor

Check the status of the laser driver, such as power, temperature, water flow, mode, alarm, etc.

Operating Premise：

- Make sure that the laser driver is in good condition.
- Ensure that NcStudio software communication is normal.

Operating Steps：

1. In the menu bar, click **Maintain** → Laser Monitor:
    - If there is a specific path in the **Laser Device Program Path** of the system parameter **Advanced Function Parameter** → **Laser Device Path**, directly open the upper computer software of the laser.

- If the parameter Laser Device Program Path does not exist, the file selection dialog box will pop up to select the path of the upper computer software of the laser device.

7.2.3. Lubricate Screw

The lead screw shall be lubricated after the machine tool runs for a period of time.

There are two methods:

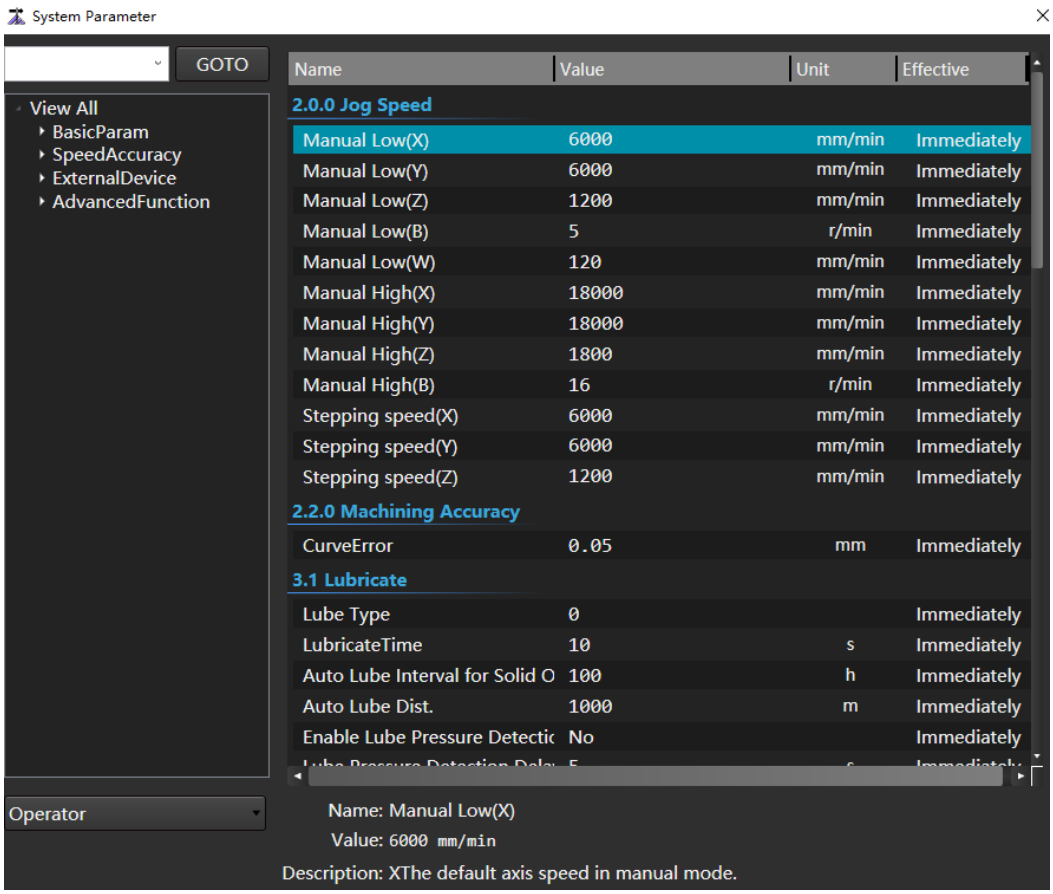
- Auto: The system can automatically perform lubrication during machining according to the set parameters by setting parameters.
- Manual: Manually control the machine tool to perform lubrication.

Operating Steps:

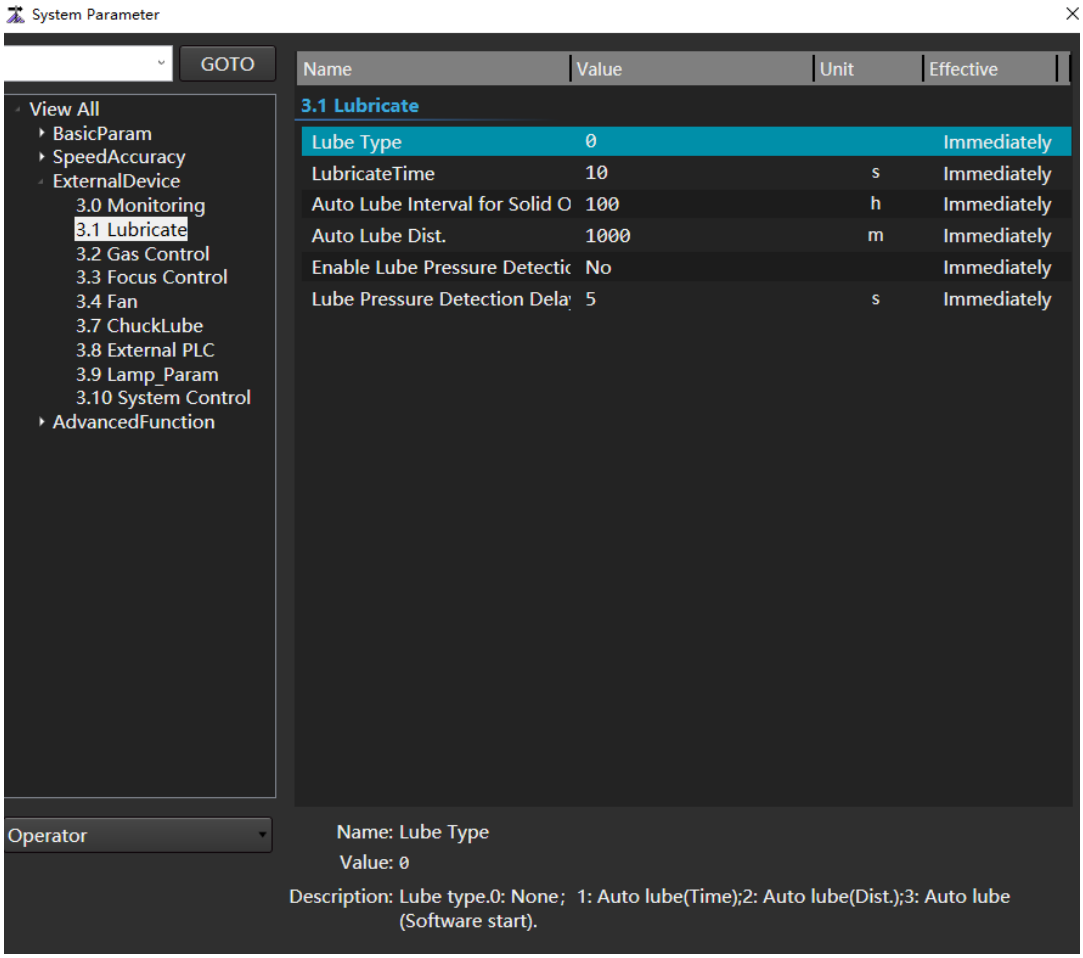
- Auto

a. In the menu bar, click **Set** → 

to open the **System Parameter** dialog box:



b. In the parameter tree on the left, select the node **View All** → **External Device** → **3.1 Lubricate**. Lubrication parameters and parameter information are displayed on the right:



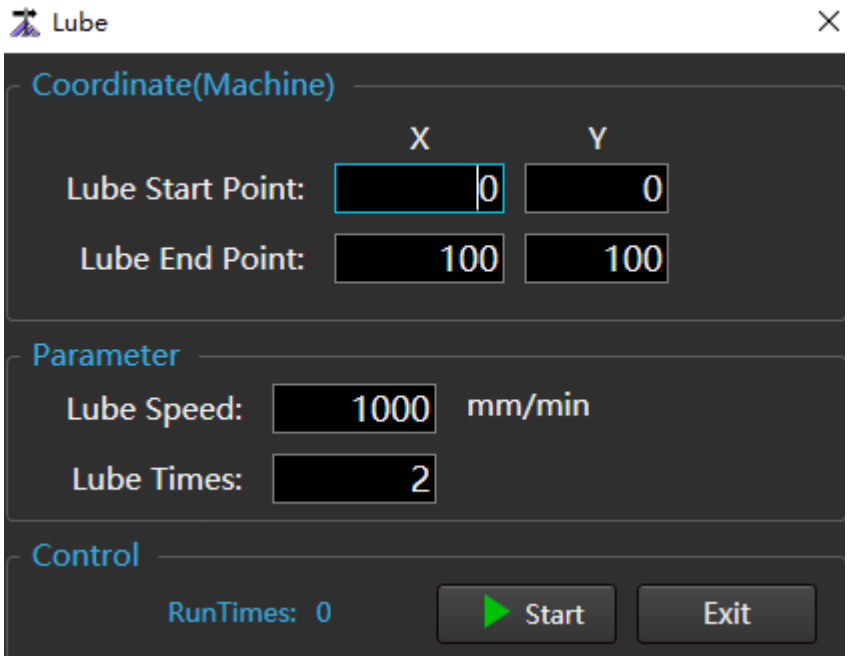
c. Set manufacturer parameters **Lube Type** :
   - **0**: Do not enable auto lube
   - **1**: Auto lube (time)
   - **2**: Auto lube (distance)

d. Set the following parameters according to the selected lubrication type:
   - **Lubricate Time**
   - **Auto Lube Interval for Solid Oil Pump**: Set when the lubrication type is **Auto Lube (Time)**.
   - **Auto Lube Dist.**: Set when the lubrication type is **Auto Lube (Distance)**.
   - **Enable Lube Pressure Detection**
   - **Lube Pressure Detection Delay**

After setting, the auto lube interval for solid oil pump/auto lube distance interval of the system every time. Auto open lube port duration **Lubricate Time**.

- Manual

a. In the menu bar, click **Maintain** → Lube to open the **Lube** dialog box:



b. Set the corresponding parameters.
c. Click **Start**, and the system starts to execute lubrication action, and displays the lubricated times.

7.2.4. Tube Support Debugging

Set the running strategy of the tube support and debug whether the tube support is running properly.

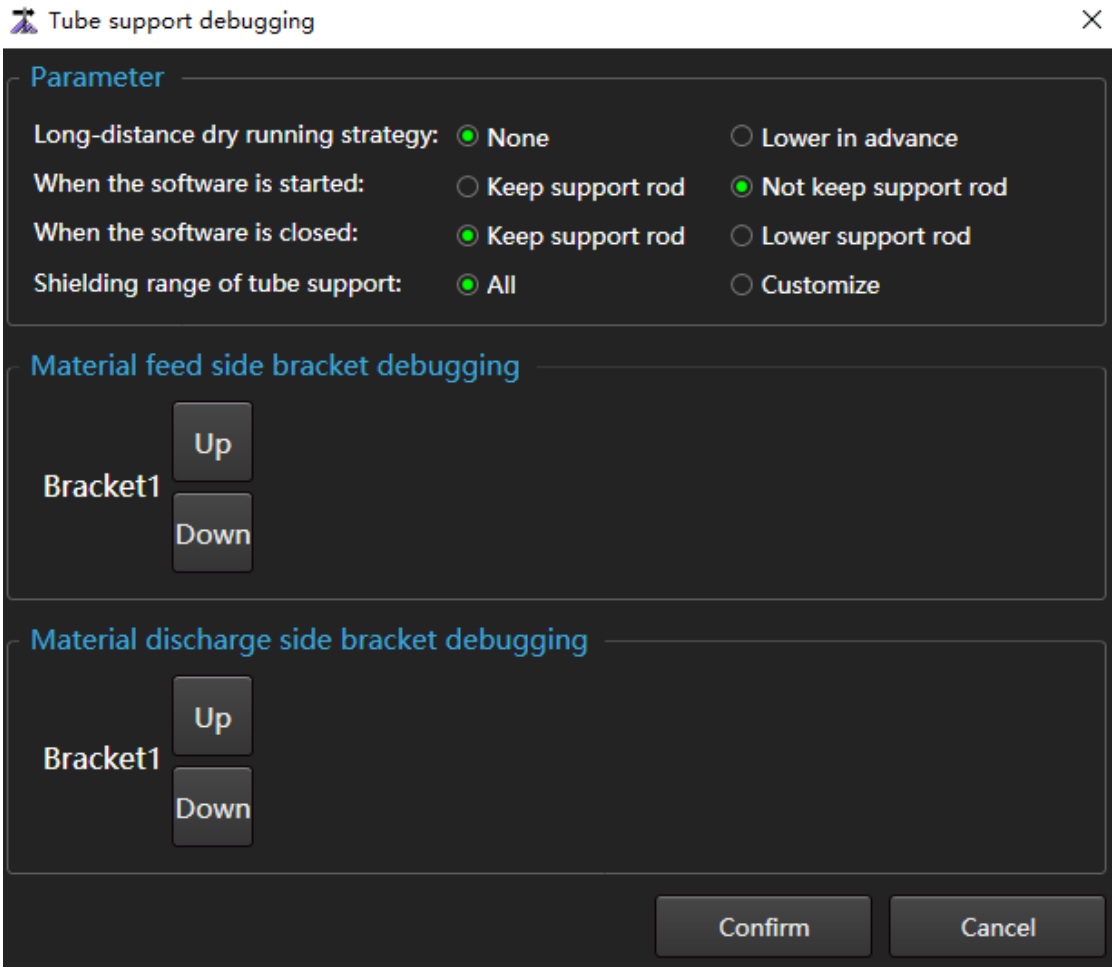
Click on the **Machine Tool Control Bar** 

. Enable or shield tube support.

