NcStudio Phoenix Stone Machining Center User Manual

Version: 2021.11.11 5th Version
Author: Product Testing Department

Preface

Thank you for using the NcStudio Phoenix Stone Machining Center software.

This manual will introduce details about the NcStudio Phoenix Stone Machining Center in aspects such as machine debugging, shape drawing, technique setting, machining operations, etc.

Please read this manual carefully before installing and using this product.

Our products are constantly being updated to provide better experience to our customers. Any document changes will not be informed separately. If you have any problems when using the product, or if you have any opinions or suggestions for our products and services, please contact us directly.

Contact Us

Shanghai Weihong Electronic Technology Co., Ltd.

Headquarters Address: 1590 Huhang Rd., Fengxian, Shanghai, China

Post Code: 201401

Telephone: 400-882-9188

Fax: 021-33587519

Email: weihong@weihong.com.cn

Official website: http://www.weihong.com.cn

Contents

1 Safety Precautions

Please read this manual, including the safety precautions, carefully and take necessary safety protection measures before using this product. If you have other requests, please contact us.

1.1 Precautions

Failure to obey the precautions may cause damage to human body, equipment or other properties.

Equipment running and debugging precautions

- Before running the equipment, ensure that the parameters are set correctly. Otherwise, the equipment may have unintentional movements.
- The value of the parameters must be within the range allowed by the machine and servo drive. If a parameter value is out of the limit, the system may have unstable performance or cause equipment damage.

1.2 Warning, Cautions, and Notes

Three types of prompts are used in this manual:

Warnings: Hazards or unsafe practices which could result in severe personal injury.

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

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

2 About the Product

2.1 Overview

The WEIHONG **NcStudio Phoenix Stone Machining Center** software is mainly used in machining of all types of high-class tabletop plates, wash basin plates and cupboard plates, engraving of man-made cupboard stones, quartzite tabletop and wash basins, and edging, independent drilling, sanding and chamfering for Western-style designs.

2.2 Functions

The **NcStudio Phoenix Stone Machining Center** software mainly supports the following functions:

- **Shapes**
  - Loading .ncex, .g code, .nc, .dxf, .dwg, .plt, and .eng files.
  - Drawing and editing shapes, and shape gallery.
  - Pre-processing shapes: single-item pre-processing and one-key pre-processing.
- **Machining techniques**
  - 7 types of techniques: **Pot hole**, **Front water**, **Back water**, **Outline**, **Hole opener**, **DrainTrough**, and **Custom**
  - At most 10 grinding steps: You can specify the tool No., machining speed, spindle rotational speed, plunging depth, and other parameters for each step.
  - The machining sequence, direction, lead line, deceleration point can be set to improve machining accuracy and efficiency.
- **Tool magazine and tools**
  - Support single linear tool magazine, dual linear tool magazine, and manual tool change.
  - Support tool and wear compensation based on the compensation settings.
  - Support 3 methods for setting the tool offset: direct input, manual setting of the Z-axis offset, and tool calibration.
- **Edge finding and centering**
  - Support automatic edge finding and centering for rectangular and oval shapes.
  - Support manual edge finding and centering for rectangular and oval shapes.
- **Machining methods**
  - Support multiple machining methods: machining wizard, machining for different layers, array and mirror machining, machining simulation, selective machining, and running machining commands.
  - Support machining statistics.

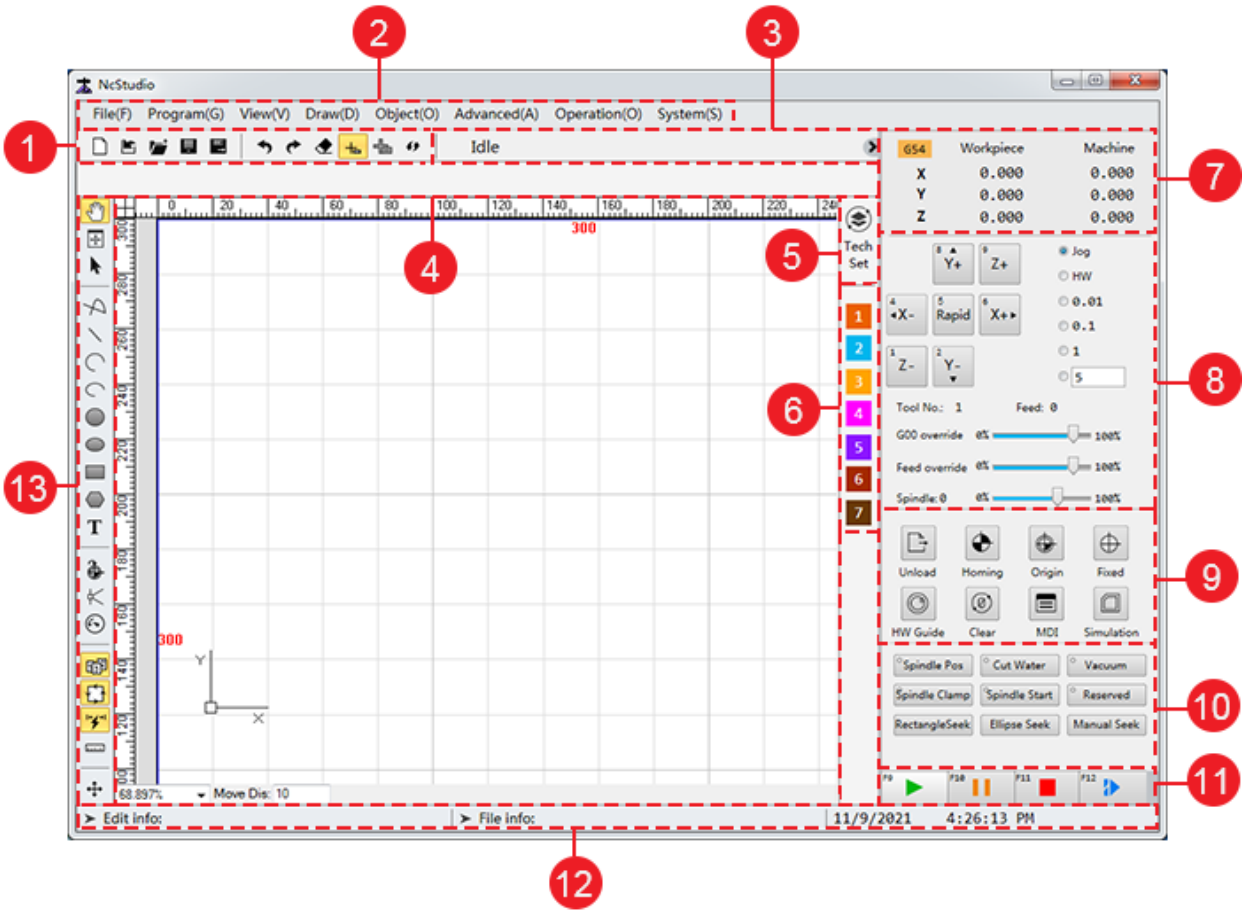
2.3 Features

- **Easy use**
  Simple UI design with integrated CAD, CAM, and CNC functions and shape gallery, drawing and parameter setting functions. User-friendly interface.
- **Stable performance**
  Advanced speed pre-processing, self-adaptive look-ahead, and track pre-processing algorithms to make the equipment move more rapid and stable.
- **High control accuracy**
  Professional algorithms are used to ensure machining efficiency, accuracy and flexibility.

3 Software Interface

This chapter will introduce the main interface of the **NcStudio Phoenix Stone Machining Center** software.

The software interface is shown below.

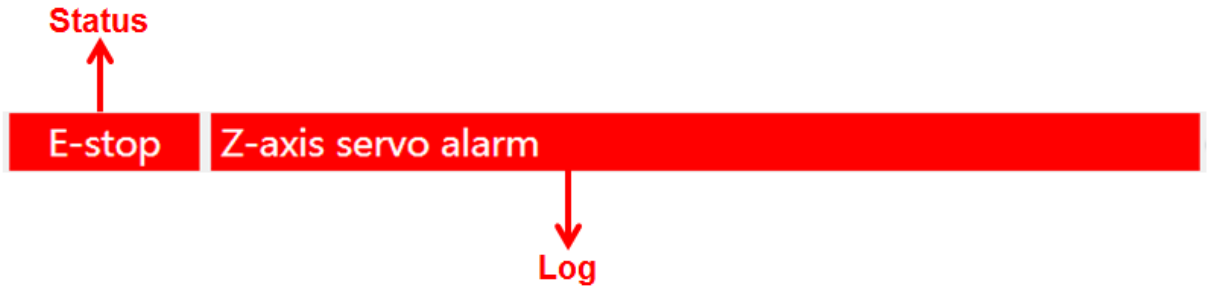


1. File button bar 2. Menu bar 3. Status bar 4. Drawing area 5. Tech Set button 6. Layers bar 7. Coordinate area 8. Machine control area 9. Operation button area 10. Port button area 11. Machining operation area 12. Information bar 13. Drawing tool bar

3.1 Status Bar

Shows the machine status and log.

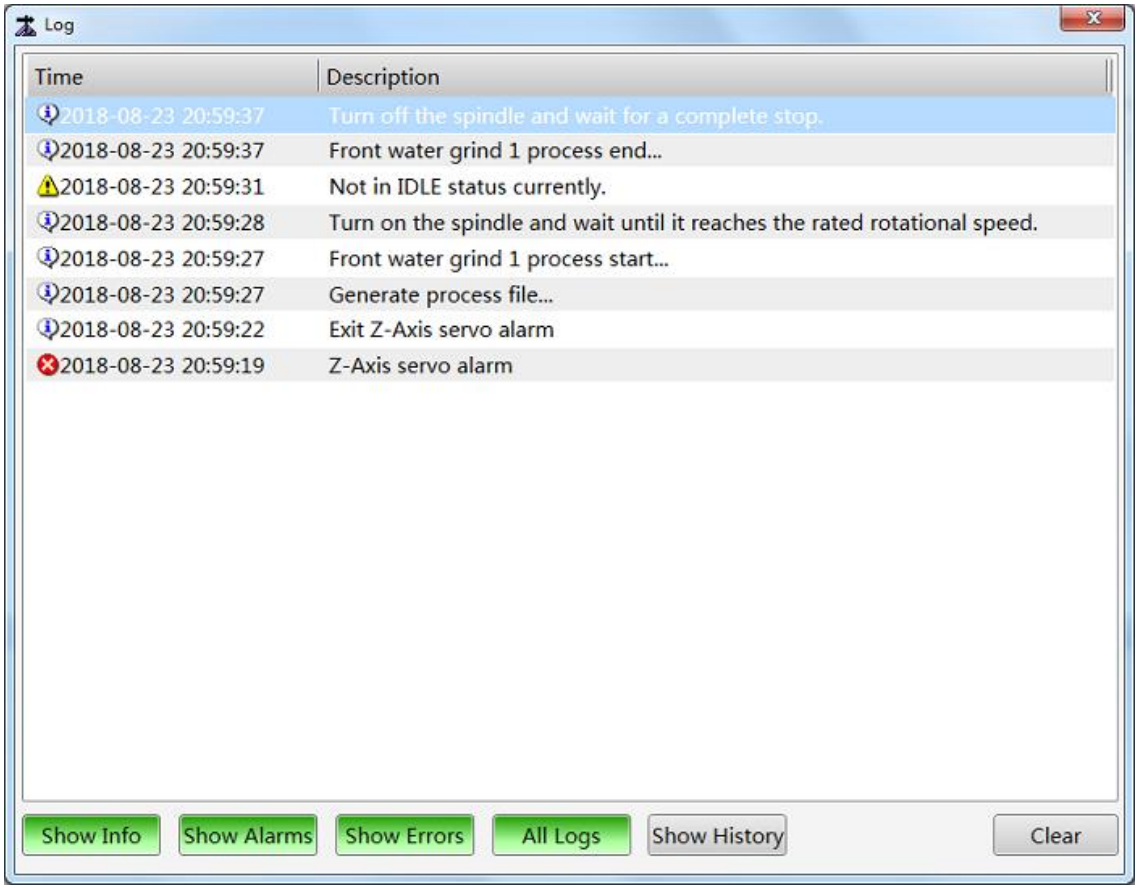
The following figure shows an example during emergency stop:



The four types of equipment status and descriptions are shown below:

Status	Description
Emergency stop	In this status, machining is stopped because events that prevent the machine from running normally or may damage the machine or operator have occurred.
Idle	In this status, there is no control input.
Running	In this status, the machine is running the commands sent from the software.
Paused	In this status, the machining is paused.

Click the log area, the **log** dialog box is displayed, which shows log details.



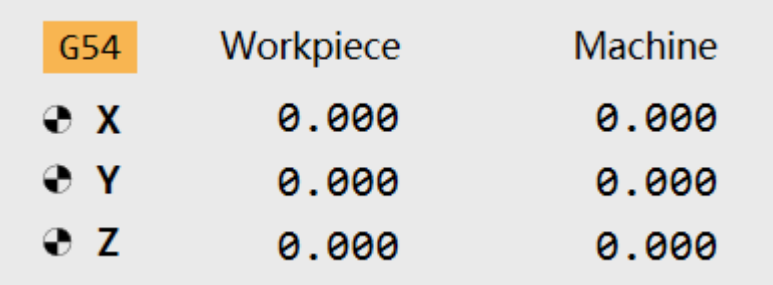
The system shows three types of logs. Their descriptions are shown below.

Log	Description
: Common information	Shows software actions.
: Warning	Suggesting potential errors. The equipment may still work in short time.
: Error	Suggesting operation errors. Machine needs to be stopped and the errors cleared before continuing machining.

3.2 Coordinate Area

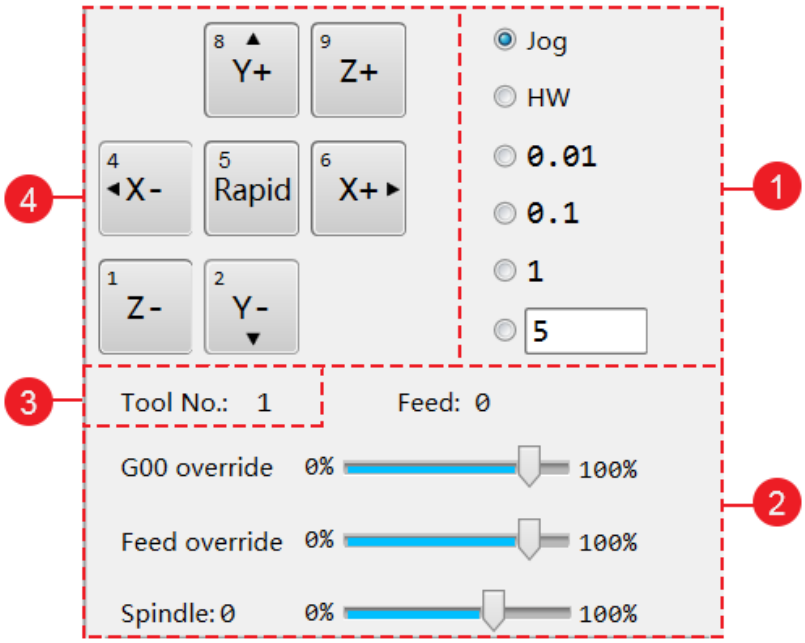
Shows the mechanical coordinate system and workpiece coordinate system.

After executing homing of an axis, the icon will be shown before the axis.



3.3 Machine Control Area

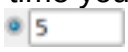
For manual control of machine movement and movement multiplier.



1. Feed mode 2. Multiplier area 3. Current tool number 4. Axis direction buttons

- **Feed mode**
  The system supports three feed modes:

Feed Mode	Description
Jog	In this mode, the target axis continues to move in the target direction until you release the axis direction button.
Step	In this mode, the target axis moves in the target direction by the

Feed Mode	Description
	step you selected (0.01/0.1/1) every time you click the axis direction button. Custom step: Click  and enter the desired step in the box. Default value: 5. Maximum value: 1000
Handwheel (HW)	In this mode, you control the movement with the handwheel.

- **Multiplier/Override control area**
  Used to adjust the actual machining speed.
  There are three override slides to control the G00 override, feed override, and spindle override.
  Actual feed rate = Feed rate setting * feed override
  Actual spindle rate = Spindle rate setting * spindle override
  You can use the following methods to adjust the override values:
  - Drag the corresponding slider.
  - Click the target value position on the bar.
  - Select the target bar and press the **PgUp** / **PgDn** key to increase/decrease the value.
  - Select the target bar and press the ↑ / ↓ / ← / → key to increase/decrease the value.
- **Axis direction buttons**
  Use the following methods to control the axes to move in the positive or negative direction:
  - Click an axis direction button to make the axis move at a default speed of 1800 mm/min.
  - Press down the **Rapid** button and then click an axis direction button, or press and hold the **5** key and click an axis direction button to make the axis move at a default speed of 2400 mm/min.