Operating Steps：

1. In the menu bar, click **Maintain** → 

to open the **Tube Support Debugging** dialog box：



2. In the **Parameter** area, set the action strategy.
3. Click **Up** or **Down** to debug the equipment in the **Material Feed Side Bracket Debugging** area and **Material Discharge Side Bracket Debugging** area.
4. After debugging, click **Confirm** to exit.

7.2.5. Servo Carrier Shaft Debugging

Term：

- Carrier mechanism： On the tube cutting machine, the mechanical structure supporting the tube.
- Servo carrier： A carrier mechanism that uses a servo motor to dynamically adjust the highest point position of the carrier mechanism.
- Bed roller carrier： The carrier mechanism is at the feeding side.
- Blanking roller carrier： The carrier mechanism is at the blanking side.

Function of Servo Carrier：

When machining long and thin tubes, when the chuck rotates, the tubes will sag, deform and shake, which will affect the machining safety and machining accuracy. The servo carrier mechanism can solve the problem well. The principle is that the highest point of the carrier mechanism is always just in contact with the lowest point of the tube when the chuck is rotating or when the chuck is stationary.

This section takes the bed roller carrier as an example to introduce the debugging methods and steps of the servo carrier.

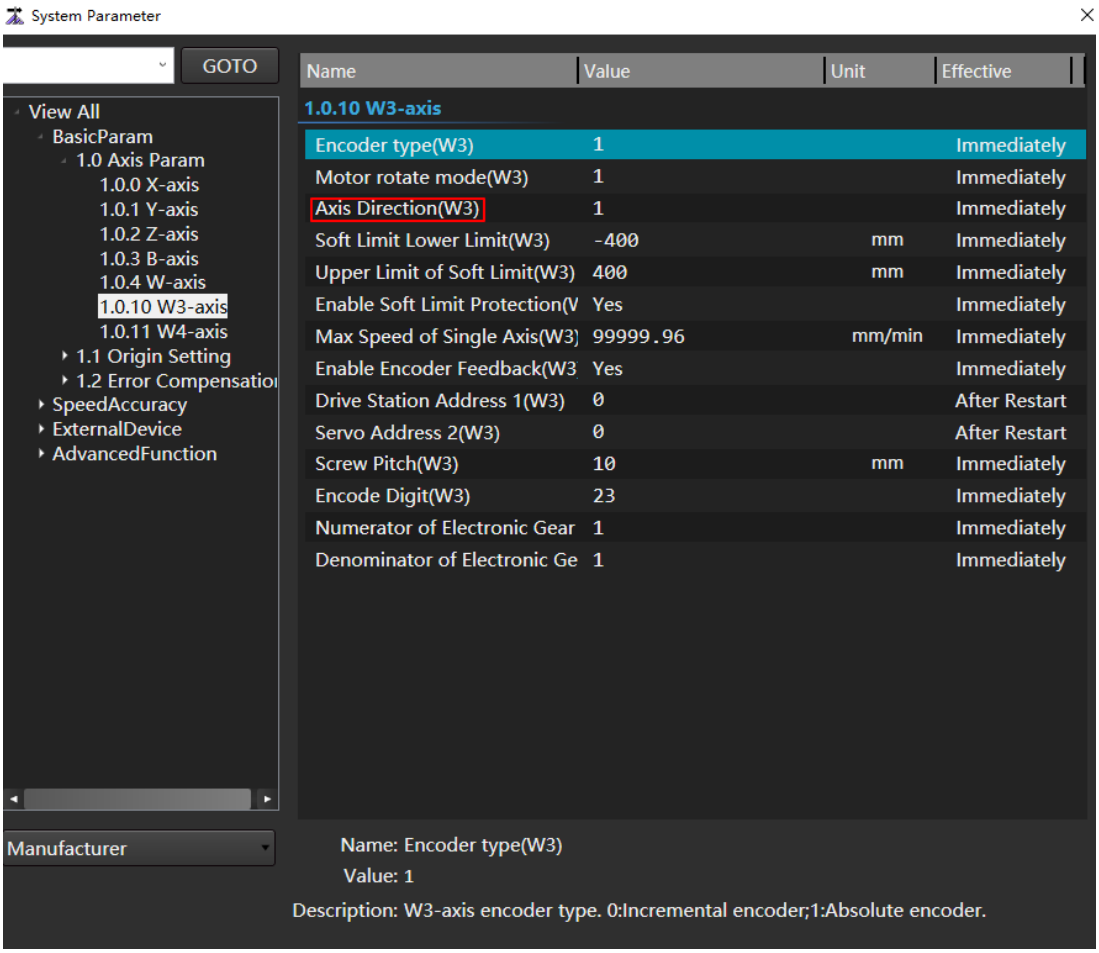
Operating Premise：

- Chuck debugging has been completed.
- The feeding shaft and servo carrier shaft are debugged and can move normally.
- The servo carrier shaft has finished returning to the mechanical zero point, or the reference has been set.
- The support cylinder can work normally.

Operating Steps：

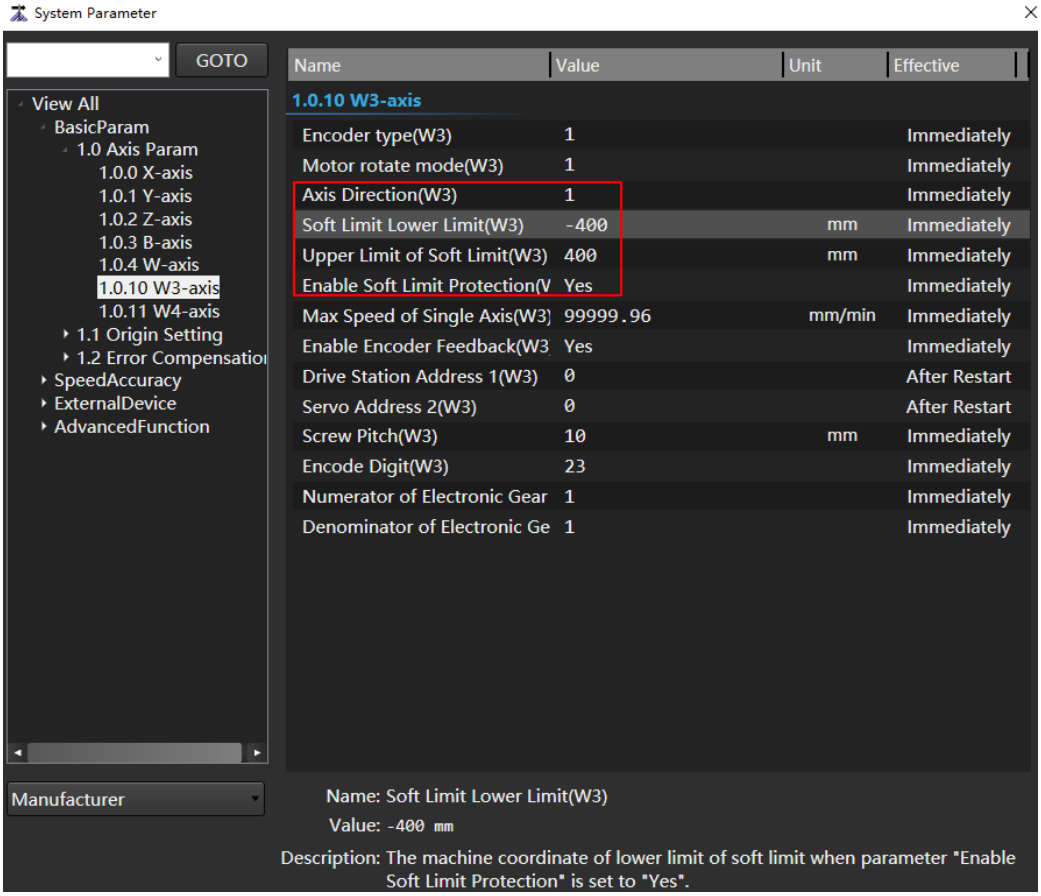
1. Adjust the movement direction of the **Servo Carrier Shaft** to ensure that when the **Servo Carrier Shaft** moves in the positive direction, the **Carrier Highest Point** rises and the reverse movement decreases.
   a. In the menu bar, click **Set** → 

to open the **System Parameter** dialog box.
   b. In the node tree, click the **Show All** → **Basic Param** → **1.0 Axis Param** node.

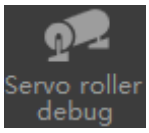


2. Set the positive and negative soft limits of **Servo Carrier Shaft** to ensure that the position of **Carrier Highest Point** will change when **Servo Carrier Shaft** moves. The purpose of this step is to ensure that when the **Support Cylinder** is supported, the **Carrier Highest Point** position is completely adjusted by the **Servo Carrier Shaft**.

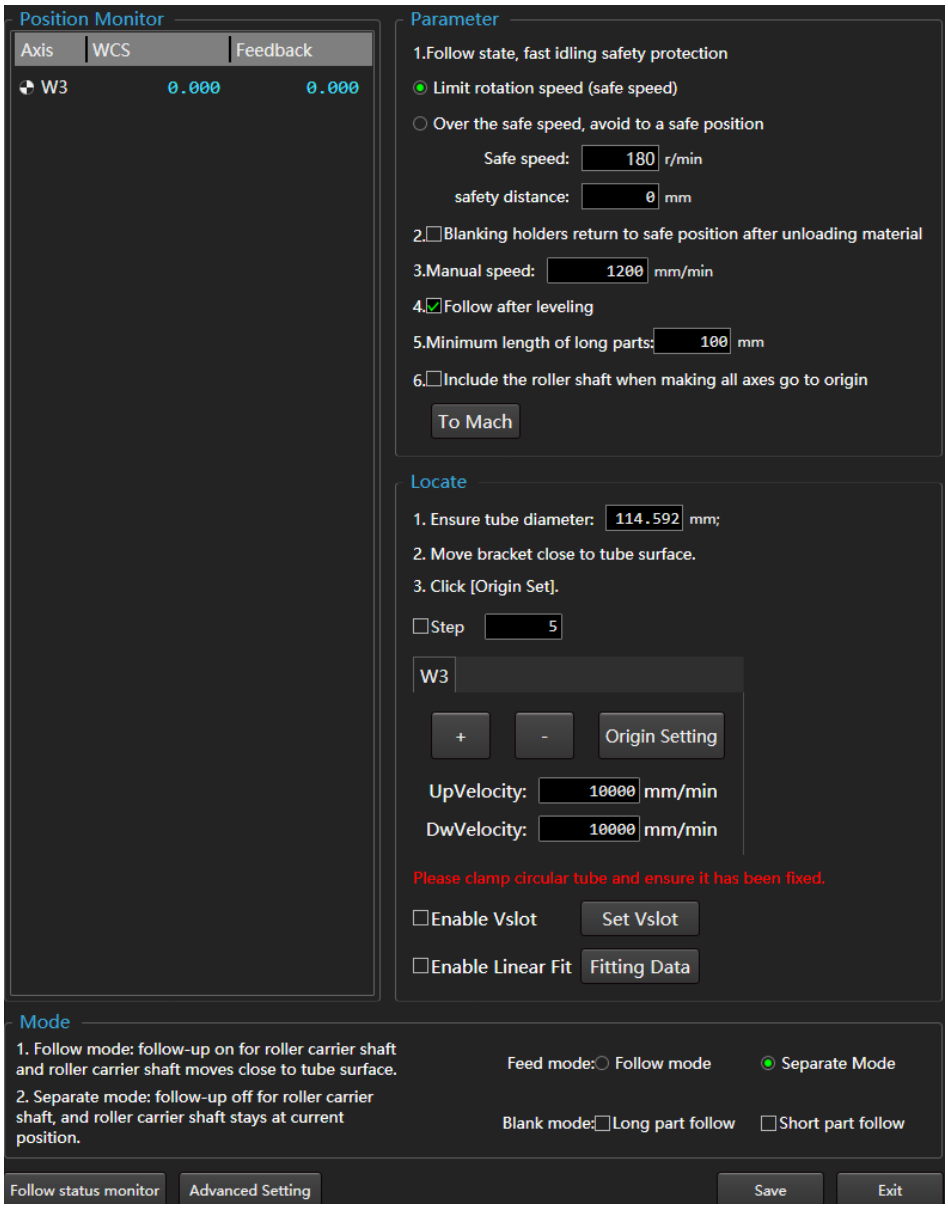
a. In the node tree, click the **Show All → Basic Param → 1.0 Axis Param** node.
b. In the **1.0 Axis Param** parameter column, adjust the positive and negative soft limit of **Servo Carrier Shaft**, and enable the soft limit protection.



3. When the **Servo Carrier Shaft** moves in a 5mm step, record several groups of **Carrier Highest Point** height values.

a. In the menu bar, click **Maintain →** 

to open the **Servo Roller Debug** dialog box:



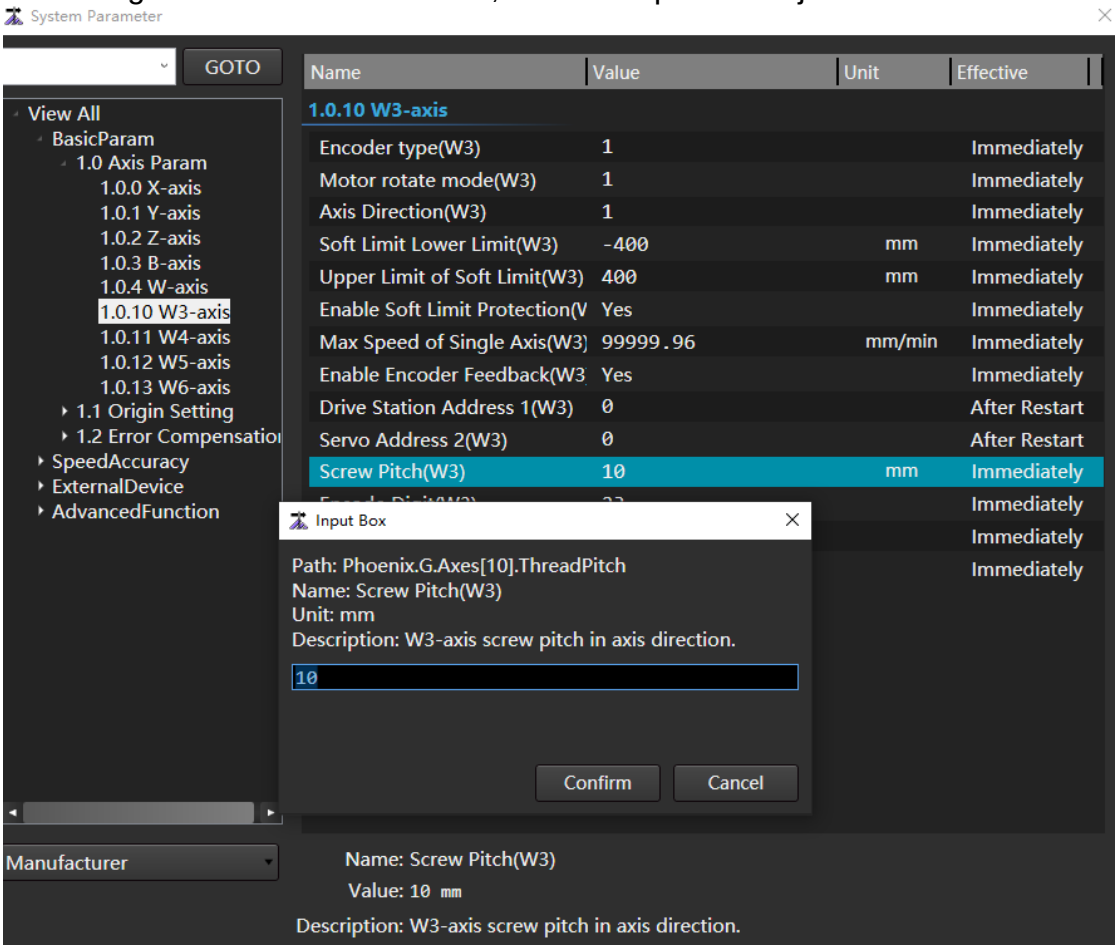
b. In the **Locate** area, click the **+** button to make the W axis (servo carrier shaft) move near the upper limit of its soft limit, and the **Carrier Highest Point** is at the highest position.
c. Record the value of **Carrier Highest Point** at the highest position.
d. Check **Step**, switch to step mode, and set the step size to 5.
e. Click the **-** button and record the corresponding **Carrier Highest Point** until the W axis (servo carrier shaft) moves to the lower limit of its soft limit.

For example: The following table shows the debugging data of a customer.

W3 Axis Step Size	Carrier Highest Point	Change of Carrier Highest Point
5	172.5	-
5	169.1	3.4
5	164.8	4.3
5	160.7	4.1
5	156.4	4.3

W3 Axis Step Size	Carrier Highest Point	Change of Carrier Highest Point
5	164.8	4.3
5	160.7	4.1
5	156.4	4.3
5	152.1	4.3
5	147.8	4.3
5	143.6	4.2

4. Calculate the average value of carrier highest point change.
   For example, in the above table, when the W3 axis (servo carrier shaft) step is 5mm, the average change of **Carrier Highest Point** is (4.3+4.1+4.3+4.3+4.2)/6=4.25. Note, that discrete data records must be discarded in this step, such as 3.4 in the table above.
5. Calculate the ratio between the average change value of **Carrier Highest Point** and the movement step size of **Servo Carrier Shaft**.
   For example, in the above table: K = 4.25 / 5 = 0.85
6. Adjust the **Servo Carrier Shaft** pitch according to the ratio between the average value of **Carrier Highest Point** change and the **Servo Carrier Shaft** motion step.
   New **Servo Carrier Shaft** pitch=original **Servo Carrier Shaft** pitch * K
   According to the above calculation, the screw pitch is adjusted from 10 to 8.5.



7. Perform **Servo Carrier Shaft** calibration or homing, and reset the upper and lower limit values of the **Servo Carrier Shaft** soft limit.

Note

This step is because the screw pitch of **Servo Carrier Shaft** has been adjusted.

8. Repeat step 3 (step 4, step 5, step 6 and step 7 will not be executed), and verify whether the **Servo Carrier Shaft** moves 1mm and the **Carrier Highest Point** height changes 1mm after adjusting the **Servo Carrier Shaft** pitch.
   For example: The following is the data of a customer's machine tool after adjusting the **Servo Carrier Shaft** pitch.

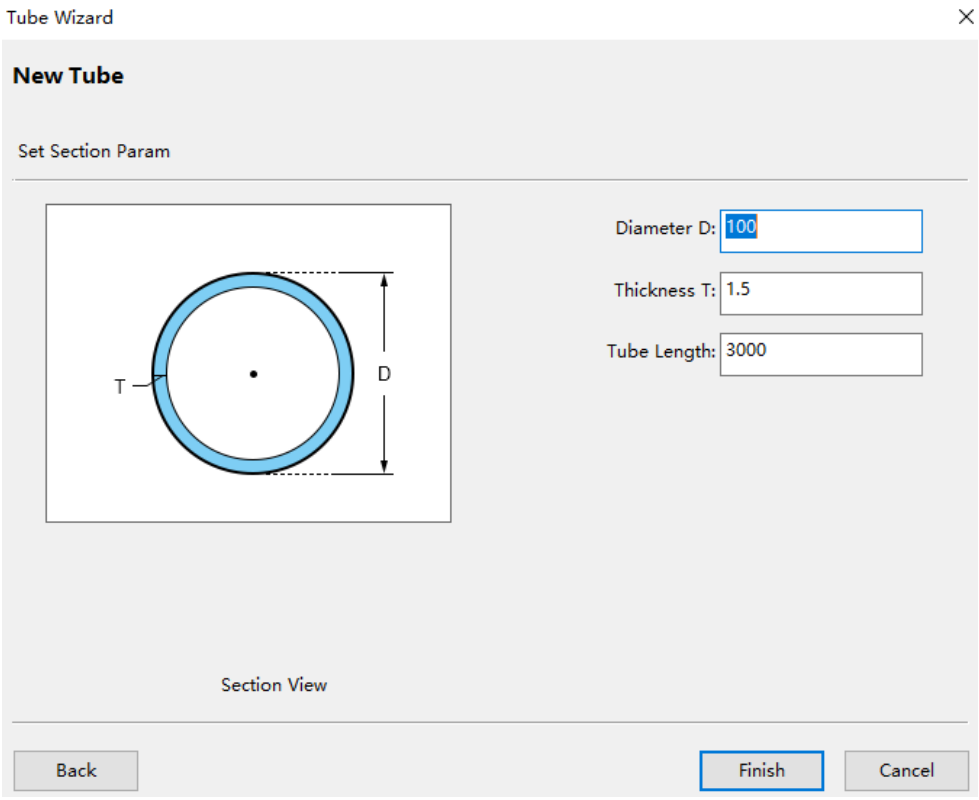
W3 Axis Step Size	Carrier Highest Point	Change of Carrier Highest Point
5	172.5	-
5	167.8	4.8
5	162.7	5.1
5	160.7	4.1
5	157.4	5
5	152.9	4.8

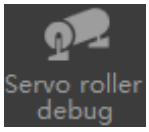
Note

Description: If the **Servo Carrier Shaft** pitch is adjusted, the **Servo Carrier Shaft** moves for 1mm, and the height change of the **Carrier Highest Point** differs greatly from 1mm, step 3 must be repeated (step 4, step 5, step 6 and step 7 must be performed at the same time).