3.4 Operation Button Area

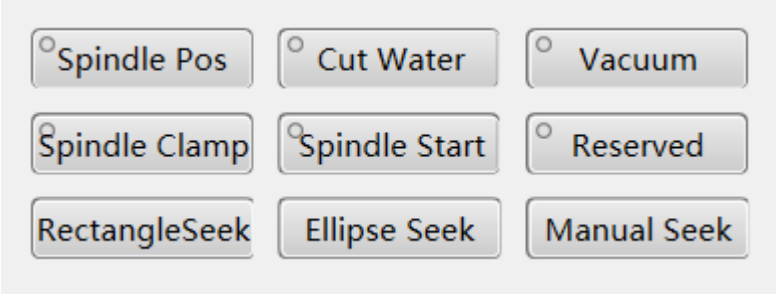
Convenient operation buttons for quick access.

Button	Description
 : Unload	Unload the current loaded program.
 : Go to mechanical origin	Make all axes go to the mechanical origin. For details, see Go to Mechanical Origin.
 : Go to workpiece origin	Make all axes go to the workpiece origin. For details, see Set Workpiece Origin.
 : Go to the fixed point	Make all axes go to the fixed point (fixed mechanical coordinates).

Button	Description
 : Handwheel guide	Enter HW mode.
 : Clear	Clear all coordinates.
 : MDI	At most 8 user commands can be entered and executed.
 : Simulation	Enter simulation mode.

3.5 Port Button Area

Shortcuts for controlling the opening/closing of some **I/O** ports.

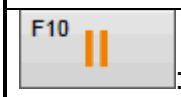
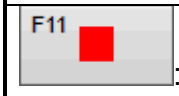
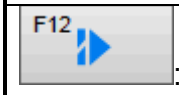


If a button is highlighted green, the port is open. Click 

to start current machining task.

3.6 Machining Operation Area

Buttons for starting or stopping machining.

Button	Description
 : Start	Start the current machining task. If you have drawn path in the software, or edited an opened/imported file, the system will start machining based on the technique and parameter settings. If a file is loaded by clicking **Open and Load** in the menu bar, the system will start machining based on the loaded file.
 : Pause	Makes the machine movement pause.
 : Stop	Stops the current machining task.
 : Resume	Resumes machining from the point stopped last time (prerequisite: the mechanical coordinates are correct).

Note

Save any changes before starting machining.

3.7 Tech Set button

Click the **Tech Set** button  to open the **Technics Parameter** window and set the technic type and other parameters.

For details about technic parameters, see Set Technic Parameters. For details about machining sequence, see Set Machining Direction.

The default technic type is **Custom**.

3.8 Layers bar

Used to select or modify a layer.

For details, see Layers.

If there are multiple shapes in the file, you can set them to be on different layers so that they can use different machining technics and shapes on the same layer can be machined separately.

Note

Only after a layer is selected for a shape, the layer technic parameters can be set.

3.9 Information Bar

Shows the currently loaded file name and the system time.

3.10 Drawing Tool Bar

Supports the following operations:

- Shape-related functions
- Setting technic parameters
- Setting edging finding and centering parameters

3.11 File Button Bar

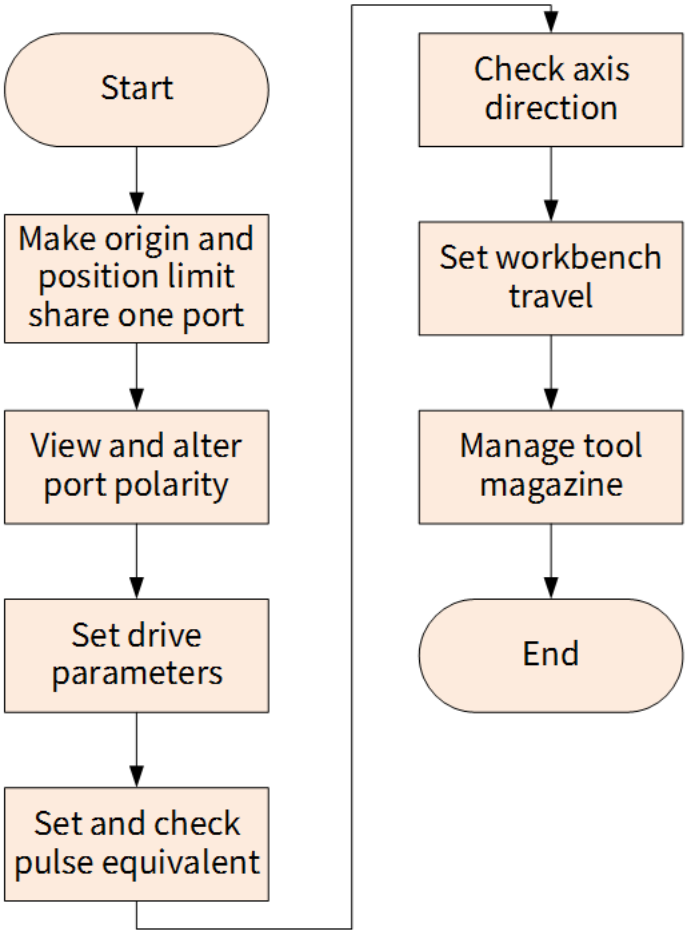
Shows common file-related buttons.

Button	Description
: New	Create a new file. After clicking this button, a prompt will be displayed, asking if you want to keep the current technic parameter settings: If you select **Yes**, the technic parameter settings on all layers will remain. If you select **No**, the technic parameter settings on all layers will be cleared.
: Import	.g code, .nc, .dxf, .plt, .dwg files can be imported.

Button	Description
: Open	.ncex files can be opened and loaded in the software. The shapes and parameter settings will be loaded in the software together.
: Save	Click this button to generate an .ncex file on the desktop and load it in the software. **Note:** Machining cannot be started until the **created**, **imported**, or **opened** file is saved.
: Save as	Click this button to save the opened/imported file as an .ncex file and set the file name and storage path.
: Undo	Undo the last step. Undo the last step to correct any shape drawing errors.
: Redo	Redo the last step to resume the state before undoing.
: Clear Track	Clear the machining tracks.
: Catch	Automatically catches feature points during drawing. For details, see Catch.
: Catch Option	Set the catch parameters.

4 Machine Debugging

After connecting the hardware and installing the software, you need to debug the system. The basic debugging flowchart is shown below:



- Make Origin and Position Limit Share One Port
- View and Alter Port Polarity
- Set Drive Parameters
- Set and Check Pulse Equivalent
- Check Axis Direction
- Set Workbench Travel Limits
- Manage Tool Magazine

Note

If the password is required during debugging, please contact the manufacturer.

4.1 Make Origin and Position Limit Share One Port

The origin switch and position limit switch need to share the same port based on the machine.

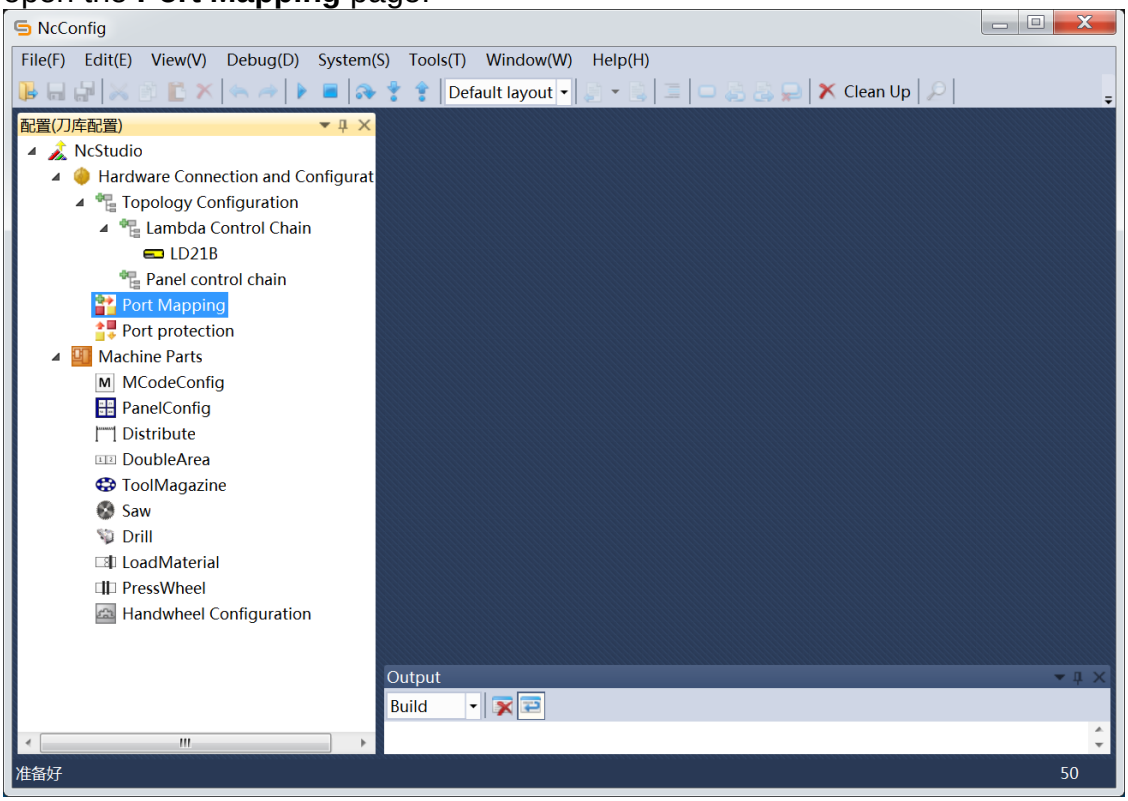
This section takes the case of the X-axis mechanical origin and X-axis positive limit sharing the same port as an example to introduce related steps:

Prerequisite:

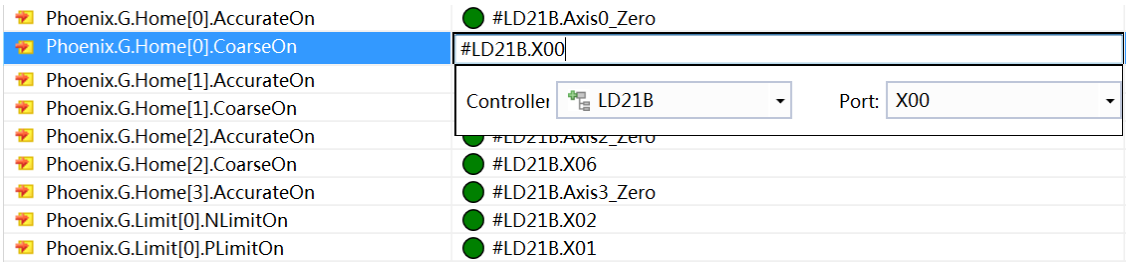
The **NcStudio Phoenix Stone Machining Center** software is closed.

Follow the steps below to make the two switches share the same port:

1. Open this path: **C:\Program Files\Weihong\NcStudio\NcConfig\Bin**. Find and double-click **NcConfig.exe** to open the NcConfig window.
2. Go to **View** > **Configuration**. In the left configuration menu, click **Port Mapping** to open the **Port Mapping** page:



3. On the **Port Mapping** screen, find the **Phoenix.G.Home.CoarseOn** and **Phoenix.G.Limit.PLimitOn** logical ports. Double-click their physical ports to show the selection box:



4. Click the **Controller** or **Port** drop-down menu and select corresponding options to make the physical ports of **Phoenix.G.Home.CoarseOn** and **Phoenix.G.Limit.PLimitOn** consistent.

5. Click  to save the setting.
6. To check the setting result: Open NcStudio. Go to **Operation** > **Port** to check the port list. In the figure below, the X-axis origin and X-axis positive limit share the same port:

4.2 View and Alter Port Polarity

After the software is started normally, modify the port polarity based on the detection switch and limit switch type:

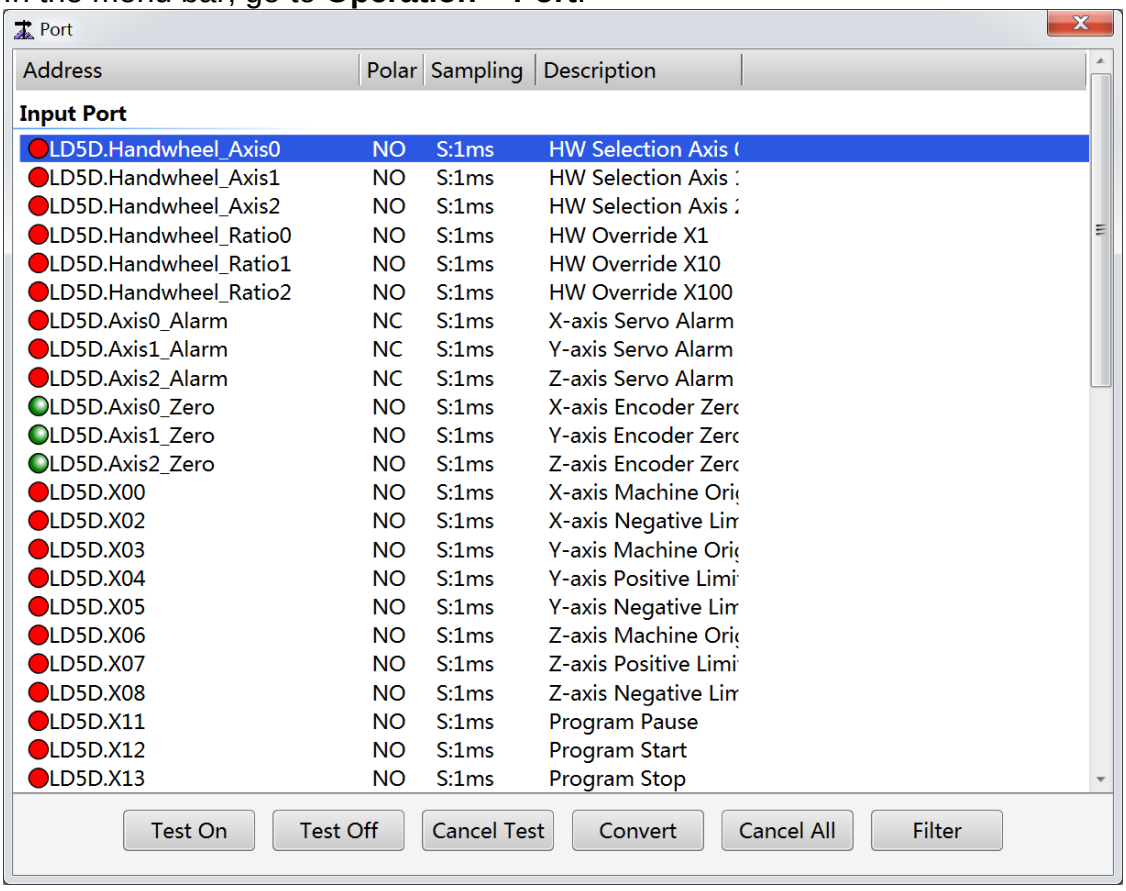
- Set the port polarity of normally closed switches to **NC**.
- Set the port polarity of normally open switches to **NO**.

The mapping between the port state and icons are shown below:

- Input port: No signal detected; Signal detected
- Output port: No signal detected; Signal detected

Follow the steps below to view and alter the port polarity:

1. In the menu bar, go to **Operation** > **Port**.



2. In the **Port** dialog box, select the target port and click the **Convert** button to set its polarity to **NC/NO**.
3. After the polarity of a port is modified, you can use the following functions based on your requirements:
   - To test if a port has output signals, click **Test On/Test Off**. Port icon or indicates that the port is under test.

- To cancel a port test, click **Cancel Test**.
- To cancel all port tests, click **Cancel All**.
- To set sampling interval, click **Filter**. In the **Sampling Setting** dialog box, drag the **Interval** slider. If the box before **Filter** is ticked, the system will rule out signals whose duration is shorter than the interval.

4.3 Set Drive Parameters

Basic drive parameters need to be set by referring to the user manual for that drive brand.

If you choose to use a drive brand other than those mentioned in this section, please note that:

- Check the servo drive SON signal type and ensure that it is active low (ON when connected to GND of a 24V power supply).
- Check the electrical level of the X/Y/Z-axis servo alarm output ports when there is no alarm:
  - Low level when there is no alarm: Set their polarity to **NC** (in the menu bar, go to **Operation** > **Port**).
  - Low level when there is alarm: Set their polarity to **NO**.
    For details about altering the port polarity, see View and Alter Port Polarity.
- Ensure that the servo drive is set to receive **pulses + direction** signals.
- Check to see if there is external emergency stop signal input among the servo drive input ports and check the input signal logic.
- The drive requires a 24V power, which flows to it via the terminal board. Before drive trial-run, connect the terminal board to a 24V power supply.
  If the drive does not work, check to see if the drive parameter **CW/CCW Input Inhibition** is set to **No**.