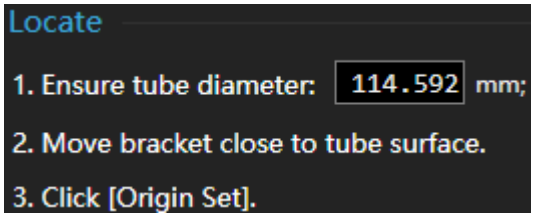
1. Use circular tube calibration.
   a. Manually install a standard circular tube.
   b. In the menu bar, click **Common**→ 

→ **Tube** to Open the **Tube Wizard** dialog box.
   c. Select the circular tube and open the **Set Section Param** dialog box:



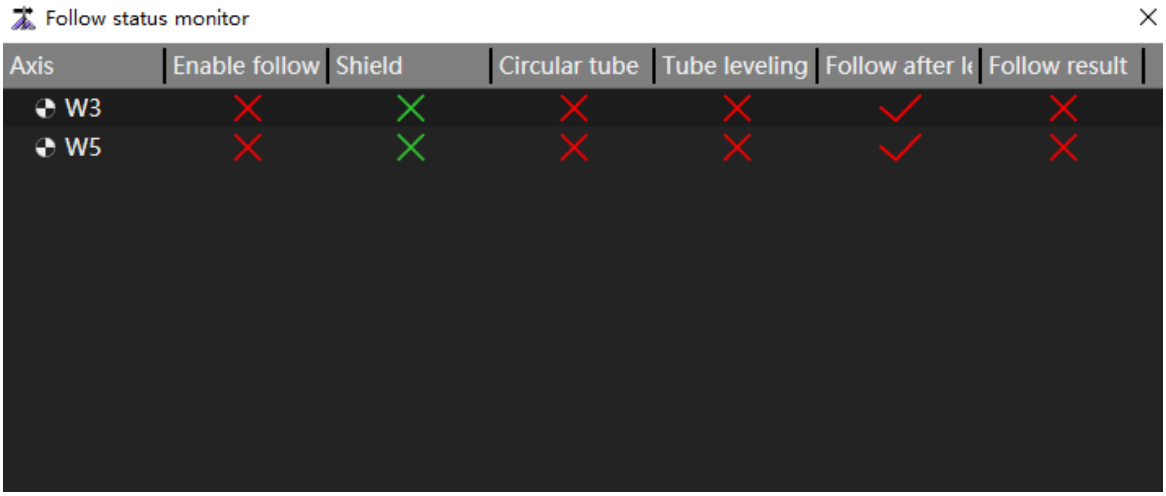
d. Set the tube section parameters. Note that the direct and thickness should be consistent with the current tube on the machine tool.
e. Ctr+s or click the **Save** button to save the machining file.
f. In the menu bar, click **Maintain** → 

to open the **Servo Roller Debug** dialog box.
g. Calibration: In the **Locate** area, complete the calibration.



h. Set the follow mode. In the **Mode** area, select **Follow Mode**.

2. In the **Servo Roller Debug** dialog box, click **Follow Status Monitor** at the bottom left corner. In the **Follow Status Monitor** dialog box, check whether the current follow takes effect normally, and the specific reason why it does not take effect.



3. Click **Save** to finish debugging.

Caution

In consideration of tube sagging, it is recommended that the carrier highest point close to the chuck should be 1~3mm away from the lower surface of the tube during calibration to avoid the carrier from pressing the tube out of the indentation during normal machining after commissioning.

7.2.6. Adjust Air Pressure

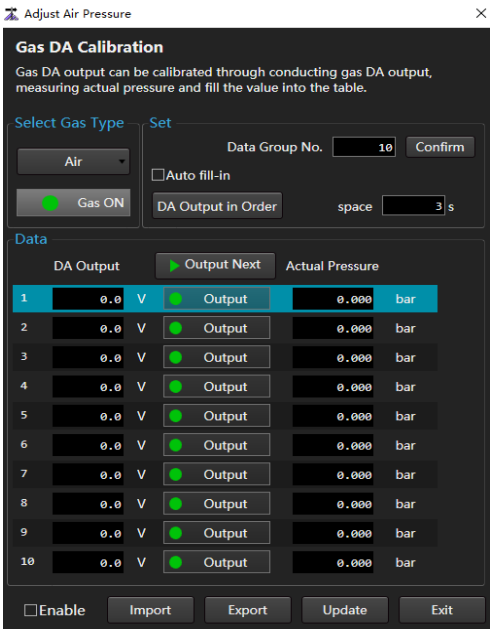
Adjust air pressure means to correct the analog output of gas, so that the actual air pressure is consistent with the analog output.

DA refers to converting digital quantity into analog quantity.

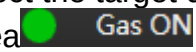
Operating Steps：

1. In the menu bar, click **Maintain** → 

to open the **Adjust Air Pressure** dialog box:



2. In the **Select Gas Type** area, select the target gas.
3. In the **Set** area, set the data group no. and click **Confirm**.
   By default, the number of data groups is 10. In the **Data** area, there are 0 to 9 rows of data.
4. Perform different operations according to different ways of filling **DA Output** column data.
   - Auto: Check **Auto Fill-in** and click **Confirm**.
   - Manual: In the **DA Output** column, fill in the values in turn.
5. Perform different operations according to the way to fill in the **Actual Pressure** column data.
   - Automatically output analog quantity according to the set time interval:
     Click the **Space** input box to input the set value, and click **DA Output in Order** in turn.
   - Manual:
     i. In the **Data** area, select the target data and turn it on 

     , **Select Gas Type** area 

     is highlighted, gas output.
     The current display value of the proportional valve is the actual air pressure value.
     ii. Fill the displayed value of the proportional valve into the corresponding **Actual Pressure** column.
6. **Enable**.

- Check: Do adjust air pressure. It is recommended to check when the DA output data is inconsistent with the actual air pressure data.
  - Uncheck: Do not adjust air pressure.

1. Click **Update**.

Related Tasks:

- Export: Click **Export** to save the current data locally.
- Import: Click **Import** to import the locally saved data into the current **Adjust Air Pressure** dialog box.

7.2.7. Focus Calibration

Correct the focus so that the position distance is consistent with the analog output.

Operating Premise:

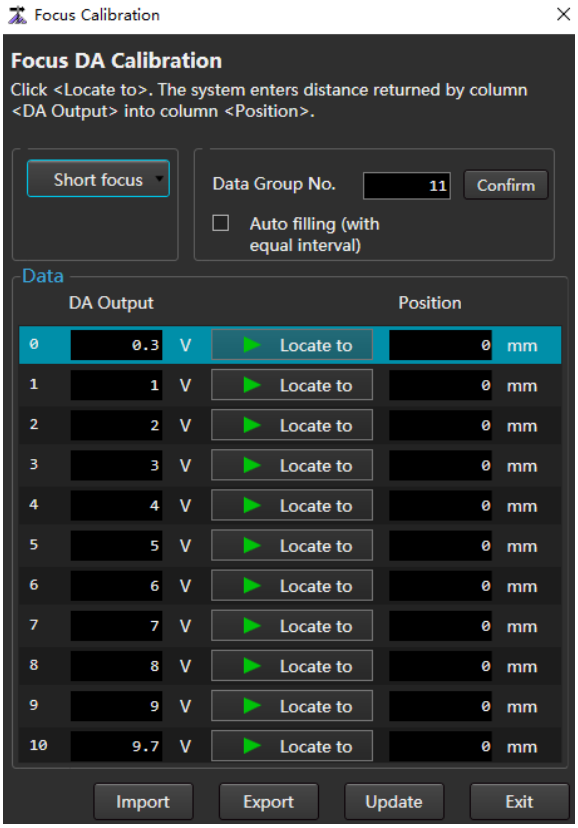
Before correcting the focus, set the following parameters and restart the software:

- Set the parameter **Enable Focus Control** to **Yes** to enable the focus control function.
- Set the parameter **Focus Control Type** to **Precitec Auto Focus**.

Operating Steps:

1. In the menu bar, click **Maintain** → 

to open the **Focus Calibration** dialog box:



2. Select **Short Focus** or **Long Focus** according to the cutting head specification.
3. Enter a value in **Data Group No.** and click **Confirm**.
   By default, the number of data groups is 10. In the **Data** area, there are 0 to 9 rows of data.
4. Perform different operations according to different ways of filling **DA Output** column data.
   - Auto: Check **Auto Filling** and click **Confirm**.
   - Manual: In the **DA Output** column, fill in the values in turn.
5. In the **Data** area, select the target data and turn on Locate to.
6. Connect the cutting head through bluetooth and fill the data viewed on the Precitec Procutter mobile phone software into the corresponding **Position** column.
7. Click **Update** to complete the focus calibration.

Related Tasks:

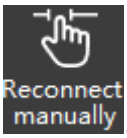
- Export: Click **Export** to save the current data locally.
- Import: Click **Import** to import the locally saved data into the current **Focus Calibration** dialog box.

7.3. Tool

7.3.1. Reconnect Manually

It is applicable to the system with bus configuration. After the communication between the system and the drive is disconnected, the communication with the drive can be reestablished through manual reconnection.

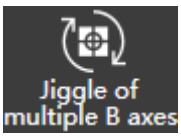
Operating Steps:

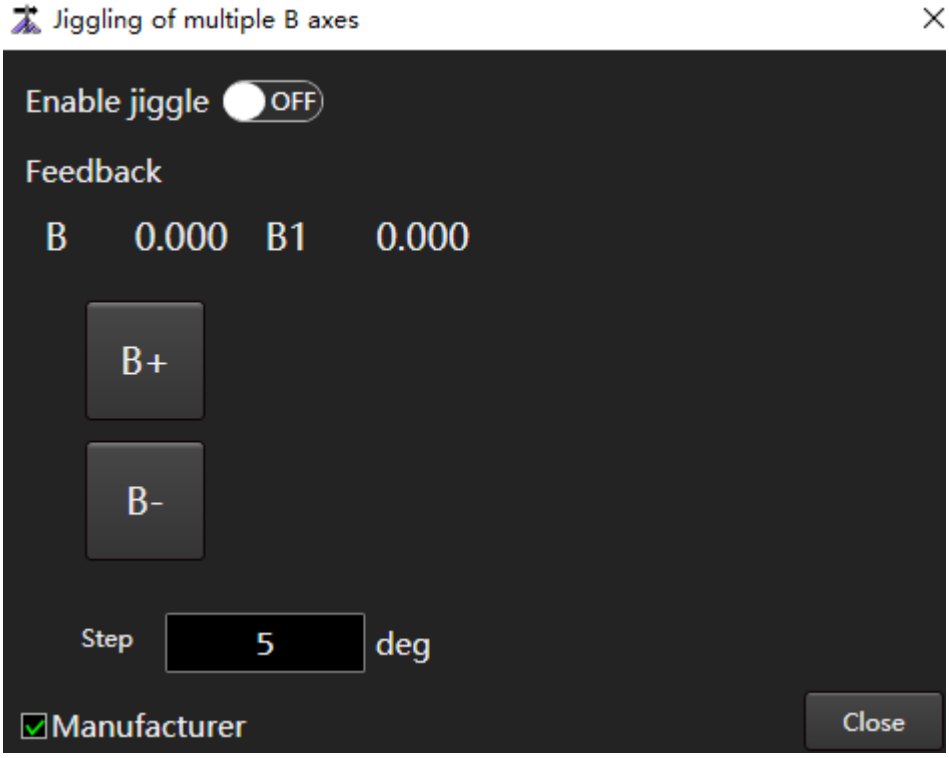
1. In the menu bar, click **Maintain** → 
.

7.3.2. Jiggle of Multiple B Axes

Juggle rotation axis provides an independent and quantitative control means for double rotation axis. According to the actual synchronous deviation of the two rotating shafts, the feedback coordinates of the two rotating shafts can be adjusted to be consistent to complete manual correction.

Operating Steps：

1. In the menu bar, click **Maintain** → 
 to open the **Jiggling of Multiple B Axes** dialog box:



2. Set **Enable Jiggle** to the **ON** state.
3. Click the step input box to set the step distance of jiggle.
   The default is 5.
4. Click **B+** / **B-** / **B1+** / **B1-**. Adjust the feedback coordinates of axis B or B1.

7.3.3. One-click Cutoff

One-click cutoff. Use large graphic technic to cutoff at the current Y axis coordinate position.

The section that is forbidden to be executed is not closed or irregular tube, otherwise there is danger.