WEIHONG tested the following drive parameters in position loop control mode. The test results are as follows:

- WISE series drive
- Yaskawa Σ-Ⅱ Series Drive
- Yaskawa Σ-Ⅴ/Σ-7 Series Drive`
- Panasonic MINAS A4 Series Drive
- Panasonic MINAS A5 Series Drive
- Fuji FALDIC-β Series Drive
- Fuji ALPHA 5 Series Drive
- Delta ASDA-A Series Drive
- Delta ASDA-A2 Series Drive
- Delta ASDA-B Series Drive
- Delta ASDA-B2 Series Drive

4.3.1 WISE Series Drive

Pr001 Control mode selection

1: Position loop control mode

2: Speed loop control mode

Value: 1

Pr528 LED initial state

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 6

Pr008 Number of command pulses per turn

If **Pr008** is set to 0, **Pr009** and **Pr010** are effective.

If **Pr008** value is not 0, **Pr008 = screw pitch/(pulse equivalent × mechanical reduction ratio)**

Value: 0

Pr009 1st command division and multiplication numeratorPr010 command division and multiplication denominator

Range: 0–2^{30}

Typical value: When screw pitch = 5 mm, encoder resolution = 10000, axis coupling joint is used, and pulse equivalent = 0.001 mm: Pr009＝10000, Pr010＝Screw pitch/pulse equivalent＝5 mm/0.001 mm = 5000, which means that Pr009/Pr010=10000/5000=2/1。

Value: manual calculation

Pr011 Number of pulse output by one motor turn

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm: Pr011=2500; when screw pitch = 5 mm: Pr011=1250.

Value: 2500 (by default)

Pr100 1st position loop gain

Unit: 0.1/s

Value: 480 (by default). May need adjustment based on machine condition.

Pr101 1st velocity loop gain

Unit: 0.1Hz

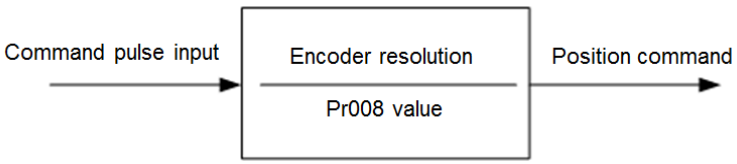
Value: 270 (by default). May need adjustment based on machine condition.

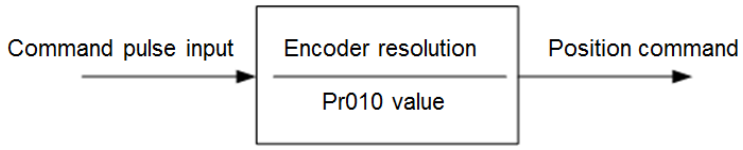
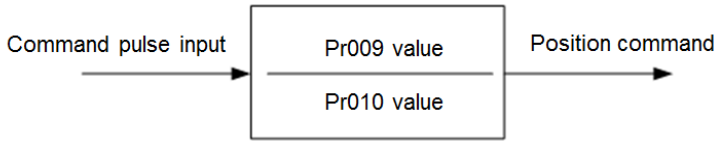
Pr102 1st velocity loop integral time constant

Unit: 0.1ms

Value: 210 (by default). May need adjustment based on machine condition.

Relation between Pr008, Pr009, and Pr010 values

Pr008	Pr009	Pr010	Description
1–2^{20}	Invalid	Invalid	 Pr009 and Pr010 values are not valid. System processing is based on the Pr008 value.

Pr008	Pr009	Pr010	Description
0	0	1–2^{30}	 If Pr008 and Pr009 are 0, system processing is based on the Pr010 value.
0	1–2^{30}	1–2^{30}	 If Pr008 is 0 but Pr009 is not 0, system processing is based on the Pr009 and Pr010 values.

4.3.2 Yaskawa Σ-Ⅱ Series Drive

Fn010 Password setting (preventing random parameter modification)

If Fn010 is set to 0000, modification of user parameters (PnXXX) and some auxiliary function parameters (FnXXX) is allowed.

If Fn010 is set to 0001, modification of user parameters (PnXXX) and some auxiliary function parameters (FnXXX) is not allowed.

Value: 0000

Un00C Input command pulse counter

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: In hexadecimal, low-order (L) 4 digits

Pn000 Rotation direction and control mode selection

Digit 0: If digit 0 is set to 0, the motor rotates CW (anti-clockwise when observing from the load/lead screw side). If digit 0 is set to 1, the motor rotates CCW.

Digit 1: If digit 1 is set to 1, the control mode is position control. Always calculates pulse commands.

Value: 0010

Pn200 Pulse command format selection

Digit 0: If digit 0 is set to 5, the command is pulse + direction, negative logic.

Digit 3: If digit 3 is set to 0, the differential signal goes into the filter.

Value: 0005

Pn50A Function selection

Digit 1: If digit 1 is set to 0, the /S-ON signal is enabled and its input pin is No.40. If digit 1 is set to 7, the /S-ON signal is always ON.

Digit 3: If digit 3 is set to 8, the CW input inhibition signal P-OT will not be used.

Value: 8100

Pn50B Function selection

Digit 0: If digit 0 is set to 8, the CCW input inhibition signal N-OT will not be used.

Value: 6548

Pn50F Function selection

Applicable when the servo motor has a brake.

Digit 2: If digit 2 is set to 3, CN1-29 and 30 output brake interlocking signal/BK to control the 24V relay for brake.

Value: 0300

Pn50E Function selection

Applicable when the servo motor has a brake.

None of the four digits can be set to 3 in case that CN1-29 and CN1-30 are used for other functions, causing braking failure.

Value: 0211

Pn506 Brake delay when servo motor is off

Unit: 10ms

Applicable when the servo motor has a brake.

Value: Subject to actual situation.

Pn201 PG division ratio setting

Range: $16–2^{14}$

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, Pn201 = 2500; when screw pitch = 5 mm, Pn201 = 1250

Value: Based on the PG division ratio (WEIHONG systems)

Pn202 Electronic gear ratio numerator, Pn203 Electronic gear ratio denominator

Equation:

$$\text{Pn202} = \text{Pulse number per encoder turn} \times 4 \times \text{Mechanical deceleration rate}$$
$$\text{Pn203} = \text{Screw pitch} / \text{Pulse equivalent}$$

Typical value: When screw pitch = 5 mm, the encoder is 17 digits, axis coupling joint is used, pulse equivalent = 0.001 mm: Pn202＝16384 and Pn203＝625 When screw pitch = 5 mm, the encoder is 17 digits, axis coupling joint is used, pulse equivalent = 0.0005 mm: Pn202＝8192 and Pn203＝625

Value: Manual calculation

4.3.3 Yaskawa Σ-Ⅴ/Σ-7 Series Drive

Fn010 Parameter input inhibition setting

If Fn010 is set to 0000, modification of user parameters (PnXXX) and some auxiliary function parameters (FnXXX) is allowed.

If Fn010 is set to 0001, modification of user parameters (PnXXX) and some auxiliary function parameters (FnXXX) is not allowed.

Value: 0000

Pn000 Function selection basic switch 0

Digit 0: If digit 0 is set to 0, the motor rotates CW after receiving CW commands.

Digit 1: If digit 1 is set to 1, the control mode is position control (pulse sequence commands).

Value: 0010

Pn200 Position control command format selection switch

Digit 0: If digit 0 is set to 5, the command format is pulse + direction, negative logic.

Value: 0005

Pn50A Input signal selection 1

Digit 1: If digit 1 is set to 0, /S-ON signal is enabled and its input pin is No.40. If digit 1 is set to 7, the servo drive is always ON.

Digit 3: If digit 3 is set to 8, the CW input inhibition signal P-OT will not be used.

Value: 8100

Pn50B Input signal selection 2

Digit 0: If digit 0 is set to 8, the CCW input inhibition signal N-OT will not be used.

Value: 6548

Pn50F Output signal selection 2

Applicable when the servo motor has a brake.

Digit 2: If digit 2 is set to 3, CN1-29 and 30 output brake interlocking signal/BK to control the 24V relay for brake.

Value: 0300

Pn50E Output signal selection 1

Applicable when the servo motor has a brake.

None of the four digits can be set to 3 in case that CN1-29 and CN1-30 are used for other functions, causing braking failure.

Value: 0211

Pn506 Brake command: Servo OFF delay

Applicable when the servo motor has a brake.

Unit: ms

Value: Subject to actual situation.

Pn20E Electronic gear ratio numerator, Pn210 Electronic gear ratio denominator

Equation: **Pn20E/Pn210 = (Encoder resolution × pulse equivalent × mechanical reduction ratio)/screw pitch**

Value: Manual calculation

Pn212 Encoder allocated pulse number

Range: 16–2^{30}

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, Pn212 = 2500; when screw pitch = 5 mm, Pn212 = 1250

Value: Based on the PG division ratio (WEIHONG systems)

4.3.4 Panasonic MINAS A4 Series Drive

Pr01 LED initial state

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 12

Pr02 Control mode selection

0: Position control

1: Speed control

2: Torque control

Value: 0

Pr40 Command pulse input selection

1: Input via dedicated differential circuit.

Value: 1

Pr42 Command pulse input format selection

3: The command pulse input format is pulse + direction, negative logic

Value: 3

Pr44 Feedback pulse division and multiplication numerator

Range: 1–32767

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, Pr44 = 2500; when screw pitch = 5 mm, Pr44 = 1250

Value: Based on the PG division ratio (WEIHONG systems)

Pr48 Command pulse division and multiplication 1st numerator, Pr4B Command pulse division and multiplication denominator

Range: 1–10000

Typical value: When screw pitch = 5 mm, the encoder resolution is 10000, axis coupling joint is used, and pulse equivalent = 0.001 mm: Pr48= 10000, Pr4B＝ screw pitch/pulse equivalent = 5/0.001=5000. Pr48/Pr4B=10000/5000=2/1.

Value: Manual calculation

4.3.5 Panasonic MINAS A5 Series Drive

Pr5.28 LED initial state

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 6

Pr0.01 Control mode setting

0: Position control

1: Speed control

2: Torque control

Value: 0

Pr0.05 Command pulse input selection

0: Opto-electronic coupler input (PULS1, PULS2, SIGN1, SIGN2)

1: Dedicated line drive input (PULSH1, PULSH2, SIGNH1, SIGNH2)

Value: 1 (in common cases)

Pr0.07 Command pulse input format selection

3: The command pulse input format is pulse + direction, negative logic

Value: 3

Pr0.08 Number of command pulses per turn

If Pr0.08 is set to 0, Pr0.09 and Pr0.10 are valid.

If Pr0.08 is not 0, **Pr0.08 = screw pitch/(pulse equivalent × mechanical reduction ratio)**

Value: 0

Pr0.09 1st command division and multiplication numerator, Pr0.10 Command pulse division and multiplication denominator

Range: $0–2^{30}$

Typical value: When screw pitch = 5 mm, the encoder resolution is 10000, axis coupling joint is used, and pulse equivalent = 0.001 mm: Pr0.09＝10000 and Pr0.10＝screw pitch/pulse equivalent=5/0.001＝5000. Pr0.09/Pr0.10=10000/5000=2/1.

Value: Manual calculation

Pr0.11 Number of pulse output by one motor turn

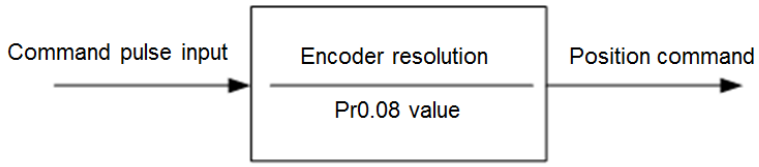
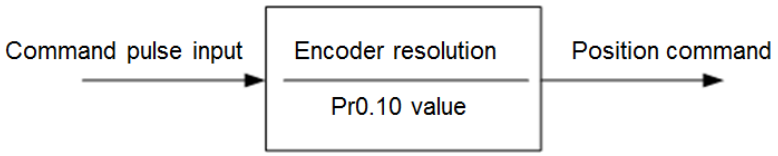
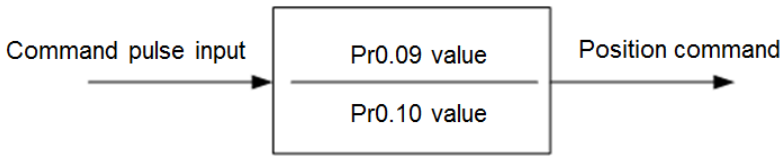
Range: 1–262144

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, Pr0.11 = 2500; when screw pitch = 5 mm, Pr0.11 = 1250

Value: Based on the PG division ratio (WEIHONG systems)

Relation between Pr0.08, Pr0.09, and Pr0.10 values

Pr0.08	Pr0.09	Pr0.10	Description

Pr0.08	Pr0.09	Pr0.10	Description
1–2^{20}	Invalid	Invalid	 Pr0.09 and Pr0.10 values are not valid. System processing is based on the Pr0.08 value.
0	0	1–2^{30}	 If Pr0.08 and Pr0.09 are 0, system processing is based on the Pr0.10 value.
0	1–2^{30}	1–2^{30}	 If Pr0.08 is 0 but Pr0.09 is not 0, system processing is based on the Pr0.09 and Pr0.10 values.

4.3.6 Fuji FALDIC-β Series Drive

01 Command pulse numerator α, 02 Command pulse denominator β

Stand for electronic gear ratio numerator and denominator.

Range: 1–32767

Equation: **α/β = (encoder resolution × pulse equivalent × mechanical reduction ratio)/screw pitch**

Typical value: When encoder resolution = 65536, pulse equivalent = 0.001, screw pitch = 5 mm, and mechanical reduction ratio = 1: α/β=65536×0.001/5=8192/625, which means α=8192 and β=625.

Value: Manual calculation

03 Pulse train input format

0: The pulse train input format is pulse + direction (symbol).

Value: 0

04 Rotation direction

0: Rotates CW (anti-clockwise when observing from the load side)

1: Rotates CCW (clockwise when observing from the load side)

Value: 0 or 1

10 CONT1 signal distribution

1: CONT1 is distributed to RUN (SON). If CONT1 is not distributed, it becomes ON if there is no alarm after power-on.

Value: 1

11 CONT2 signal distribution

2: CONT2 is distributed to RST (CLR: servo alarm clearing). Parameter 12, 13, and 14 are 0, which means that CONT3, CONT4, and CONT5 cannot be distributed to OT (overtravel) or EMG (external emergency stop).

Value: 2

15 OUT1 signal distribution

1: OUT1 is distributed to alarm output contact a, which is normally open.

2: OUT1 is distributed to alarm output contact b, which is normally closed.

Value: 1

27 Parameter modification inhibition

0: Drive parameter values can be modified.

1: Drive parameter values cannot be modified.

Value: 0 or 1

74 CONT 1 constant validity 1

1: Valid upon servo motor start (RUN).

Value: 1

4.3.7 Fuji ALPHA 5 Series Drive

PA1_01 Control mode selection

0: Position control

1: Speed control

Value: 0

PA1_06 Electronic gear ratio numerator 0, PA1_07 Electronic gear ratio denominator

Range: 1–32767

Equation: **PA1_06/PA1_07 =(Encoder resolution × pulse equivalent × mechanical reduction ratio)/screw pitch**

Typical value: When encoder resolution = 65536, pulse equivalent = 0.001, screw pitch = 5 mm, and mechanical reduction ratio = 1: PA1_06/PA1_07 =65536×0.001/5=8192/625. Therefore, PA1_06 = 8192 and PA1_07 = 625.

Value: Manual calculation

PA1_03 Command pulse format

0: The pulse train input format is pulse + direction (symbol).

Value: 0

PA1_04 Rotation direction

0: Rotates CW (anti-clockwise when observing from the load side)

1: Rotates CCW (clockwise when observing from the load side)

Value: 0 or 1

PA3_01 CONT1 signal distribution

1: CONT1 is distributed to RUN (SON). If CONT1 is not distributed, it becomes ON if there is no alarm after power-on.

Value: 1

PA3_02 CONT2 signal distribution

2: CONT2 is distributed to RST (CLR: servo alarm clearing). Parameter 12, 13, and 14 are 0, which means that CONT3, CONT4, and CONT5 cannot be distributed to OT (overtravel) or EMG (external emergency stop).

Value: 2

PA3_51 OUT1 signal distribution

1: OUT1 is distributed to alarm output contact a, which is normally open.

2: OUT1 is distributed to alarm output contact b, which is normally closed.

Value: 1

PA3_26 CONT 1 constant validity 1

1: Valid upon servo motor start (RUN).

Value: 1

PA1_08 Number of pulse output by one motor turn

Range: 16–214

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, PA1_08 = 2500; when screw pitch = 5 mm, PA1_08 = 1250

Value: Based on the PG division ratio (WEIHONG systems)

4.3.8 Delta ASDA-A Series Drive

P0-02 Drive status display

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 02

P1-00 External pulse input format

Format: ZYX

X=2: The external pulse input format is pulse + direction

Z=1: negative logic

Value: 102

P1-01 Control mode setting

Format: ZYX1X0

Z=0: DIO value does not change when the control mode is switched. Control mode was not switched; therefore, Z=0.

Y=0: Rotates CW (anti-clockwise when observing from the load side). Y=1: Rotates CCW.

X1X0=00: The control mode is position control.

Value: 0000

P1-32 Motor stopping mode

Format: YX

Y=0: When the servo motor is disabled, dynamic braking is used. Y=1: When the servo motor is disabled, the motor moves freely.

X=0: The motor is stopped instantly. X=1: The motor decelerates before stops completely.

Value: 00

P1-44 Electronic gear ratio numerator N1, P1-45 Electronic gear ratio denominator M

Range: 1–32767

Equation: **N1/M =(Encoder pulse number × 4 × pulse equivalent × mechanical reduction ratio)/screw pitch**.

Typical value: When the encoder pulse number = 2500, pulse equivalent = 0.001, screw pitch = 5 mm, and mechanical reduction ratio = 1, N1/M=2500×4×0.001/5=2/1. Therefore, N1=2 and M=1.

Multi-segment electronic gear ratio was not used. Therefore, there was no need to set P2-60–P2-62.

Value: Manual calculation

P2-10 Digital input pin DI1 function setting

Format: X2X1X0

X1X0=01: Set digital input DI1 to SON, matching pin No.9 of CN1.

X2=1: Set input DI1 to the normally open contact a.

Value: 101

P2-15 Digital input pin DI6 function setting, P2-16 Digital input pin DI7 function setting

DI6 are DI7 are NC position limit signal input by default. The drive cannot work before the CN1 pin No.32 and pin No.31 are connected.

Format: P2-15=P2-16=X2X1X0

X2=1: Set DI6 and DI7 input to the NO contact a.

X1X0=00: Drive position limit input was not used.

Value: P2-15=P2-16=100

P2-17 Digital input pin DI8 function setting

Format: X2X1X0

X2X1X0=100: External EMG (emergency stop input) was not used.

Value: 100

P2-21 Digital output pin DO4 function setting

DO4 pins are pin No.1 and No.26, which are used for Z-axis clamping position braking signals.

Format: X2X1X0

X2=1: Set DO4 output to the NO contact a. X2=1: Set DO4 output to the NC contact b.

X1X0=08: Set pin No.1 and No.26 to BK+ and BK- respectively.

Value: 108

P2-22 Digital output pin DO5 function setting

DO5 pins are pin No.28 and No.27, which are used for servo alarm signals.

Format: X2X1X0

X2=0: Set DO5 output to the NC contact b.

X1X0=07: Set pin No.28 and No.27 to ALRM+ and ALRM- respectively.

Value: 007

P2-51 Servo enablement SON setting

0: Servo motor enablement must be triggered by digital signals.

1: Servo motor is automatically enabled after powered on if there is no alarm.

Value: 0 (1 when SON signal cables are unavailable)

4.3.9 Delta ASDA-A2 Series Drive

P0-02 Drive status display

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 02

P1-00 External pulse input format

Format: ZYX

X=2: The external pulse input format is pulse + direction

Z=1: negative logic

Value: 102

P1-01 Control mode setting

Format: ZYX1X0

Z=0: DIO value does not change when the control mode is switched. Control mode was not switched; therefore, Z=0.

Y=0: Rotates CW (anti-clockwise when observing from the load side). Y=1: Rotates CCW.

X1X0=00: The control mode is position control.

Value: 0000

P1-44 Electronic gear ratio numerator N1, P1-45 Electronic gear ratio denominator M

Range: 1–32767

Equation: **P1-44/P1-45 = (Encoder resolution × pulse equivalent × mechanical reduction ratio)/screw pitch**.

Typical value: When the encoder pulse number = 2500, pulse equivalent = 0.001, screw pitch = 5 mm, and mechanical reduction ratio = 1, N1/M=2500×4×0.001/5=2/1. Therefore, N1=2 and M=1. Multi-segment electronic gear ratio was not used. Therefore, there was no need to set P2-60–P2-62.

Value: Manual calculation

P1-46 Detector pulse output number setting

Setting of the revolving single-direction pulse number.

Range: 20–320000

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, P1-46 = 10000; when screw pitch = 5 mm, P1-46 = 5000

Value: Based on the PG division ratio (WEIHONG systems)

P2-10 Digital input pin DI1 function setting

Format: X2X1X0

X1X0=01: Set digital input DI1 to SON, matching pin No.9 of CN1.

X2=1: Set input DI1 to the normally open contact a.

Value: 101

P2-15 Digital input pin DI6 function setting

DI6 are DI7 are NC position limit signal input by default. The drive cannot work before the CN1 pin No.32 and pin No.31 are connected.

Format: X2X1X0

X2=1: Set DI6 and DI7 input to the NO contact a.

X1X0=00: Drive position limit input was not used.

Value: 100

P2-16 Digital input pin DI7 function setting

Format: X2X1X0

Value: 100

P2-17 Digital input pin DI8 function setting

Format: X2X1X0

X2X1X0=100: External EMG (emergency stop input) was not used.

Value: 100

P2-21 Digital output pin DO4 function setting

DO4 pins are pin No.1 and No.26, which are used for Z-axis clamping position braking signals.

Format: X2X1X0

X2=1: Set DO4 output to the NO contact a. X2=1: Set DO4 output to the NC contact b.

X1X0=08: Set pin No.1 and No.26 to BK+ and BK- respectively.

Value: 108

P2-22 Digital output pin DO5 function setting

DO5 pins are pin No.28 and No.27, which are used for servo alarm signals.

Format: X2X1X0

X2=0: Set DO5 output to the NC contact b.

X1X0=07: Set pin No.28 and No.27 to ALRM+ and ALRM- respectively.

Value: 007

4.3.10 Delta ASDA-B Series Drive

P0-02 Drive status display

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 02

P1-00 External pulse train input format

Format: ZYX

X=2: The external pulse input format is pulse + direction

Z=1: negative logic

Value: 102

P1-01 Control mode setting

Format: YX1X0

Y=0: Rotates CW (anti-clockwise when observing from the load side). Y=1: Rotates CCW.

X1X0=00: The control mode is position control.

Value: 000

P1-32 Motor stopping mode

Format: YX

Y=0: When the servo motor is disabled, dynamic braking is used. Y=1: When the servo motor is disabled, the motor moves freely.

X=0: The motor is stopped instantly. X=1: The motor decelerates before stops completely.

Value: 00

P1-44 Electronic gear ratio numerator N1, P1-45 Electronic gear ratio denominator M

Range: 1–32767

Equation: **P1-44/P1-45 = (Encoder resolution × pulse equivalent × mechanical reduction ratio)/screw pitch**.

Typical value: When the encoder pulse number = 2500, pulse equivalent = 0.001, screw pitch = 5 mm, and mechanical reduction ratio = 1, N1/M=2500×4×0.001/5=2/1. Therefore, N1=2 and M=1.

Multi-segment electronic gear ratio was not used. Therefore, there was no need to set P2-60–P2-62.

Value: Manual calculation

P2-10 Digital input pin DI1 function setting

Format: X2X1X0X1X0=01: Set digital input DI1 to SON, matching pin No.17 of CN1.X2=1: Set input DI1 to the normally open contact a.

Value: 101

P2-15 Digital input pin DI6 function setting

DI6 is NC position limit signal input by default. The drive cannot work before the CN1 pin No.32 and pin No.31 are connected.

Format: X2X1X0

X2=1: Set input DI6 to the normally open contact a.

X1X0=00: Drive position limit input was not used.

Value: 100

P2-18 Digital output pin DO1 function setting

DO1 pin is pin No.16, which is used for Z-axis clamping position braking signals.

Format: X2X1X0

X2=1: Set DO1 output to the NO contact a. X2=1: Set DO4 output to the NC contact b.

X1X0=08: Set pin No.16 to BK+.

Value: 108

P2-20 Digital output pin DO3 function setting

DO3 pin is pin No.1, which is used for servo alarm signals.

Format: 2X1X0

X2=0: Set DO3 output to the NC contact b.

X1X0=07: Set pin No.1 to ALRM+.

Value: 007

4.3.11 Delta ASDA-B2 Series Drive

P0-02 Drive status display

Used to monitor if the number of pulses sent and received are correct.

In a WEIHONG control system, pulse monitor can be used to check if the communication card has sent pulses correctly and determine if there is electrical interference.

Value: 02

P1-00 External pulse train input format

Format: ZYX

X=2: The external pulse input format is pulse + direction

Z=1: negative logic

Value: 102

P1-01 Control mode setting

Format: ZYX1X0

Z=0: DIO value does not change when the control mode is switched. Control mode was not switched; therefore, Z=0.

Y=0: Rotates CW (anti-clockwise when observing from the load side). Y=1: Rotates CCW.

X1X0=00: The control mode is position control.

Value: 0000

P1-44 Electronic gear ratio numerator N1, P1-45 Electronic gear ratio denominator M

Range: 1–32767

Equation: **P1-44/P1-45 = (Encoder resolution × pulse equivalent × mechanical reduction ratio)/screw pitch**.

Typical value: When the encoder pulse number = 2500, pulse equivalent = 0.001, screw pitch = 5 mm, and mechanical reduction ratio = 1, N1/M=2500×4×0.001/5=2/1. Therefore, N1=2 and M=1.

Multi-segment electronic gear ratio was not used. Therefore, there was no need to set P2-60–P2-62.

Value: Manual calculation

P1-46 Detector pulse output number setting

Setting of the revolving single-direction pulse number.

Range: 20–40000

Typical value: When pulse equivalent = 0.001, there is no reducer, and screw pitch = 10 mm, P1-46 = 10000; when screw pitch = 5 mm, P1-46 = 5000

Value: Based on the PG division ratio (WEIHONG systems)

P2-10 Digital input pin DI1 function setting

Format: X2X1X0

X1X0=01: Set digital input DI1 to SON, matching pin No.9 of CN1.

X2=1: Set input DI1 to the normally open contact a.

Value: 101

P2-15 Digital input pin DI6 function setting

DI6 are DI7 are NC position limit signal input by default. The drive cannot work before the CN1 pin No.32 and pin No.31 are connected.

Format: X2X1X0

X2=0: Set DI6 and DI7 input to the NC contact b.

X1X0=00: Drive position limit input was not used.

Value: 000

P2-16 Digital input pin DI7 function setting

Format: X2X1X0

Value: 000

P2-17 Digital input pin DI8 function setting

Format: X2X1X0

X2X1X0=000: External EMG (emergency stop input) was not used.

Value: 000

P2-18 Digital output pin DO1 function setting

DO1 pins are pin No.6 and No.7, which are used for Z-axis clamping position braking signals.

Format: X2X1X0

X2=1: Set DO1 output to the NO contact a. X2=1: Set DO4 output to the NC contact b.

X1X0=08: Set pin No.6 and No.7 to BK- and BK+ respectively.

Value: 108

P2-22 Digital output pin DO5 function setting

DO5 pins are pin No.28 and No.27, which are used for servo alarm signals.

Format: X2X1X0

X2=0: Set DO5 output to the NC contact b.

X1X0=07: Set pin No.28 and No.27 to ALRM+ and ALRM- respectively.

Value: 007

4.4 Set and Check Pulse Equivalent

Pulse equivalent is the linear distance that the lead screw travels or the angle that the rotary axis rotates when one pulse from the system is received. It is the minimum movement unit that the system can control. The smaller the pulse equivalent, the better the machining accuracy. The bigger the pulse equivalent, the larger the machine maximum feed rate.

This section introduces the steps for setting and checking pulse equivalent.

Related Information

Concepts related to pulse equivalent are shown below:

- **Electronic gear ratio** Determines whether the servo motor amplifies or reduces the pulse frequency received from the upper computer. Electronic gear ratio larger than 1 indicates amplification while that smaller than 1 indicates attenuation.
- **Screw pitch:** The distance traveled per motor turn.
- **Mechanical reduction ratio: Mechanical reduction ratio** = **Speed reducer input rotation speed** / **Speed reducer output rotation speed** or **Teeth number of driven wheel** / **Teeth number of driving wheel** or **Motor axis rotation speed** / **lead screw rotation speed**.

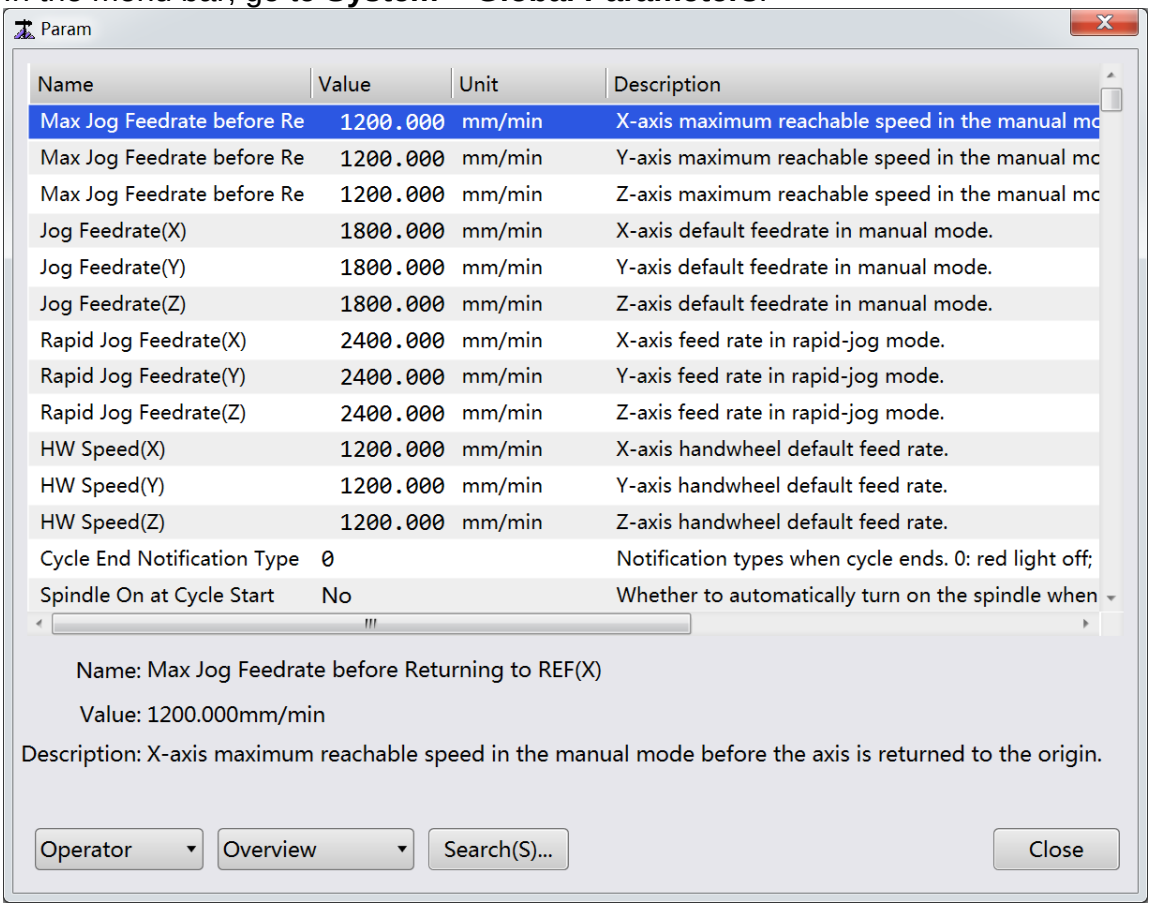
- **Encoder resolution:** The number of pulses required to make the servo motor rotate by one turn. Check the servo motor nameplate or drive user manual to find the encoder resolution.
- **Stepping angle:** The angle that the motor rotates per step pulse.
- **Subdivision:** The ratio of the motor real stepping angle to the fixed stepping angle (whole step).

Equation for calculating the pulse equivalent varies based on the motor type:

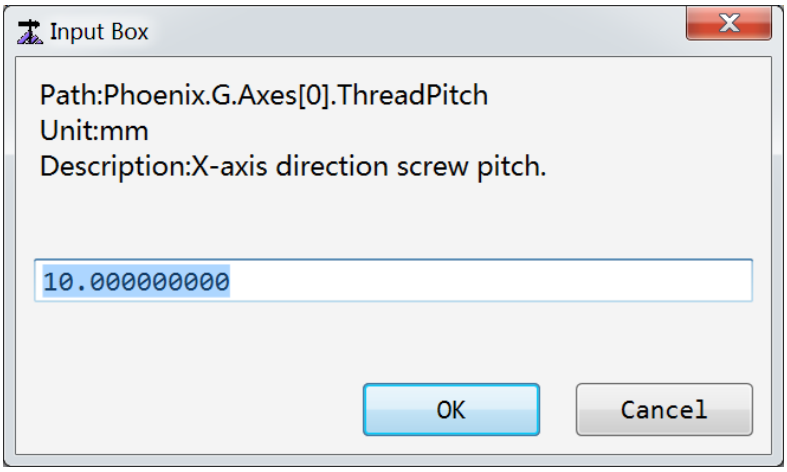
- **Stepping motor**
  - Linear axis
    Pulse Equivalent = Screw Pitch / (360 / Stepping Angle * Subdivision * Mechanical Deceleration Ratio)
  - Rotary axis
    Pulse Equivalent = 360 / (360/Stepping Angle * Subdivision * Mechanical Deceleration Ratio)

Follow the steps below the set pulse equivalent:

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



2. In the **Param** window, click the **Operator** drop-down menu in the lower left corner and select **Manufacturer**.
3. In the parameters list, find the pulse equivalent related parameters (electronic gear ratio numerator, screw pitch, mechanical reduction ratio, etc.). Double-click a parameter to open the entry dialog box:



4. Enter values in the dialog box and click **OK**.

After setting the parameters related to pulse equivalent, you need to check to see if they are set correctly.