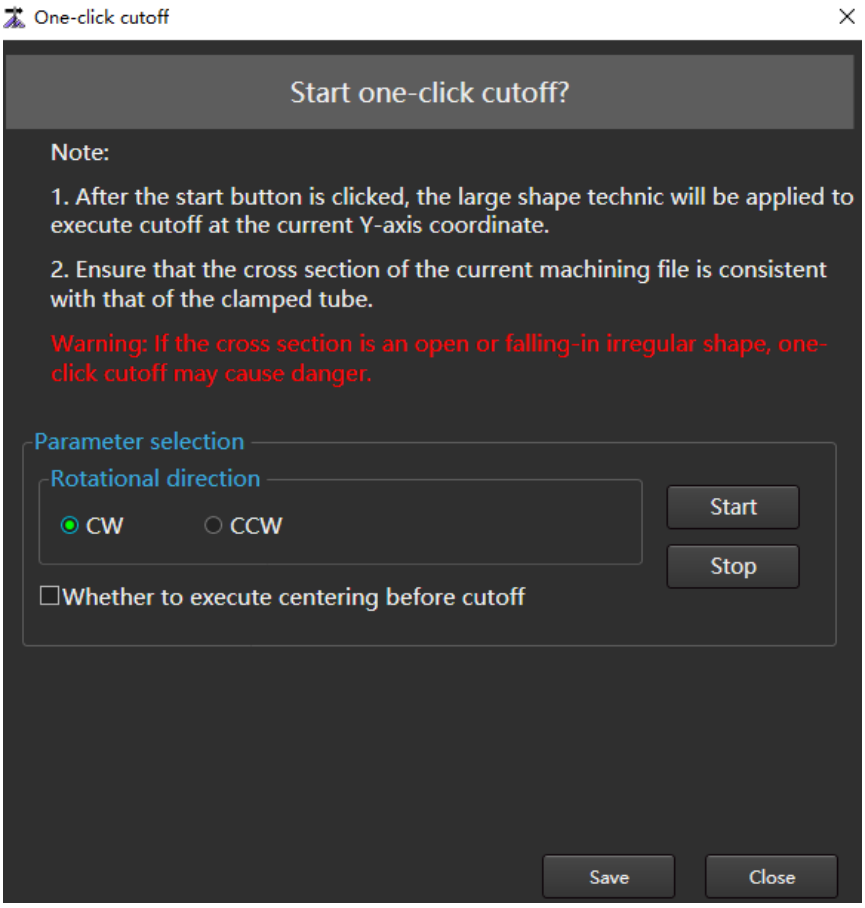
Operating Premise：

Ensure that the section of the current machining document is consistent with the actual clamped tube.

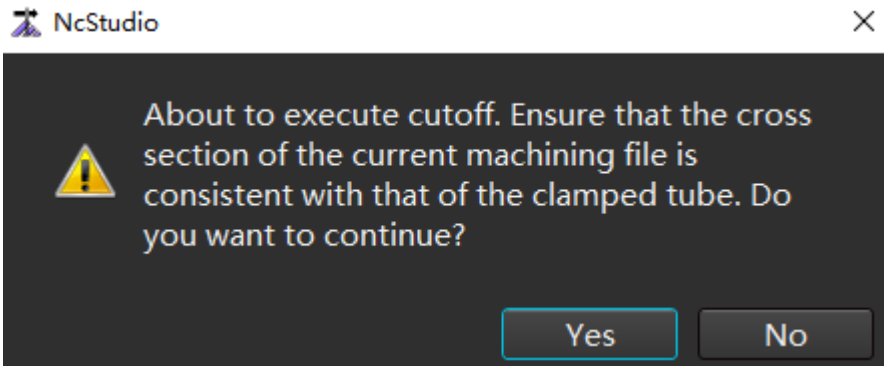
Operating Steps：

1. In the menu bar, click **Maintain** → 

to open the **One-click Cutoff** dialog box:



2. Select the rotational direction.
3. Check **Whether to Execute Centering Before Cutoff** as required.
4. Click **Start** to pop up a prompt box:



5. Click **Yes** to start the cutoff operation.

7.3.4. Custom Instructions Debug

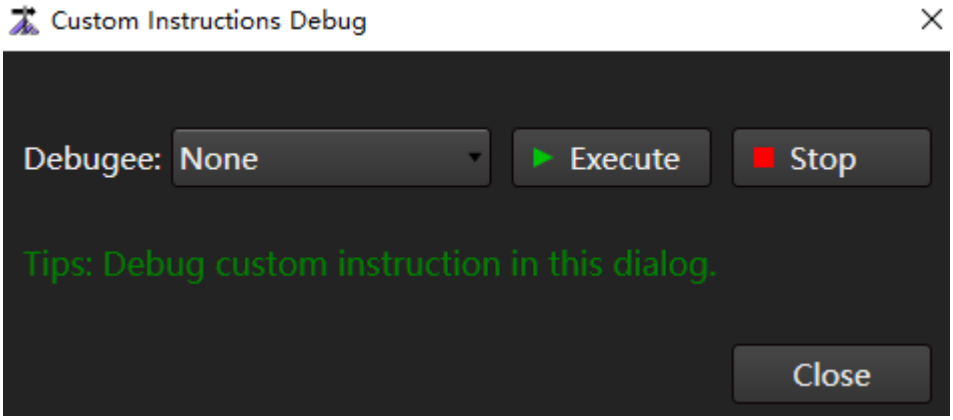
Whether the custom instructions debug is reasonable. The customized instructions are configured in **NcConfig** → **Configuration** → **Flow Edit**.

Caution

If the process is written incorrectly, the machine tool may be damaged. Please stop debugging in time when executing.

Operating Steps：

1. In the menu bar, click **Maintain**→ Custom Instructions Debug to enter the manufacturer password and open the **Custom Instructions Debug** dialog box：



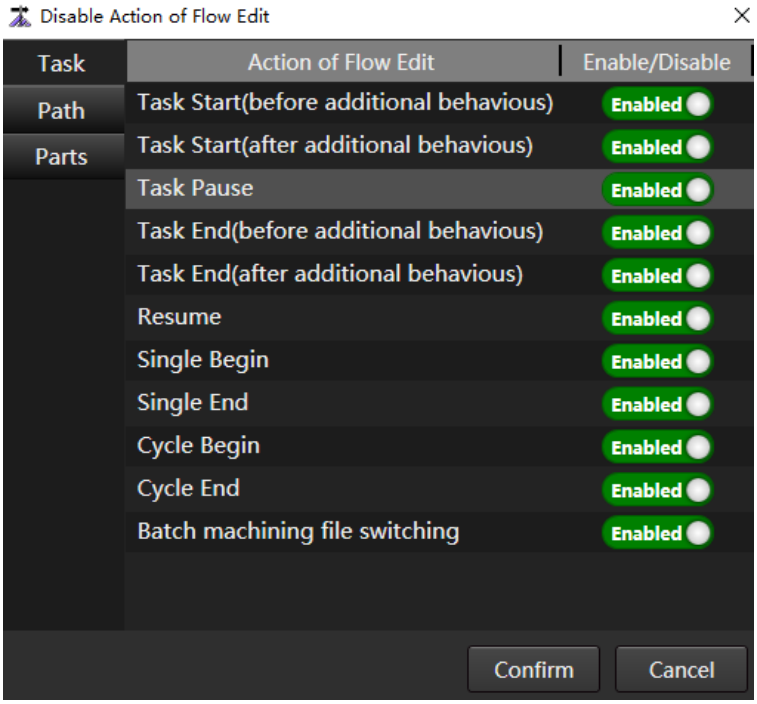
2. Select the instruction in the **Debugee** drop-down box.
3. Click **Execute** to check whether the instruction is reasonable.

7.3.5. Disable Action of Flow Edit

Disable or enable action of flow edit to include user defined instructions.

Operating Steps：

1. In the menu bar, click **Maintain** → Disable Action of Flow Edit to open the **Disable Action of Flow Edit** dialog box：



The actions in the **General** tab are user defined instructions.

2. Disable or enable action of flow edit as required.
   - Disable: That is, the button is set to the 

state.
   - Enabled: That is, the button is set to the 

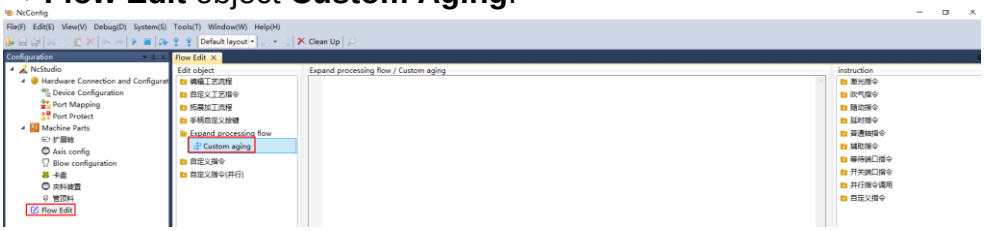
state.
3. After setting, click **Confirm**.

7.3.6. Trial Run

At the initial commissioning stage of the machine tool, trial run is required to ensure the stability of the motion of each axis of the machine tool.

The system provides two methods:

- User defined trial: used for external devices such as aging chucks. The action executed is the command action of the **User Defined Trial** configuration edited by the process.
  The instruction action configuration entry is: **NcConfig** application **Configuration → Flow Edit** object **Custom Aging**.



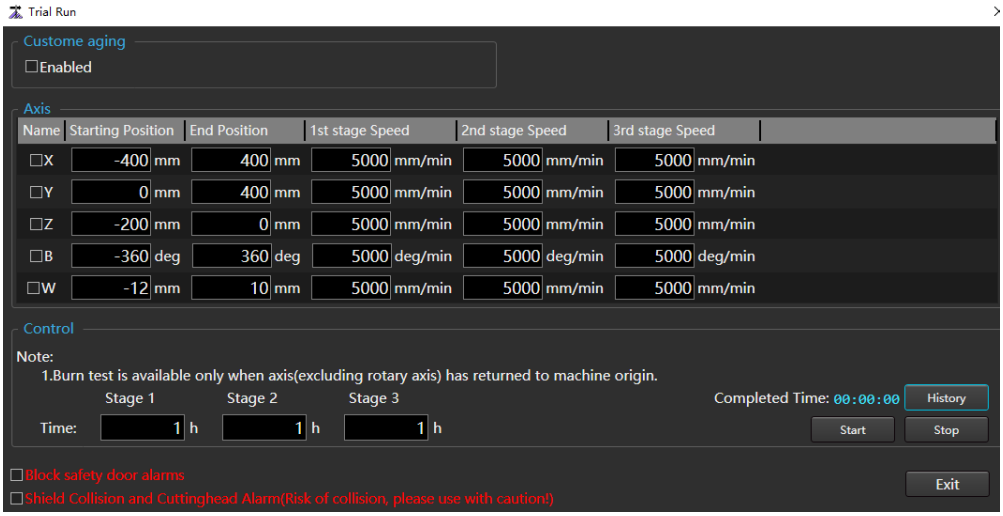
- Axis aging: Used for the movement of each axis of aging machine tool. Configure shaft parameters and control aging time through the **Trial Run** dialog box to start aging.

Operating Premise:

To ensure the safety of the machine tool, before aging the machine tool, make sure that all axes have executed Execute Return to Mechanical Origin or Set Datum.

Operating Steps:

1. In the menu bar, click **Maintain** → Trial Run to open the **Trial Run** dialog box:



2. Depending on the method selected, select to perform the following operations:
   - If you select custom aging, check **Enabled** in the **Custom Aging** area.
   - If axis aging is selected, select the axis to be aged in the **Axis** area, and set the starting/ending position and the first/second/third stage speed of aging.
3. In the **Control** area, set the time of the first/second/third aging stage.
4. At the bottom of the dialog box, check **Block Safety Door Alarms** and **Shield Collision and Cutting Head Alarm** as required.

There is a risk of cutting head collision, please use with caution!

5. Click **Start**, and the machine tool begins to age.
   During machine tool aging, click **Stop** to stop machine tool aging.

Related Tasks:

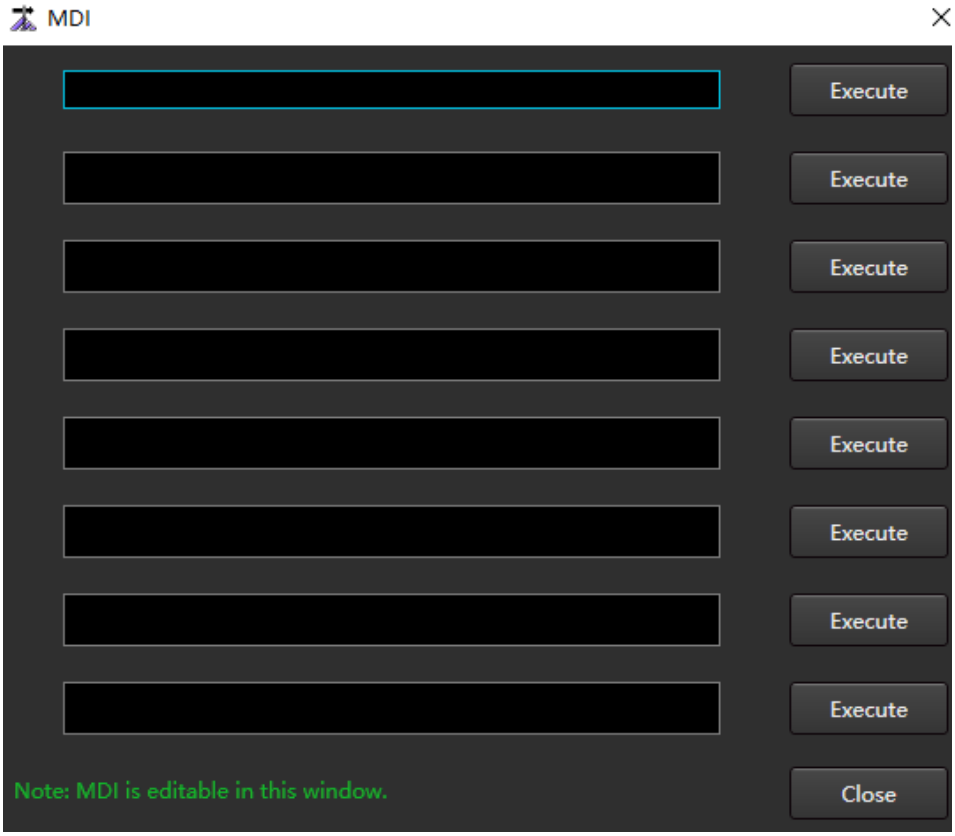
To view the machine tool aging history, click **History**.

7.3.7. MDI

The user can freely input and execute up to seven simple programming instructions in this area to realize rapid movement, change the system state or carry out simple machining.

Operating Steps:

1. In the menu bar, click **Maintain** → MDI to open the **MDI** dialog box:



2. Select the target command line, enter the command in the input box, and use; (semicolon) to wrap.
3. Click **Execute** corresponding to the line, and the system will automatically execute the entered instructions.

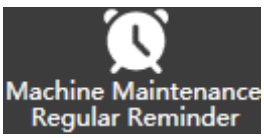
7.3.8. Machine Maintenance Regular Reminder

The machine maintenance regular reminder function can set the maintenance cycle. When the specified cycle is reached, the system will automatically pop up a prompt box to remind the user to maintain the machine tool.

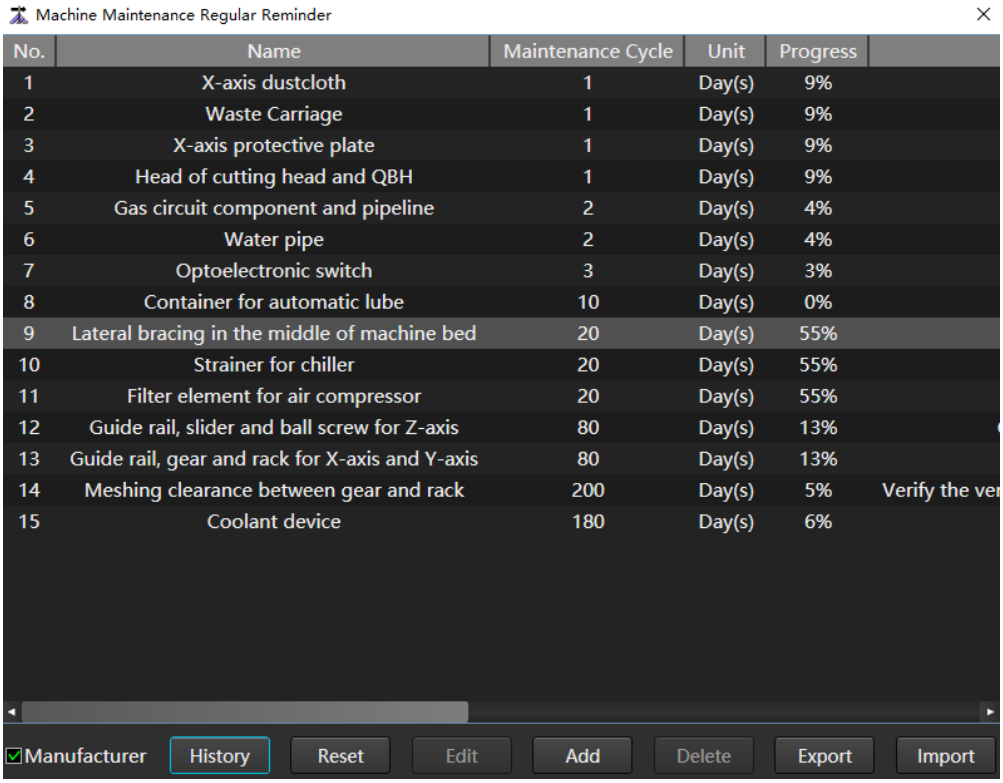
The maintenance cycle can be set according to the time cycle or the running distance of the machine tool.

On the machine maintenance regular reminder page, reset, edit, add and delete functions are also provided.

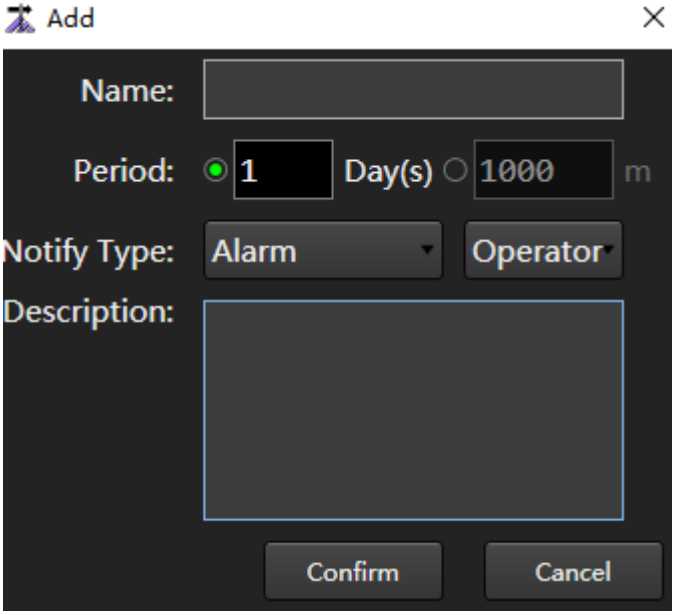
Operating Steps：

1. In the menu bar, click **Maintain** → 

to open the **Machine Maintenance Regular Reminder** dialog box:



2. Check **Manufacturer**.
3. Click **Add** to pop up the **Add** dialog box:



4. Set parameters as required. The parameters are described as follows:

Parameter	Description
Name	The name of the maintenance item must be unique.
Period	Either time/distance period. ▪ Time period description: How many days should the maintenance be carried out; The number of days is the world time obtained by computer. ▪ Distance period description: The machine tool needs to be maintained every few meters; The distance adopts the machine tool movement distance.

Parameter	Description
Notify type	Either alarm/notification. ▪ Alarm：The progress is displayed in red, and the alarm dialog box pops up. ▪ Notification：The progress is displayed in red and a "warning" log is generated.
Description	The service items were described and the maintenance contents were required to be clarified. There was no misunderstanding.

1. Click **Confirm**.

Related Tasks：

- Modify an existing maintenance item：Select the target item and click **Edit**.
- Restore the set value of the existing maintenance item to the initial value：Click **Reset**.
- Delete maintenance Item：Select the target item and click **Delete**.

7.3.9. Encoder Detection

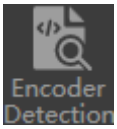
It is used to detect whether the encoder feedback is consistent with the motor rotation mode, and automatically calculate the PG frequency division ratio, so as to avoid affecting the second-generation flying cut and follow-up effects, and the inconsistency between the actual coordinates of the machine tool and the software coordinates under the non servo alarm E-stop state.

Only applicable to non bus control systems.

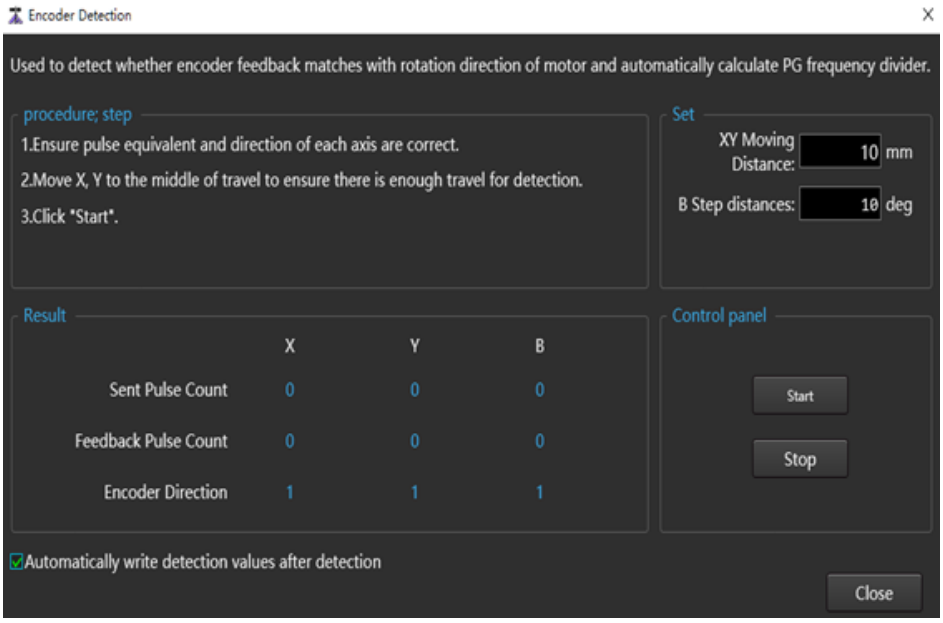
Operating Premise：

- The drive parameters are set correctly.
- The pulse equivalent of each axis, axis direction and the number of command pulses per cycle have been set correctly.
- The X axis and Y axis have been moved to the middle of the machine tool stroke, and there is enough stroke for detection.

Operating Steps：

1. In the menu bar, click **Maintain** → 

to open the **Encoder Detection** dialog box：

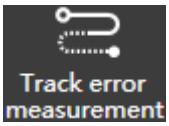


2. **（Optional：）** Set XY moving distance in the **Set** area.
   The default distance is 10mm, which is generally set as a long pitch to minimize the error of detection.
3. In the **Control Panel** area, click **Start**.
   If **Automatically Write Detection Values after Detection** is checked, the feedback data results will be automatically written into the system parameters.

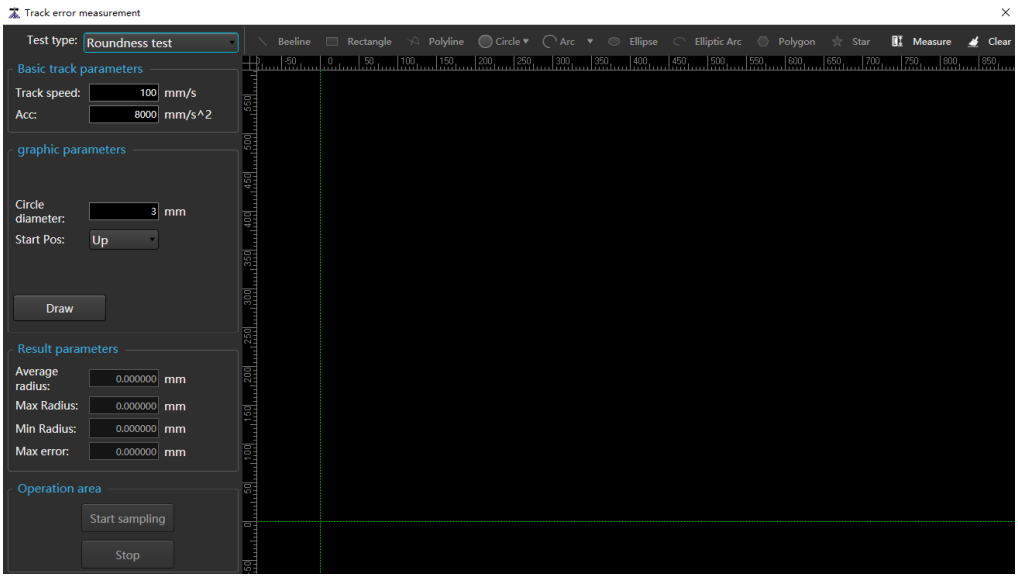
7.3.10. Track Error Measurement

Through multi axis linkage, the difference between transmission and feedback tracks is displayed, which is the basis for subsequent adjustment of driver parameters.