Follow the steps below to check to see if the pulse equivalent is correct:

1. Use the handwheel or manually move an axis to any point and set the point to be the workpiece origin.
   For details, see Set Workpiece Origin.
2. Control the axis to travel a certain distance.
   For details, see the feed method description in section Machine Control Area.
3. Use a vernier caliper to measure if the actual distance and the distance shown by the coordinate system are consistent.
   - Yes: The pulse equivalent is correct.
   - No: The pulse equivalent is wrong. You need to adjust the parameter settings.

4.5 Check Axis Direction

Determine the positive direction for each axis with the right hand rule. The purpose is to avoid machine damage caused by axis moving in the wrong direction.

Operations for checking each axis direction are similar and this section takes those for the X axis as an example.

Follow the steps below to check the X axis direction:

1. Determine the X-axis positive direction based on the right hand rule.
2. In the software, click the **X+/X-** button in the machine control area to control the spindle to travel a certain distance.
3. Check to see if the axis movement direction is consistent with that determined by the right hand rule.
   - Yes: The axis direction setting is correct.
   - No: In the menu bar, go to **System** > **Global Parameters**. Set the parameter **Axis Direction(X)** to the opposite of the current value.
     - 1: Positive direction
     - -1: Negative direction

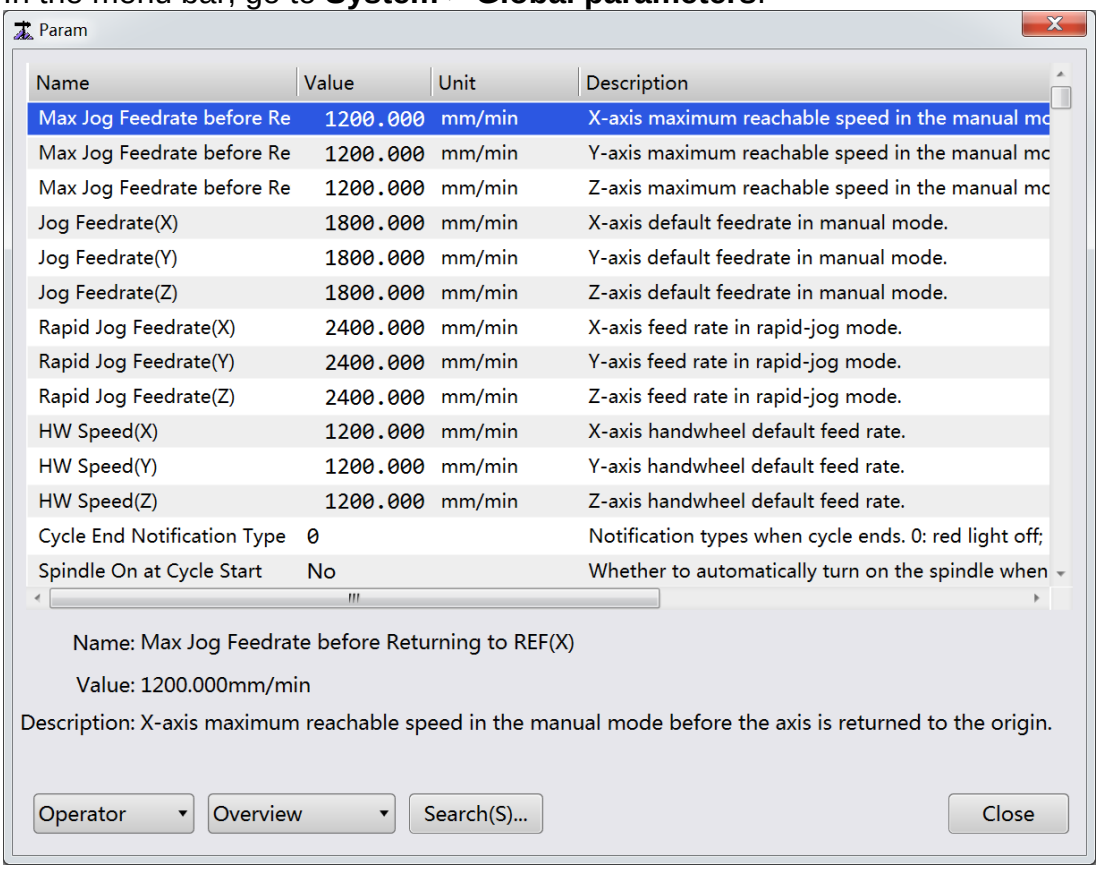
     Default value: 1

4.6 Set Workbench Travel Limits

After checking the axis movement direction and setting the mechanical origin, you need to set the axis travel limits based on the machine size to achieve soft limit functions.

Follow the steps below to set the workbench travel limits:

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



2. In the **Param** window, click the **Operator** drop-down menu in the lower left corner and select **Manufacturer**.
3. Find the **upper limit of worktable stroke**, **lower limit of worktable stroke**, **check worktable stroke** parameters for each axis.
   Descriptions of the parameters are shown below:

Parameter	Description	Allowed Range	Default Value
Check worktable stroke	Indicates whether to enable the workbench stroke soft limits.	Yes or No	Yes
Upper limit of worktable stroke	When the parameter **Check worktable stroke** is set to **Yes**, this parameter indicates the mechanical coordinate of the stroke upper limit.	[0, 99999]	400
Lower limit of worktable stroke	When the parameter **Check worktable stroke** is set to **Yes**, this parameter indicates the mechanical coordinate of the stroke lower limit.	[-99999, 0]	400

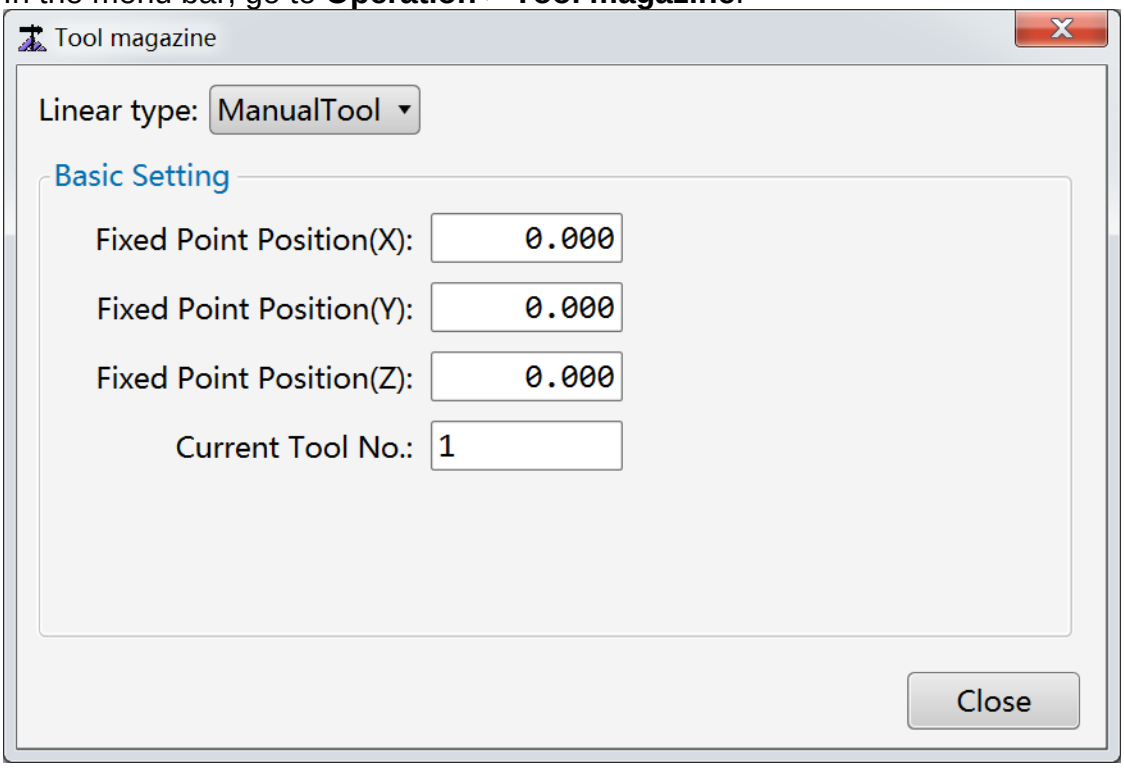
4. Double-click the target parameter and enter a value in the field. Click **OK**.
5. After settings are finished, restart the software to make the settings take effect.

4.7 Manage Tool Magazine

The system supports linear tool magazine and manual tool changing.

Follow the steps below to manage the tool magazine:

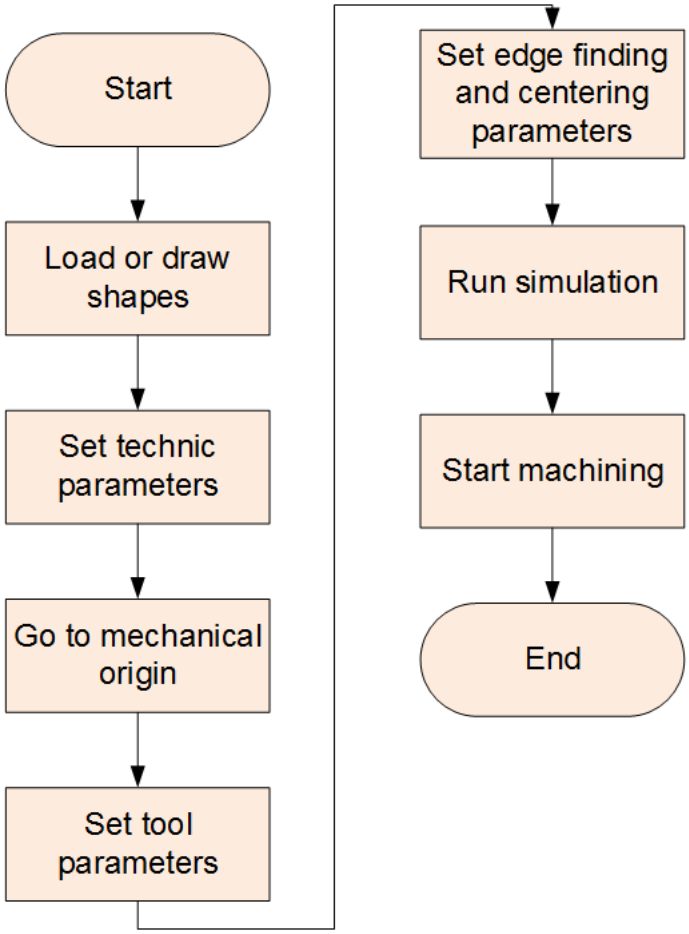
1. In the menu bar, go to **Operation** > **Tool magazine**.



2. In the **Tool magazine** dialog box, click the **Linear type** drop-down menu and select one from the following options:
   - ManualTool
     Selected by default.
   - ParallelX
     Single linear tool magazine.
   - ParallelY
     Includes single-line linear tool magazine and dual-line linear tool magazine. The system automatically selects the single-line linear tool magazine or dual-line linear tool magazine based on the number of tools on the left and right sides of the parallel Y axes.
3. Enter values for the parameters in the **Basic setting** area.
   Fixed Point Position (when the linear type is set to **ManualTool**)
   Manually change the tool after the spindle moves to a fixed point.
   Current Tool No.
   Modifying this value will change the current tool o. and use corresponding tool offset. If this value is modified, the machine does not change the tool.

5 Machining Process

The suggested machining process is shown below:



6 Shape-Related Functions

This chapter introduces how to import machining files and how to use the shape-related functions in the following aspects:

- Load Machining Files
- Draw Shapes
- Edit Shapes
- Shape Editing Assistance
- Pre-processing

6.1 Load Machining Files

The system supports .ncex, .g code, .nc, .dxf, .dwg, .plt, and .eng files.

Machining files can be loaded in different ways. Their differences are shown below:

Method	Supported Format	Description
Drag and drop.	.ncex, .dxf, .dwg, .plt, .g code, and .nc	Convenient
Click  .	.dxf, .dwg, .plt, .g code, and .nc	File modification is allowed. Saving after modification is required. The loaded file will be saved in .ncex format.
Click  .	.ncex	File modification is allowed. Saving after modification is required.
File > **Open and Load** (in the menu bar)	.ncex, .dxf, .dwg, .plt, .g code, .nc	File modification is not allowed. Shape parameters will be loaded in the software automatically.
File > **Load History File** (in the menu bar)	.ncex, .dxf, .dwg, .plt, .g code, .nc	Helps quickly locate previously loaded files. File modification is not allowed. Shape parameters will be loaded in the software automatically.

Based on your requirements, follow the steps below to load a file:

- Drag and drop:
  Click and hold a .ncex/.dxf/.dwg/.plt/.g code/.nc file and release the mouse pointer over the software **drawing area**.
- To modify an existing file: Click  or  to open/import the file.
  Note: Saving is required after modification.

- No need to modify the file: In the menu bar, go to **File** > **Open and Load**, the shape parameters will be loaded in the software.
- Import files by clicking  :

  In the **file button bar**, click  . In the displayed **Open** dialog box, select the target file.
- Open files by clicking  :

  In the **file button bar**, click  . In the displayed **Open** dialog box, select the target file.
- Open and Load:
  In the menu bar, go to **File** > **Open and Load**. In the displayed **Open** dialog box, select the target file.
- Load history files:
  In the menu bar, go to **File** > **Load History File**. In the displayed **History File** dialog box, select the target file.

Related task

To unload the loaded file, go to **File** > **Unload** in the menu bar.

6.2 Draw Shapes

The system supports drawing of **Polyline**, **Line**, **Arc**, **Ellipse arc**, **Circle**, **Ellipse**, **Rectangle**, **Polygon**, and **Text**. The system also provides a shape gallery that stores common shapes for direct use.

Shape	Use Method
 **Polyline**	For details, see Draw a Polyline.
 **Line**	For details, see Draw a Line.
 **Arc**	For details, see Draw an Arc.
 **Ellipse arc**	For details, see Draw an Ellipse Arc.
 **Circle**	For details, see Draw a Circle.
 **Ellipse**	For details, see Draw an Ellipse.
 **Rectangle**	For details, see Draw a Rectangle.
 **Polygon**	For details, see Draw a Polygon.
 **Text**	For details, see Add Text.

Shape	Use Method
Gallery	For details, see Use Gallery Shapes.

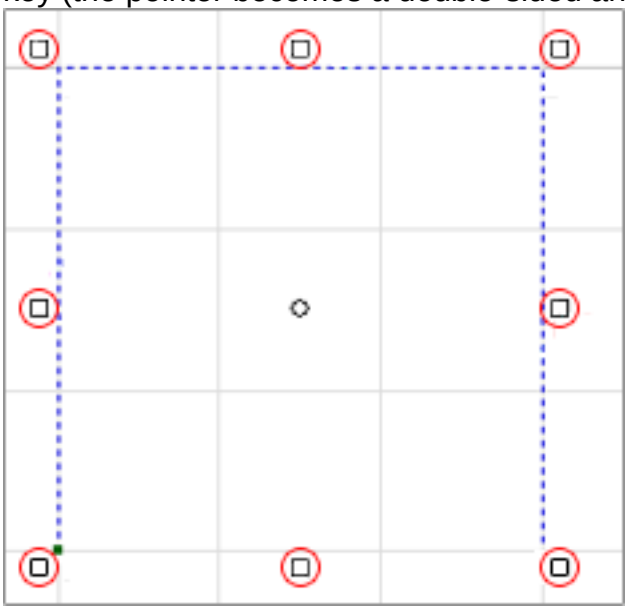
The following section introduces how to select a shape tool and how to adjust the size of a shape.

Use one of the following methods to select a shape tool:

- In the **drawing tool bar**, click the icon of the target shape tool.
- In the menu bar, click **Draw** and then select the name of the target shape tool.