Operating Steps：

1. In the menu bar, click **Maintain** → 

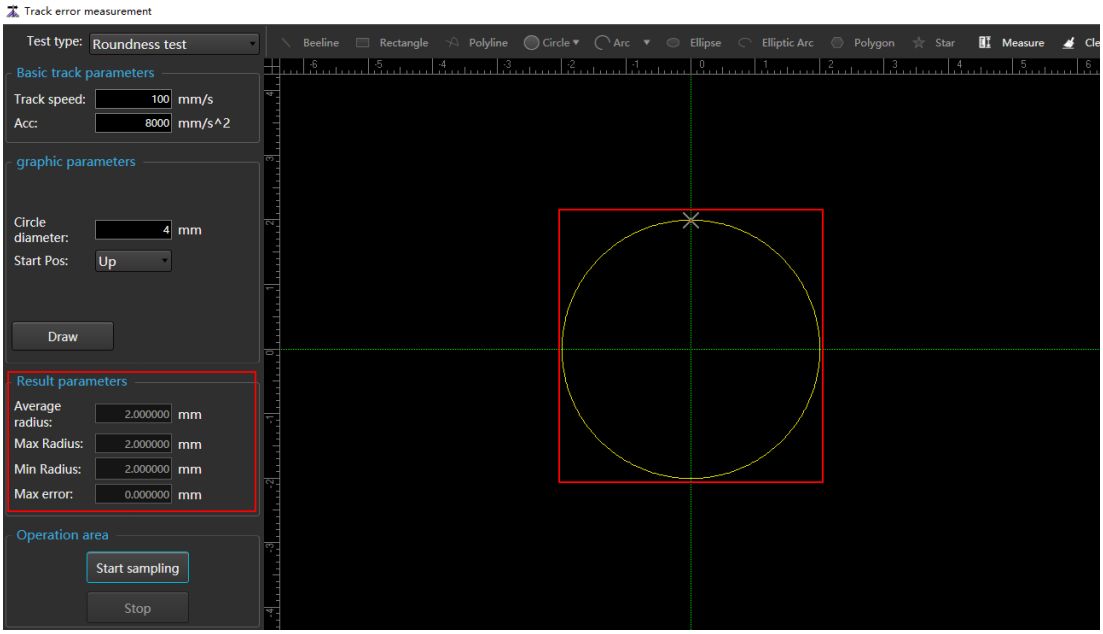
to open the **Encoder Detection** dialog box:



2. Set track speed in the **Basic Track Parameters** area.
3. Set graphic.

a. Select a graphic type from the **Test Type** drop-down box.
b. Depending on the selected graphic, do the following:
- The test type is **Roundness Test** or **Rectangularity Test**:
  i. In the **Graphic Parameters** area, set parameter information.
  ii. Click **Draw** to display the corresponding graphic in the middle drawing area.
- The test type is **Custom Track Test**:
  - Draw: Click the corresponding graphic button in the **Track Error Measurement** dialog box, and then draw a graphic in the drawing area.
  - Import: In the **Graphic Parameters** area, click **Import DXF** to select a file.

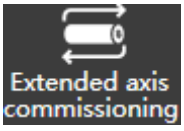
4. Click **Start Sampling**, and you can see the test track in the **Track Error Measurement** dialog box. If the measurement type is **Roundness Test**, the results are displayed in the **Result Parameters** area.
Effect picture:



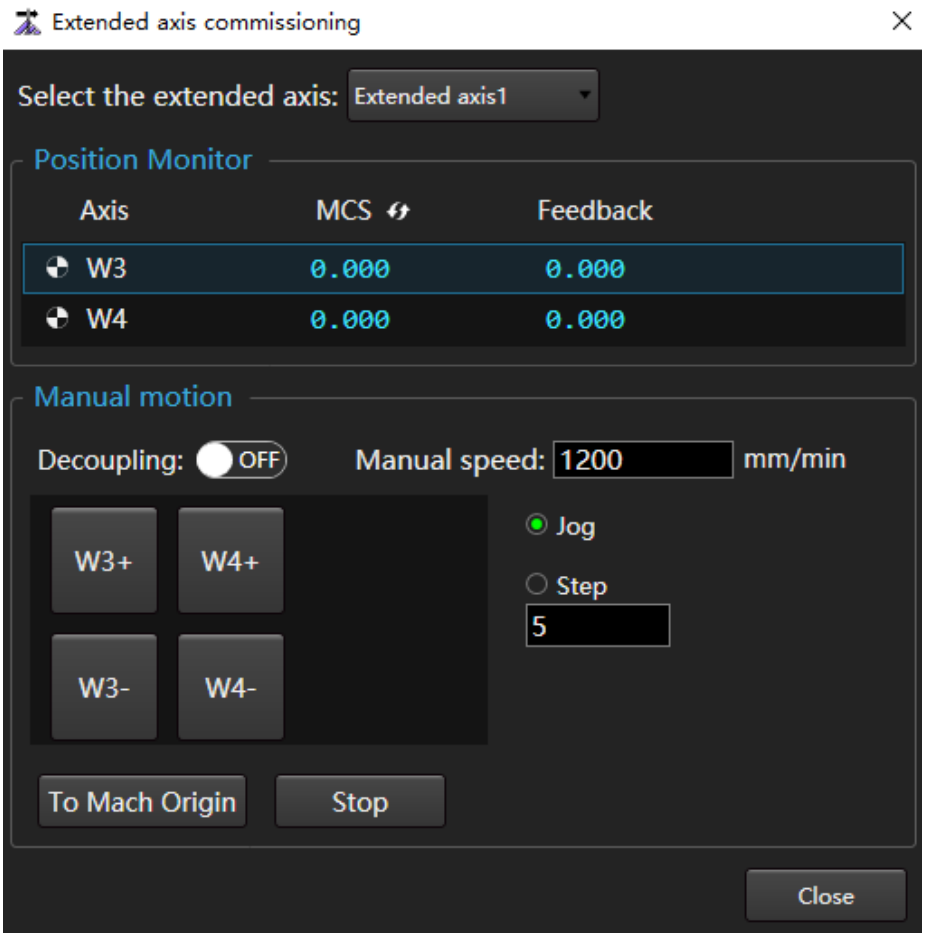
7.3.11. Extended Axis Commissioning

Adjust the movement and speed of the extension axis.

Operating Steps:

1. In the menu bar, click **Maintain** → 

to open the **Extended Axis Commissioning** dialog box:



2. In the **Select the Extended Axis** drop-down box, select the extension axis to debug.
3. In the **Manual Motion** area, set the speed and the movement mode of the extension axis.
4. Click **W1+**, **W1 -**, **To Mach Origin**, **Stop** according to the debugging requires, and judge whether the extension axis moves normally by observing the mechanical coordinate and actual axis movement in the **Position Monitor** area.

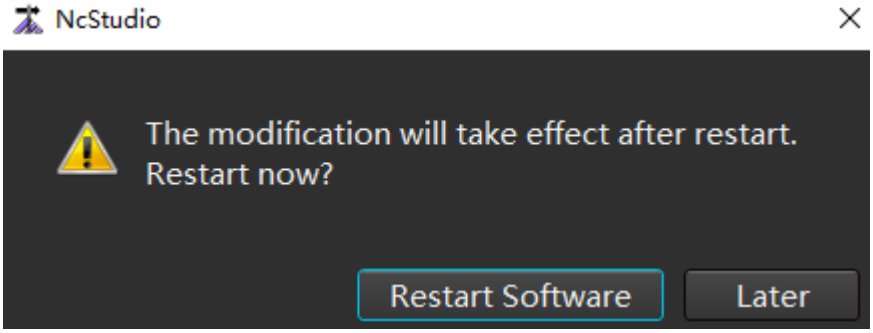
8. System Management

8.1. Switch Language

At present, the software supports Chinese and English.

Operating Steps：

1. In the menu bar, click **About** → 

to select the language to be switched, and the restart prompt box pops up：


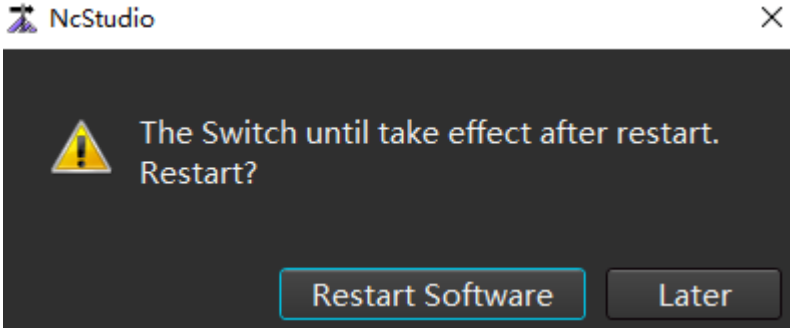
2. Click **Restart Software** to restart the software.

8.2. Switch Unit

Currently, the software supports metric and inch systems.

Operating Steps：

1. In the menu bar, click **About** → 

to select the unit to be switched, and the restart prompt box pops up：


2. Click **Restart Software**, and it will take effect after the software is restarted.

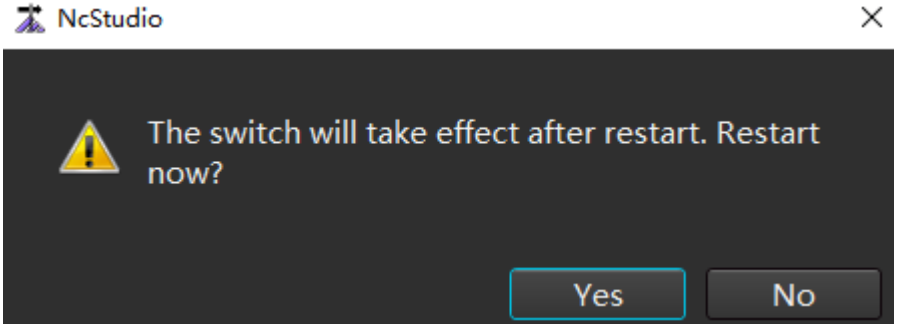
8.3. Switch Theme

Currently, the software supports white and black themes.

Operating Steps：

1. In the menu bar, click **About** → 

to select the theme to be switched, and the restart prompt box pops up:



2. Click **Restart Software**, and it will take effect after the software is restarted.

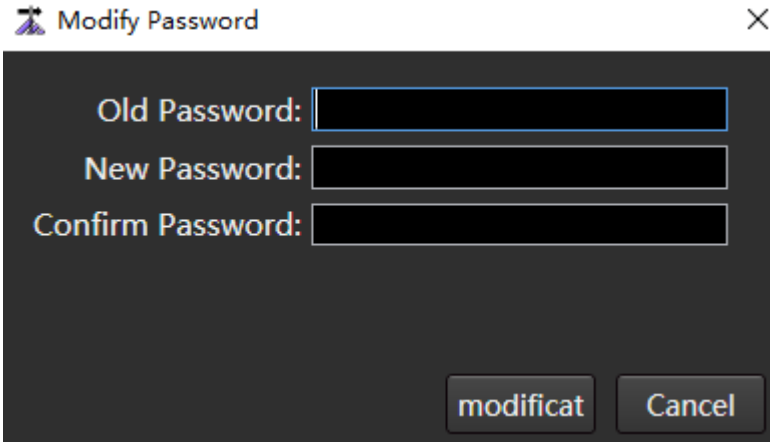
8.4. Modify Password

Modify the manufacturer password, which is only used to view and modify the manufacturer parameters.