After selecting the target shape, follow the steps below to adjust the shape size or position:

- Alter the **X**, **Y**, **Z**, **W**, and **L** values above the drawing area.
- Move the mouse pointer over a corner point while pressing and holding the **Shift** key (the pointer becomes a double-sided arrow) and drag to adjust the shape size:



6.2.1 Draw a Polyline

A polyline is a single object composed of multiple straight lines or arcs. You can switch between straight lines and arcs.

Follow the steps below to draw a polyline:

1. In the drawing area, click the left mouse button where you want to start the shape.
2. **Optional**: To draw a polyline composed of arcs, click the right mouse button and select **Tangent Arc**.

   Note

   A polyline is composed of straight lines by default.

3. Click the left mouse button to select the next point and so on.
4. To finish a polyline, click the right mouse button and select an option in the context menu based on your requirements:

- **Confirm**: The current point becomes the end point of the polyline, which remains open.
- **Close**: The current point becomes the polyline end point, which will be connected to the start point with a straight line automatically and the polyline becomes a closed shape.
- **Cancel**: Cancel all lines drawn.

6.2.2 Draw a Line

Follow the steps below to draw a line:

1. In the **drawing area**, click the left mouse button where you want to start the shape.
2. Click the left mouse button where you want to end the shape.

6.2.3 Draw an Arc

Follow the steps below to draw an arc:

1. In the **drawing area**, click the left mouse button where you want the circle center.
2. Click the left mouse button where you want the arc start point. The distance between the circle center and the start point will be the circle radius.
3. Click the left mouse button where you want the arc end point. The default machining direction is anti-clockwise.

6.2.4 Draw an Ellipse Arc

Follow the steps below to draw an ellipse arc:

1. In the **drawing area**, click the left mouse button where you want the center.
2. Click the left mouse button where you want the arc start point and end point. The default machining direction is anti-clockwise.

6.2.5 Draw a Circle

Follow the steps below to draw a circle:

1. In the **drawing area**, click the left mouse button where you want the circle center.
2. Click the left mouse button to select a point to determine the circle radius.

6.2.6 Draw an Ellipse

Follow the steps below to draw an ellipse:

1. In the **drawing area**, click the left mouse button where you want the center.
2. Click the left mouse button to select two points, the distance from which to the center will be the ellipse's longest diameter and shortest diameter.

6.2.7 Draw a Rectangle

Follow the steps below to draw a rectangle:

1. In the **drawing area**, click the left mouse button where you want the start point.
2. Click the left mouse button where you want the end point.

6.2.8 Draw a Polygon

Follow the steps below to draw a polygon:

1. In the **drawing area**, click the left mouse button where you want the center.
2. Click the left mouse button where you want the end point.
3. To change the number of edges, select the shape and change the value of the **Edges** field above the drawing area.
   The default edge number is 6.

6.2.9 Add Text

Follow the steps below to add text:

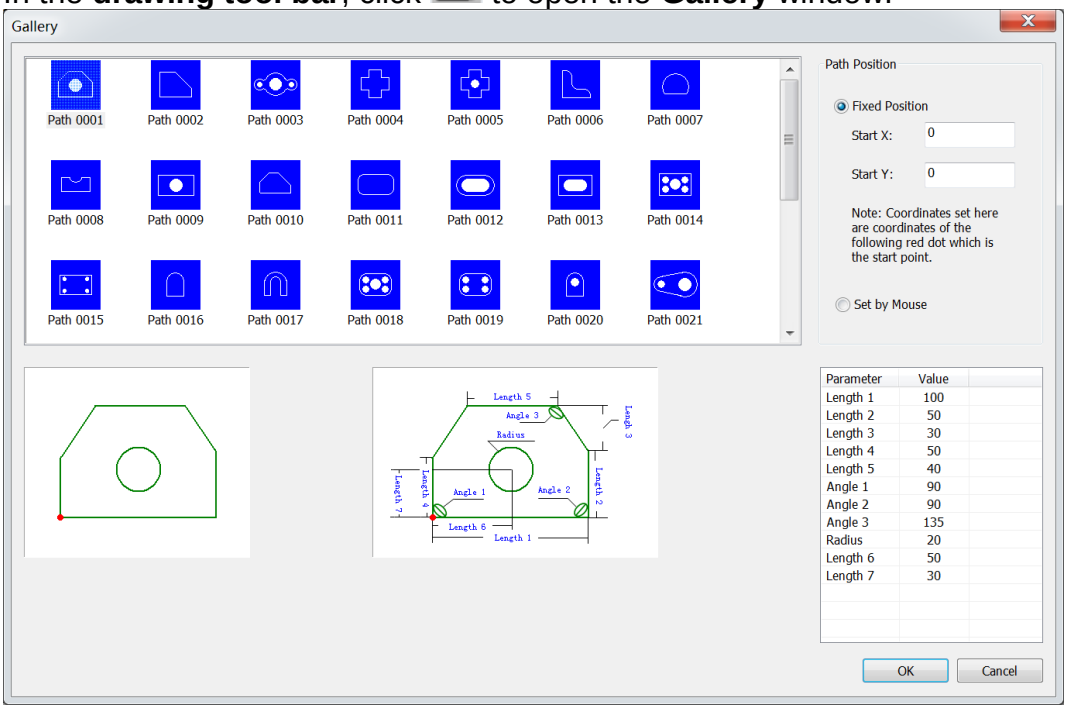
1. Drag the mouse to create a rectangle text box.

2. Enter text in the box.
3. **Optional**: To create line breaks, press **Ctrl** + **Enter** at the same time.
4. Press **Enter** to finish text entering.

6.2.10 Use Gallery Shapes

Follow the steps below to use shapes in the gallery:

1. In the **drawing tool bar**, click  to open the **Gallery** window:



2. After a shape in the gallery is selected, you can see its preview below and set the parameters by double-clicking the target value.
3. In the **Path Position** area, you can set the shape position with one of the following methods:
   - Select **Fixed Position** and enter values in the **X** and **Y** fields on the right; or

     Note

     The X and Y coordinates are those of the red point in the preview.

   - Select **Set by Mouse** and click **OK**. Close the **Gallery** window and click the left mouse button where you want the shape.

6.3 Edit Shapes

You can edit a shape as shown below:

- Translate
- Rotate
- Chamfer
- Mirror
- Scale
- Group and Break
- Merge

- Explode
- Break
- Layers

6.3.1 Translate

Translating a shape means to move it in a straight direction/change its location coordinates instead of changing its size.

Follow the steps below to translate a shape:

1. Select the target shape.
2. Use one of the following methods to translate the shape:
   - Click . Click the left mouse button in the drawing area to select a base point. Click the left mouse button on a workspace edge to select a second point. In the field between the two points, enter a distance value to move the shape.
   - In the menu bar, go to **Object** > **Translate**. Click any point in the drawing area. Click the left mouse button to select a target position.
   - Move the mouse pointer over the shape. Click and hold the left mouse button and move the shape.
   - Enter the target distance in the **Move** field below the drawing area and press **Enter**. Press ← / ↑ / ↓ / → keys to move the shape in the corresponding direction.

6.3.2 Rotate

Rotating a shape means to rotate the shape around a selected center point by a certain angle.

Follow the steps below to rotate a shape:

1. Select the target shape.
2. Use one of the following methods to rotate the shape:
   - Follow the steps below to rotate the shape around a specific point:
     A. In the menu bar, go to **Object** > **Rotate**.
     B. Click the left mouse button to select the rotation center point.
     C. Move the mouse pointer to adjust the rotation angle.
     D. Click the left mouse button to confirm the rotation.
   - While pressing and holding **Ctrl**, move the mouse pointer over any of the four corner small rectangles, press and hold the left mouse button to rotate the shape.

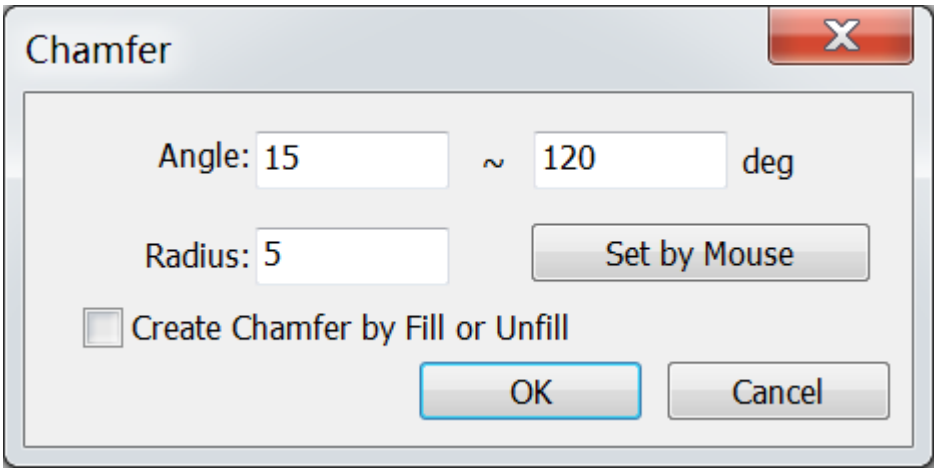
6.3.3 Chamfer

Chamfering means to chamfer all angles smaller than 180°. Two chamfering modes are available:

- **Automatic chamfering:**
  The system chamfers shapes that are selected and meet the conditions.
- **Manual chamfering:**
  Select chamfering positions manually.

Follow the steps below to add chamfered angles:

1. Select the target shape.
2. Use one of the following methods to open the **Chamfer** dialog box:
   - In the menu bar, go to **Object** > **Chamfer**.
   - Click the right mouse button, and select **Chamfer**.



3. Follow corresponding steps based on whether you choose to use automatic chamfering or manual chamfering.
    - Automatic chamfering:
      A. Enter the range of angles which you want to be chamfered in the Angle fields. Valid range: [0,180°]
      B. Enter the chamfering radius in the **Radius** field.
      C. **Optional**: To chamfer angles based on the fill/unfill property of the selected closed shape, tick **Create Chamfer by Fill or Unfill**.
      D. Click **OK**. The system chamfers angles that meet the conditions.
    - Manual chamfering:
        i. Enter the chamfering radius in the **Radius** field.
        ii. **Optional**: To chamfer angles based on the fill/unfill property of the selected closed shape, tick **Create Chamfer by Fill or Unfill**.
        iii. Click **Set by Mouse**. The mouse pointer becomes  .
        iv. Left-click on angles which you want to be chamfered.
        v. To exit chamfering mode, click the right mouse button.

6.3.4 Mirror

The system supports two types of mirroring:

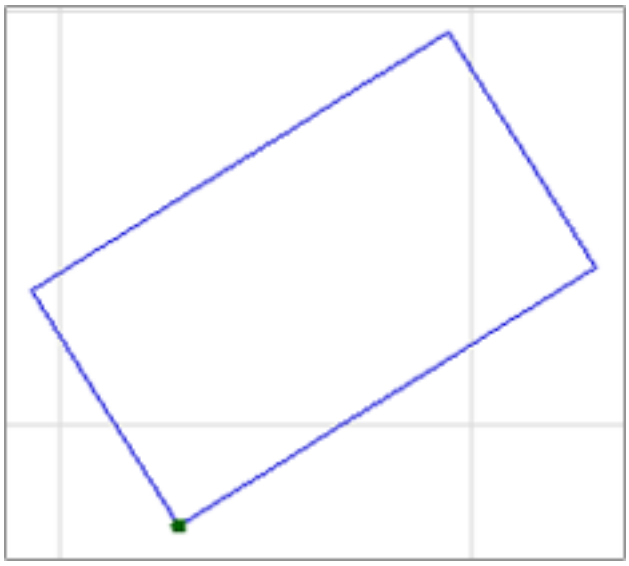
- Mirror H (horizontal): The system mirrors the shape left part and right part along the vertical axis.
- Mirror V (vertical): The system mirrors the shape upper part and lower part along the horizontal axis.

Follow the steps below to mirror a shape:

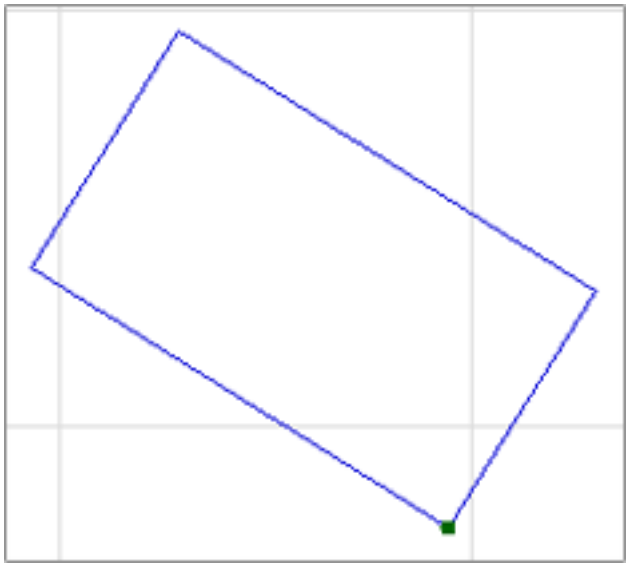
1. Select the target shape.
2. In the menu bar, go to **Object** > **Mirror** > **Mirror H/Mirror V**.

Examples of mirrored images are shown below:

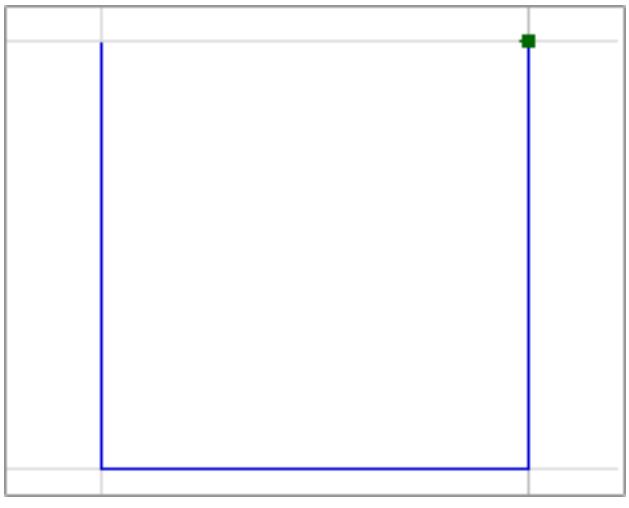
- Mirror H
    - Before



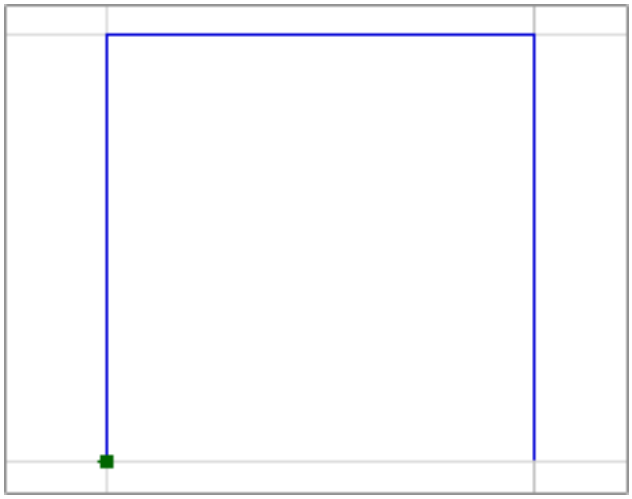
- After



- Mirror V
  - Before



- After



6.3.5 Scale

Shape scaling means to enlarge or shrink the shape without changing its proportion.

Follow the steps below to scale a shape:

1. Select the target shape.
2. Use one of the following methods to scale the shape:
   - Enter the scaling factor in the **Scale** field above the drawing area. Press **Enter** to confirm.
   - Go to **Object** > **Scale** in the menu bar:
     A. Click the left mouse button to select the scale center point.
     B. Click the left mouse button to select the target point.
     C. Move the mouse pointer to adjust the scaling factor.
     D. Click the left mouse button to confirm.

6.3.6 Group and Break

Grouping means to make multiple shapes belong to the same group.

Breaking means to release the shapes from the group.

Follow the steps below to group shapes:

1. Select the target shapes.
2. Use one of the following methods to group the shapes:

- In the menu bar, go to **Object** > **Group/Break** > **Group**.
- Click the **Group** button above the drawing area.

Follow the steps below to break a group:

1. Select the target group.
2. Use one of the following methods to break the group:

- In the menu bar, go to **Object** > **Group/Break** > **Ungroup**.
- Click the **Break** button above the drawing area.

6.3.7 Merge

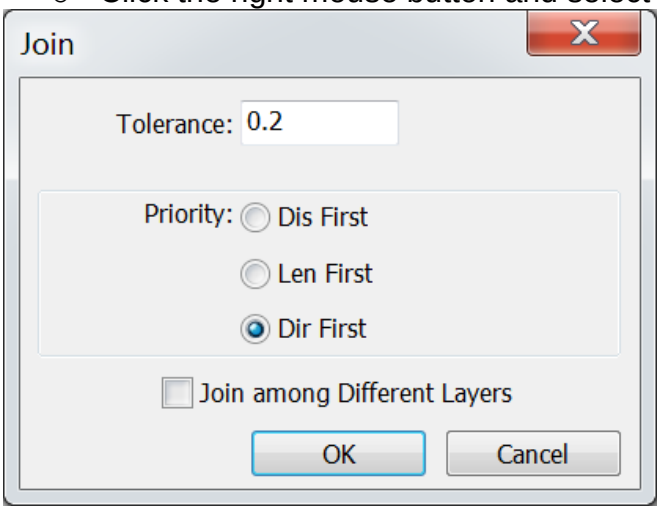
Merging means to joining unconnected shapes into a single shape.

Only applicable to open shapes, non-text objects, and groups.

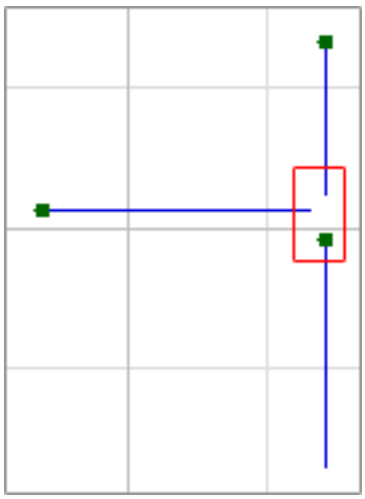
Before merging shapes, it is recommended that you enable the catch function. For details, see Catch.

Follow the steps below to merge shapes:

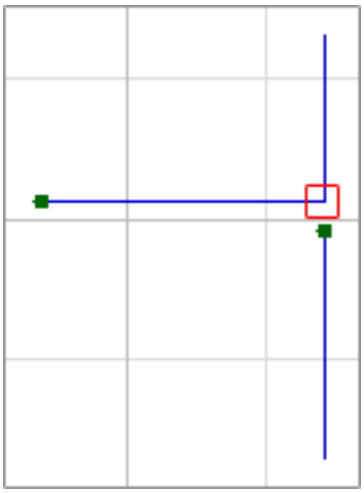
1. Select the target shapes.
2. Use one of the following methods to open the **Join** dialog box:
   - In the **drawing tool bar**, click  .
   - In the menu bar, go to **Object** > **Merge**.
   - Click the right mouse button and select **Combine**.



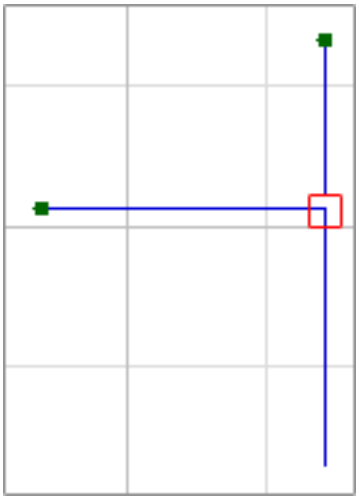
3. Enter the tolerance value in the **Tolerance** field.
   The tolerance value needs to be the maximum distance between the shapes to be merged.
   Default tolerance range: [0.01, 10] mm.
4. Select a merging priority: distance first (**Dis First**), length first (**Len First**), or direction first (**Dir First**).
   The priority strategy indicates that the system whether to merge the two shapes whose distance are the closet, length the longest, or direction the same first when there are three or more than three points that meets the tolerance requirement.
   Examples of merged shapes are shown below:
   - Before



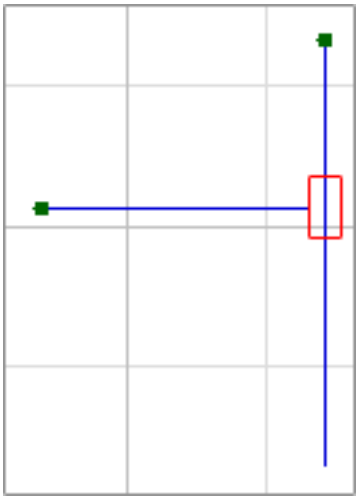
   - After
     - Dis First



- Len First



- Dir First



5. **Optional**: To merge shapes in different layers, tick **Join among Different Layers**.

6.3.8 Explode

Exploding means to delete unnecessary lines and is often used in polyline shapes.

Exploding can be used together with shape merging to correct drawing errors and ensure machining quality.

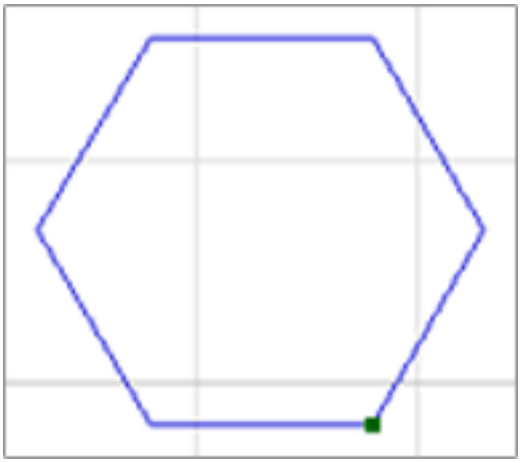
- Exploding shape groups equals to breaking/ungrouping the groups.
- Exploding text objects equals to translating text into shape.

Follow the steps below to explode a shape:

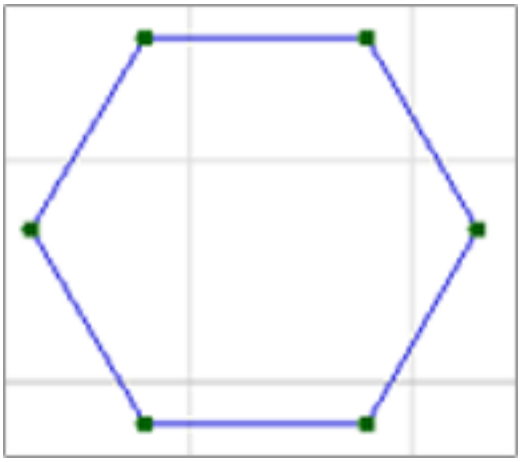
1. Select the target shape.
2. Use one of the following methods to explode a shape:
    - In the **drawing tool bar**, click  .
    - In the menu bar, go to **Object** > **Explode**.
    - Click the right mouse button and select **Explode**.

Examples of an exploded shape are shown below:

- Before



- After



6.3.9 Break

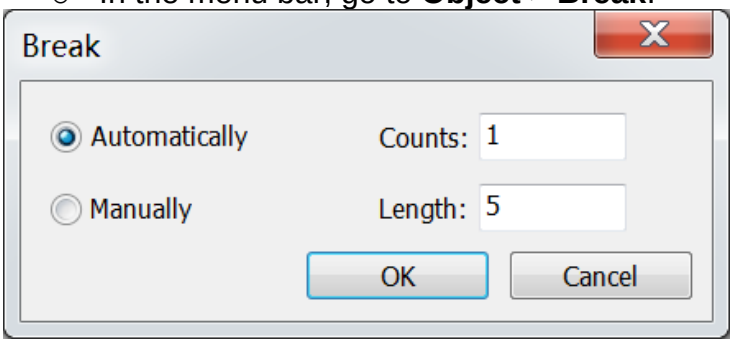
Breaking means to cut a shape into multiple polylines. The system supports two breaking methods:

- Automatic breaking: The system breaks the selected shape based on the settings.
- Manual breaking: Select the breaking positions manually. This method supports breaking for one shape at a time.

Follow the steps below to break a shape:

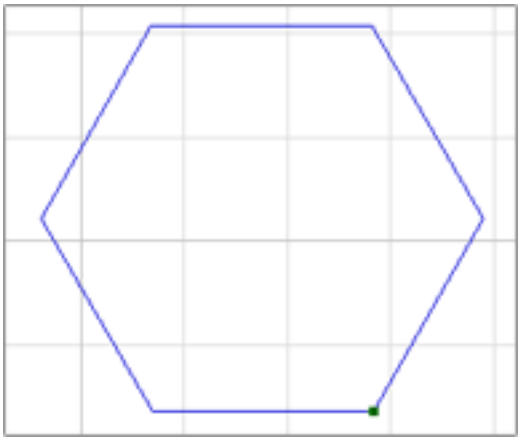
1. Select the target shape.
2. Use one of the following methods to open the **Break** dialog box:

- In the **drawing tool bar**, click .
- In the menu bar, go to **Object** > **Break**.

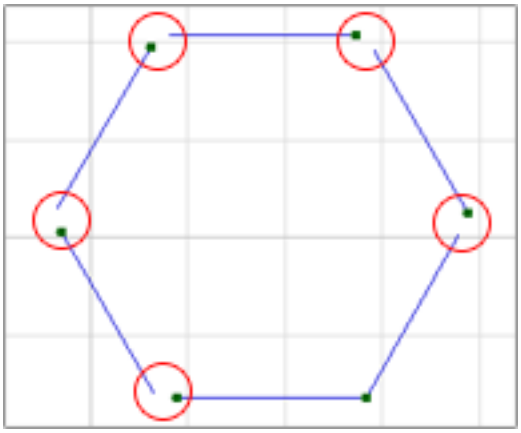


3. Follow corresponding steps based on whether you choose to use automatic breaking or manual breaking.
   - Automatic breaking:
     A. Select **Automatically**. Enter the desired number and length of broken segments in the **Counts** and **Length** fields.
     B. Click **OK**.

   Examples of an automatically broken shape are shown below:
   Before



   After

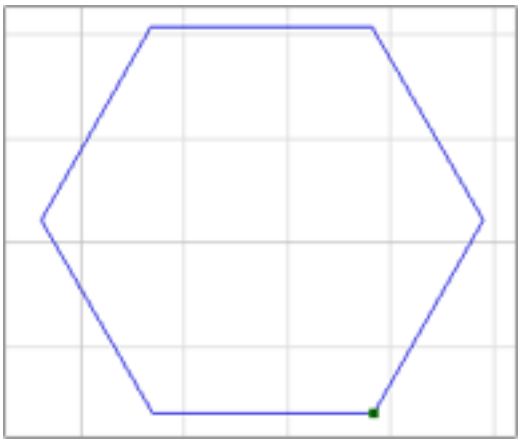


   - Manual breaking:
     A. Select **Manually**. Enter the desired length of broken segments in the **Length** field.

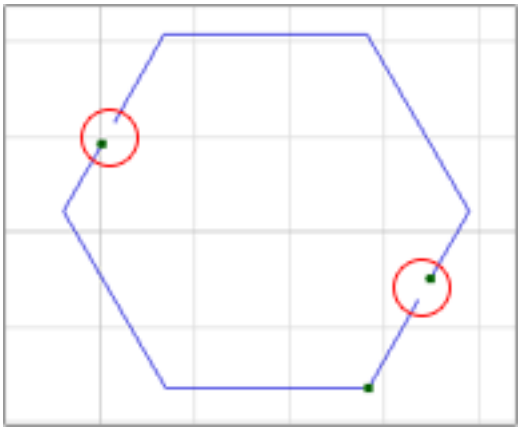
B. Click **OK**. The mouse pointer becomes  .
C. Click the left mouse button to select the breaking positions.
D. Click the right mouse button to confirm breaking.

Examples of a manually broken shape are shown below:

Before



After



6.3.10 Layers

Layers are mainly used to distinguish different technics. Each layer can use different technic settings. Objects on the same layer (with the same color) use the same technic settings. For details about setting technic parameters, see Set Technic Parameters.

The system supports 7 layers.

The default layer is layer 0, the color of the shapes on this layer is deep blue.

Follow the steps below to change the layer of a shape/shapes:

1. Select the target shape(s).
2. In the **layers bar**, click a numbered color/layer, the selected shape becomes the selected color/on the selected layer.

6.4 Shape Editing Assistance

This section introduces the following auxiliary functions for shape editing:

- Select a Shape

- Adjust View
- Catch
- Measure

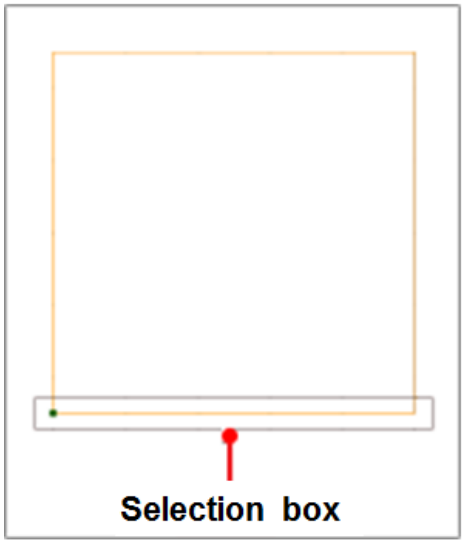
6.4.1 Select a Shape

After a shape is selected, its node numbers, feature points, and center point are shown.

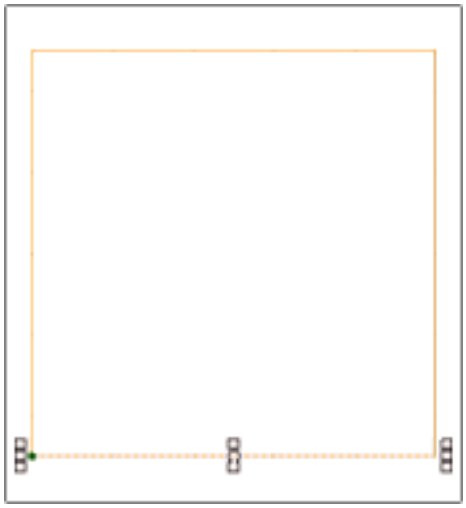
Follow the steps below to select a shape:

1. In the **drawing tool bar**, click  .
2. Use one of the following methods to select a shape/shapes:

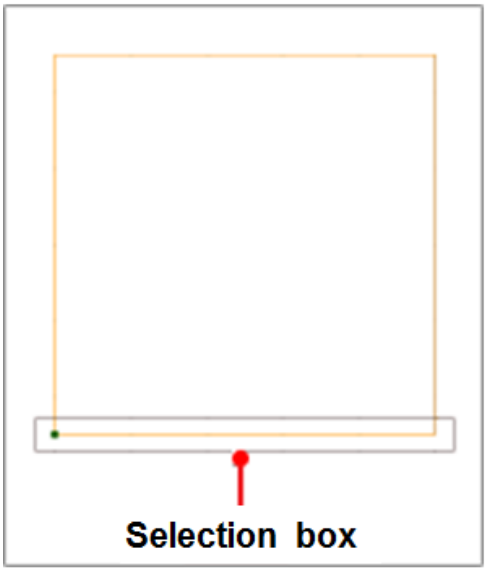
- Click the target shapes while pressing **Ctrl**.
- Press and hold the left mouse button to draw a box and select shapes:
  - When drawing a box **from the upper left corner to the lower right corner**, the shapes entirely within the box will be selected.
    - Before



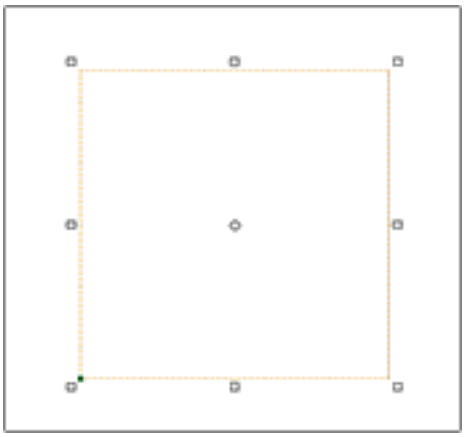
    - After



  - When drawing a box **from the lower right corner to the upper left corner**, the shapes partially or entirely within the box will be selected.
    - Before



- After



6.4.2 Adjust View

This function only applies to the view and does not change the actual shape size or coordinates. You can:

- Move View
- Best View

6.4.2.1 Move View

Redefine the location of the shape in the window for clearer observation of the shape's different parts.

Follow the steps below to move the view:

Use one of the following methods to move the view:

- Press and hold the mouse scroll wheel and move the mouse to the target position.
- Use the "view move" function:
  a. Use one of the following methods to select the "view move" tool:
     - In the **drawing tool bar**, click  .
     - In the menu bar, go to **View** > **View move**.
  b. Press and hold the left mouse button where you want the base point and move the mouse. Release the button in the target position.

c. Click the right mouse button or press **Esc** to exit "view move" mode.

6.4.2.2 Best View

In best view, the shapes are fully displayed in a way that suits the window.

Follow the steps below to switch to best view:

Use one of the following methods to switch to best view:

- In the **drawing tool bar**, click  .
- In the menu bar, go to **View** > **Best view**.
- Press the ***** key.