Operating Steps：

1. In the menu bar, click **About** → 

to pops up the **Modify Password** dialog box:



2. Enter the old password, the new password set, and confirm the new password.
3. Click **Modification**.

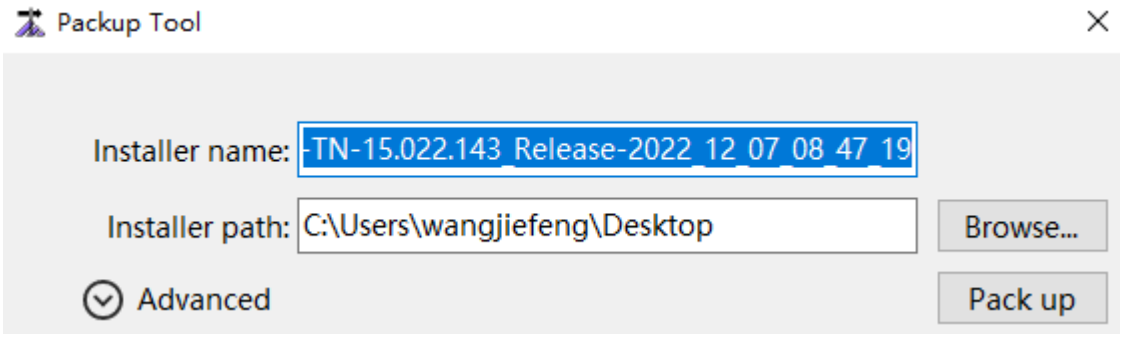
8.5. Installation Package

Generating a complete installation program in the current NC system is conducive to backing up the system files and saving the stable version of the system software.

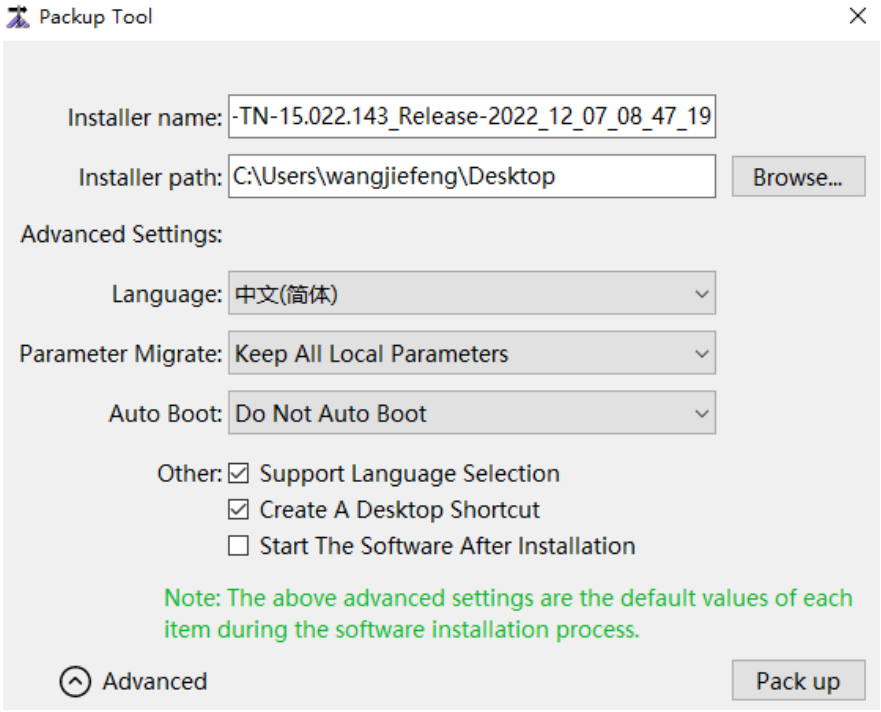
Operating Steps：

1. In the menu bar, click **About** → 

to open the **Packup Tool** dialog box:



2. Modify the installation package name and select the storage path of the installation package.
3. **（Optional：）** Click **Advanced** to set the following parameters:



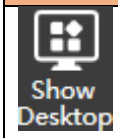
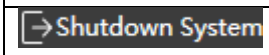
Parameter	Description
Language	Support Chinese and English.
Parameter migrate	▪ Keep all local parameters. ▪ Keep only machine-specific parameters. ▪ Fresh installation： Do not retain any parameters, and use the initial parameters of the software.
Auto boot	Whether to start the software automatically when starting up.
Support language selection	Whether Chinese or English is supported during installation.
Create a desktop shortcut	The computer desktop creates a shortcut icon for the installation package.
Start the software after installation	Start the software automatically after installation.

1. Click **Pack Up**.

After the installation package is created, view the generated installation package under the selected storage path.

8.6. System Button

Description:

Button	Description
	Minimize the system software interface and display the current computer desktop.
	Shut down the system software and start it again.
	Shut down the system software and the current computer.
	Turn off the system software, and start the computer after turning off the current computer.

8.7. Register Board

Register board to specifies the usage time of the system.

Before register board, make sure that the machine tool is in idle or emergency stop state.

Follow these steps to register board:

1. Get Register Code.
2. Use Duration of Register Board.

8.7.1. Get Register Code

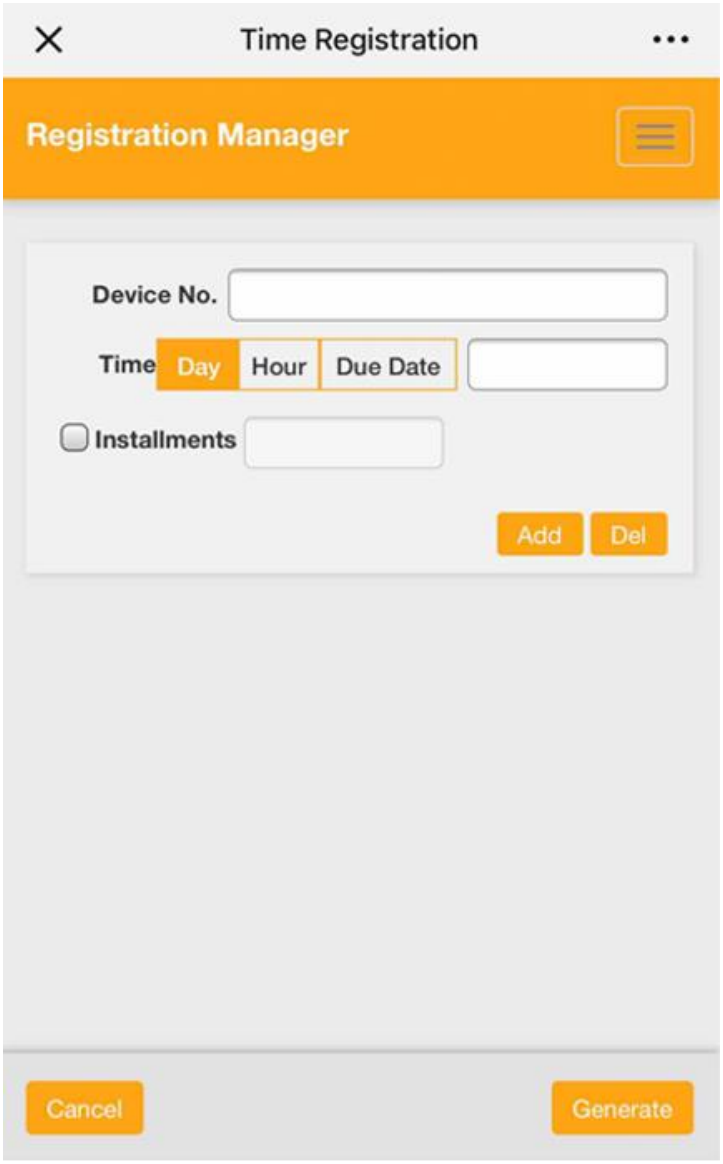
Operating Premise:

1. Get the account number and company information record.
   a. Select one of the following methods to get the account:
      - Contact local sales.
      - Call our customer service number: 400-882-9188.
   b. To put on records, fill in 《Registration Confirmation Letter》, seal and send it to Weihong company. Weihong company records the information in the confirmation letter you have returned.

Operating Steps:

1. In the menu bar, click **About** →  to open the **NcStudio** dialog box:

2. Record the device number.
   The device number changes with the number of registrations, which can be determined by the last three digits of the number.
   For example, when the number of registrations is 0, the last three digits are 000; When the number of registrations is 1, the last three digits are 001.
3. Scan the QR code at the lower left corner to enter **WEIHONG** WeChat official account.
4. Click **Service** → **Registration** → **Activate Account**, enter the mobile phone number, and get the temporary login password.
   The temporary login password is sent to the entered mobile phone number in the form of SMS. Please check the SMS.
5. Return to the login interface, enter the temporary login password to log in, fill in the information according to the following prompts, and get the register code:



Related Tasks：

To reset your password, click enter the account management interface to reset the password.

8.7.2. Use Duration of Register Board

Operating Premise：

Get Register Code.

Operating Steps：

1. In the menu bar, click **About** → About to open the **NcStudio** dialog box.
2. Click **Register**, and input the registration code.
3. Click **Confirm**.

After registration, restart the software to take effect.

When using the software later, you can view the remaining time of registration in the **NcStudio** dialog box.

9. Appendix

9.1. Shortcut Key List

When using the **NcStudio-V15 Laser Cutting Control System**, you can refer to the following list to get familiar with the shortcut keys of the software for easy operation.

Or get the shortcut key description on the system software: In the menu bar, click **About** → ShortCut Key Info.

Shortcut Key List

Shortcut Key	Function	Shortcut Key	Function
F1	Show shortcut keys	Ctrl + C	Copy
F2	E-stop	Ctrl+ V	Paste
F5	Set workpiece origin	Delete	Delete
F8	Simulate	Ctrl + Z	Undo
F12	Clear track	Ctrl + Y	Redo
Alt + 0	Ports setting	Ctrl + G	Gallery
Num+	Zoom in	Ctrl + T	Shape check
Num-	Zoom out	Ctrl + 1	Instant setting
Num*	Fit to window	Ctrl + 2	Layer setting
Ctrl + N	New	Ctrl + J	Combine
Ctrl + O	Open	Ctrl + W	Set lead-in/out line
Ctrl + S	Save	Ctrl + Q	Start cut point
Ctrl + I	Import	Ctrl + P	System parameters
Ctrl + A	Select all	Ctrl + D	Set machining direction
Ctrl + Shift + A	Select invert	Ctrl + E	Auto set machining order
Shift + A	Clear selected	Ctrl + R	Set kerf compensation
Ctrl + X	Cut	End	Middle current point

9.2. Description of Handle Keys

9.2.1. Weihong Professional Tube Cutting Wireless Handle

In the menu bar, click **About** → Handle Keys(_H) → **Tube**.

9.2.2. Weihong Sheet and Tube Wireless Handle

In the menu bar, click **About** → Handle Keys(_H) → **Sheet and Tube**.

WEIHONG

维宏股份 创业板:300508

Keys of WHB05L (V5) Wireless CNC Handwheel

Keys for Machining Control

- Start
- Stop
- Resume
- Emergency stop

Keys for Axis Direction

- Y moves in positive direction
- Y moves in negative direction
- Z moves in positive direction
- Z moves in negative direction
- X moves in positive direction
- X moves in negative direction
- B rotates in CW direction
- B rotates in CCW direction

Function Keys

- Height sensor
- Burst
- Laser ON at manual movement
- Return Z to machine origin
- Move backwards
- Blow
- Shutter
- Red light
- Return all to machine origin
- Move forwards
- Rapid
- Set workpiece origin
- Cut frame
- K1 Customize subroutine R1
- Step
- Return to workpiece origin
- Dry run
- K2 Customize subroutine R2

Combination Keys

- Aux + Z↕ Calibrate
- Aux + Simulate
- Aux + Leveling and centering
- Aux + Edge finding
- Aux + K1 Customize subroutine AuxR1
- Aux + ← → or Aux + B↺ B↻ X moves at rapid speed
- Aux + ↑ ↓ or Aux + B↺ B↻ Y moves at rapid speed
- Aux + Z↕ or Aux + B↺ B↻ Z moves at rapid speed
- Aux + B↺ B↻ or Aux + B↺ B↻ B moves at rapid speed
- Aux + Return to mark
- Aux + Set as mark
- Aux + Return B to machine origin
- Aux + Locate to last stop position
- Aux + K2 Customize subroutine AuxR2

Shanghai Weihong Electronic Technology Co., Ltd.
Address: No. 1590, Huhang Rd., Fengxian, Shanghai, China, 201401
Hot-line: 400 882 9188
e-mail: weihong@weihong.com.cn
Website: www.weihong.com.cn/en

Legal Notices

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

Requirements of Use

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

Intellectual Property Rights Notice

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

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

Termination

This license may be terminated by Weihong Company at any time once you violate any terms or conditions made in this agreement. Once the license is terminated, you are

obliged to destroy all the copies of the Software Product or return them to Weihong Company.

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

Weihong Electronic Technology Co., Ltd.