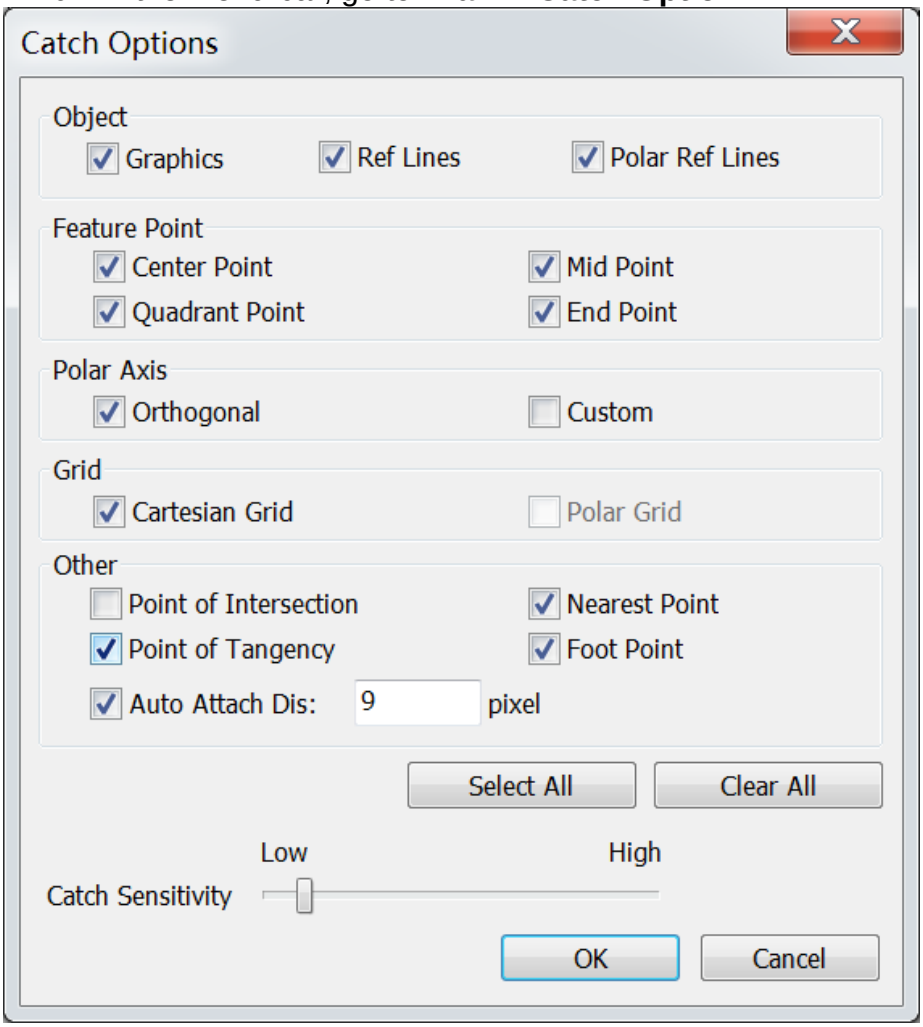
6.4.3 Catch

This function allows the system to locate shape feature points more accurately during shape drawing.

When the mouse pointer approaches a feature point, the system can easily recognize it to accurately connect shapes as required.

Follow the steps below to use the catch function:

1. Use one of the following methods to open the **Catch Options** window:
   - In the **file button bar**, click  .
   - In the menu bar, go to **Draw** > **Catch Option**.



2. Tick the target feature points and adjust the catching sensitivity by dragging the **Catch Sensitivity** slider.
   The higher the catching sensitivity, the easier it is for the system to capture the feature points.
3. Use one of the following methods to enable the catch function:
   - In the **file button bar**, click .
   - In the menu bar, go to **Draw** > **Catch**.

6.4.4 Measure

This function supports measuring of the distance between any two points, X/Y-axis deviation, and X-axis positive angle.

Follow the steps below to use the measuring function:

1. In the **drawing tool bar**, click .
2. Click the left mouse button to select the start point.
3. Move the mouse pointer to the end point. The measuring result is displayed below the mouse pointer.

6.5 Pre-processing

Pre-processing of shapes before formal machining helps with a better machining result.

The system supports pre-processing of a single shape, such as smoothing curves and reducing polylines, and batch pre-processing of shapes, such as instant processing.

6.5.1 Smoothing Curve

This function smoothens multiple polylines for smoother machining.

Follow the steps below to smooth curves:

Select the target shape and use one of the following methods to smooth it:

- In the menu bar, go to **Object** > **Smooth curve**.
- Click the right mouse button and select **Smooth Curve**.

A message indicating curve smoothing success will be displayed.

6.5.2 Reduce Polyline

A polyline is a single object composed of multiple straight lines and curves and whose shape is controlled by nodes.

Reducing a polyline means that the system automatically reduce the number of nodes as allowed by the tolerance to improve shape operation response speed.

Follow the steps below to reduce the number of polyline nodes:

1. Select the target polyline and use one of the following methods to open the **Simplify Curve** dialog box:
   - In the menu bar, go to **Object** > **Reduce polyline**.
   - Click the right mouse button and select **Simplify Curve**.
2. Enter the tolerance value in the **Tolerance** field and click **OK**.

A message indicating polyline reduction success will be displayed.

6.5.3 Translate Text

This function converts text into polylines for subsequent technic setting.

Use the following method to convert text into polyline:

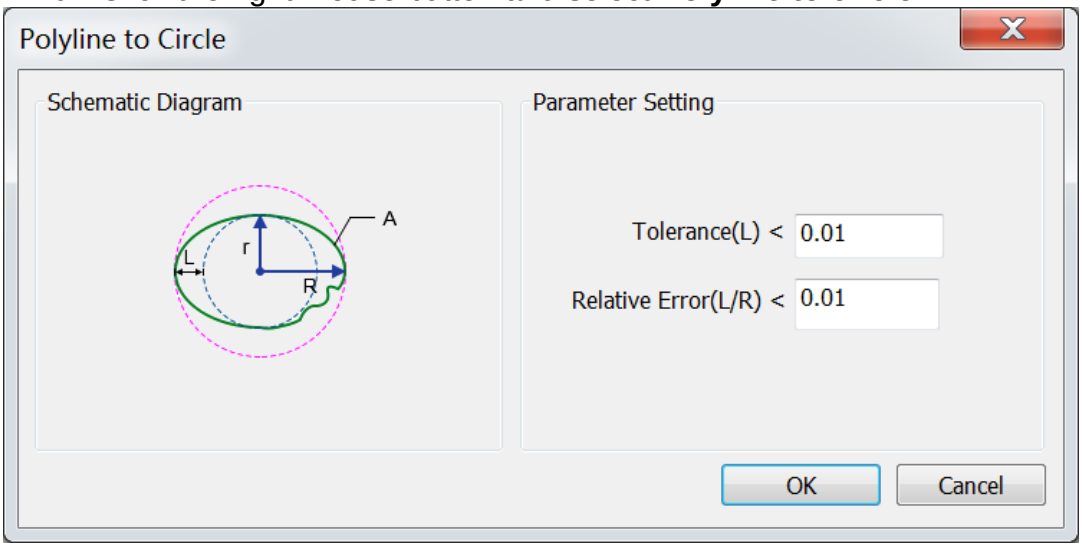
Select the target text. In the menu bar, go to **Object** > **Translate text**.

6.5.4 Polyline to Circle

This function converts closed polylines that are similar to circles into circles.

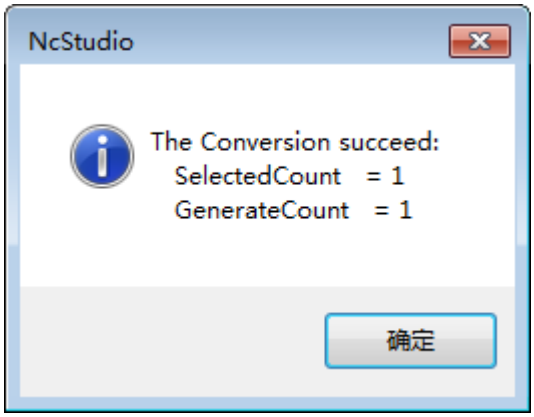
Follow the steps below to use the function:

1. Select the target shape and use one of the following methods to open the **Polyline to Circle** dialog box:
   - In the menu bar, go to **Object** > **Trans to circle**.
   - Click the right mouse button and select **Polyline to circle**.



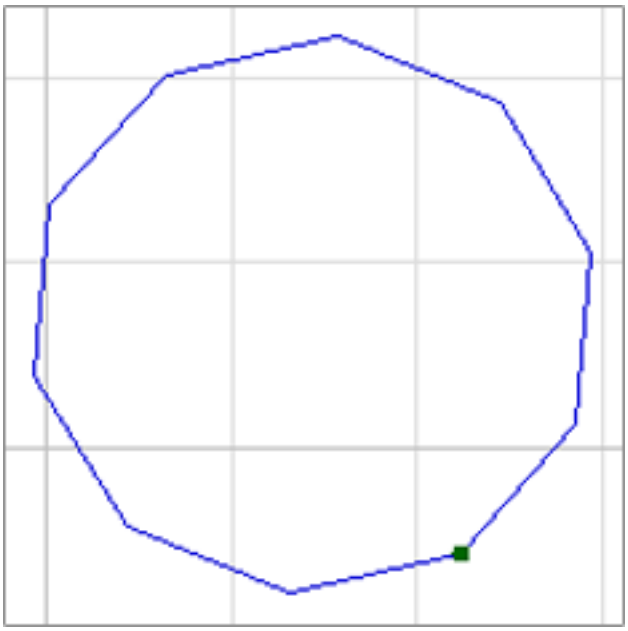
2. Enter values in the **Tolerance** and **Relative Error** fields.

After conversion, a prompt will be displayed as shown below:

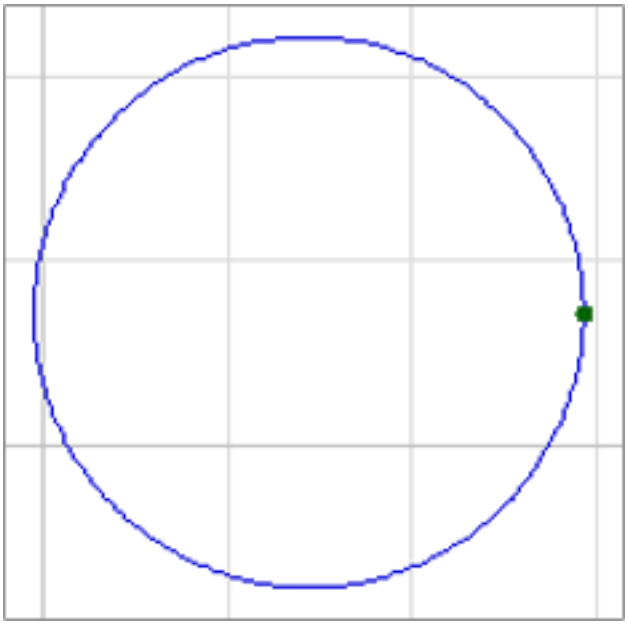


Examples of a converted shape are shown below:

- Before



- After

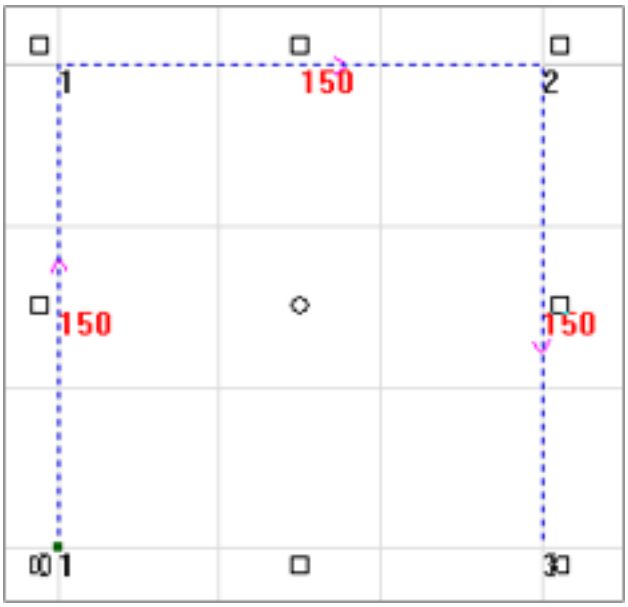


6.5.5 Edit Polylines

The system supports editing of polylines, such as enlarging/shrinking, modifying the position of polyline nodes and altering the line segment length.

Follow the steps below to edit polylines:

1. After a polyline is selected, a node editing red box and node numbers are displayed. The red box indicates the node currently selected.



2. To select other nodes, click the **+/-** buttons or enter the node No. in the **ChooseNode** field above the drawing area.

3. Follow corresponding steps based on your needs:
   - To enlarge or shrink a polyline, select the target polyline, enter the enlarging/shrinking factor in the **Scale** field above the drawing area and press **Enter**.
   - To change the width or height of a polyline, select the target polyline, enter values in the **W** or **H** field above the drawing area, and press **Enter**.
   - To change the position of the selected node, enter values in the **X** or **Y** field above the draw area, and press **Enter**.
     If the entered X and Y coordinates for a node are the same as those of the adjacent nodes, the modification does not take effect.
   - To alter the distance between the current node and the previous node, enter a value in the **Length** field above the drawing area, and press **Enter**.

     Note

     If the selected node is the end point of an tangent arc, the **Length** value is the arc radius and cannot be modified.

6.5.6 Instant Processing

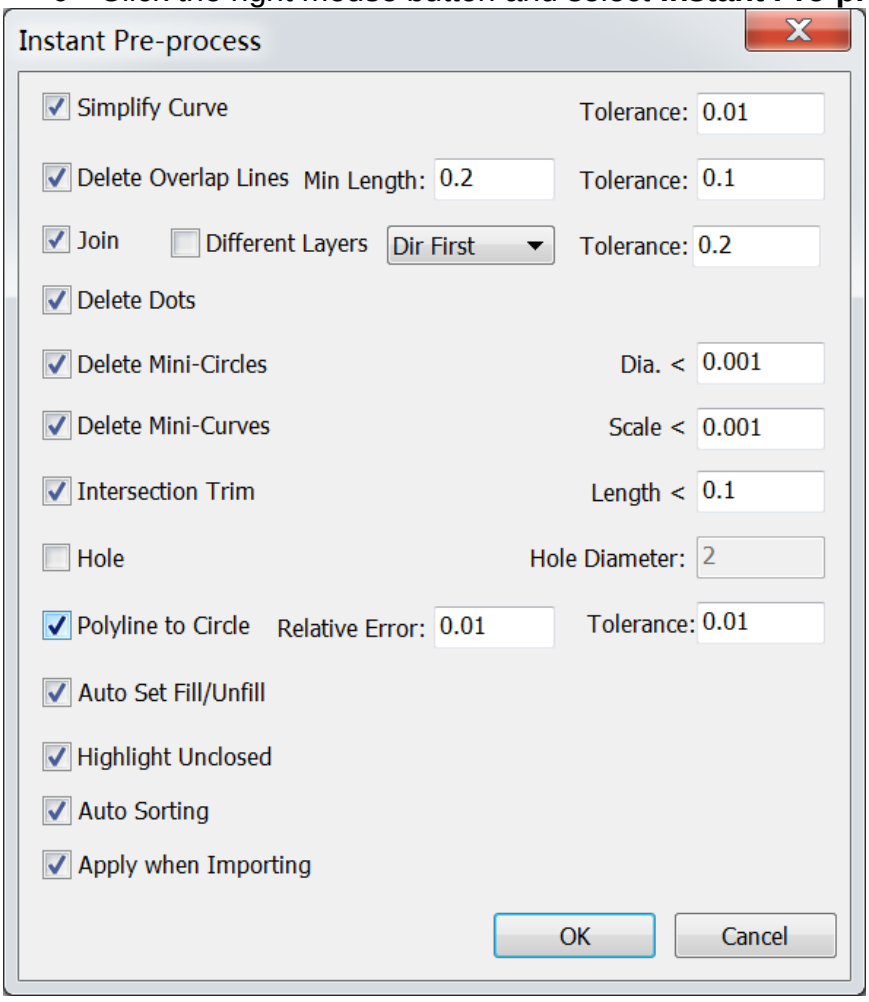
The instant processing function includes common shape processing items for your selection to achieve better machining results.

The instant processing options include **Simplify Curve**, **Delete Overlap Lines**, **Join**, **Delete Dots**, **Delete Mini-Circles**, **Delete Mini-Curves**, **Intersection Trim**, **Polyline to Circle**, **Auto Set Fill/Unfill**, etc.

Follow the steps below to use the instant processing function:

1. Select the target shape. Use one of the following methods to open the **Instant Pre-process** dialog box:
   - In the menu bar, go to **Object** > **Instant Processing**.

- Click the right mouse button and select **Instant Pre-process**.



2. Tick target options and set the parameters.
3. **Optional**: If **Apply when Importing** is ticked, the system automatically runs the selected processing items when a file is imported.

7 Set Technic Parameters

7.1 Set Basic Technic Parameters

You can set the layer color, machining sequence, allowance, plate information, cutting type, and grinding type parameters differently for each technic.

The system supports seven technics, including a custom technic.

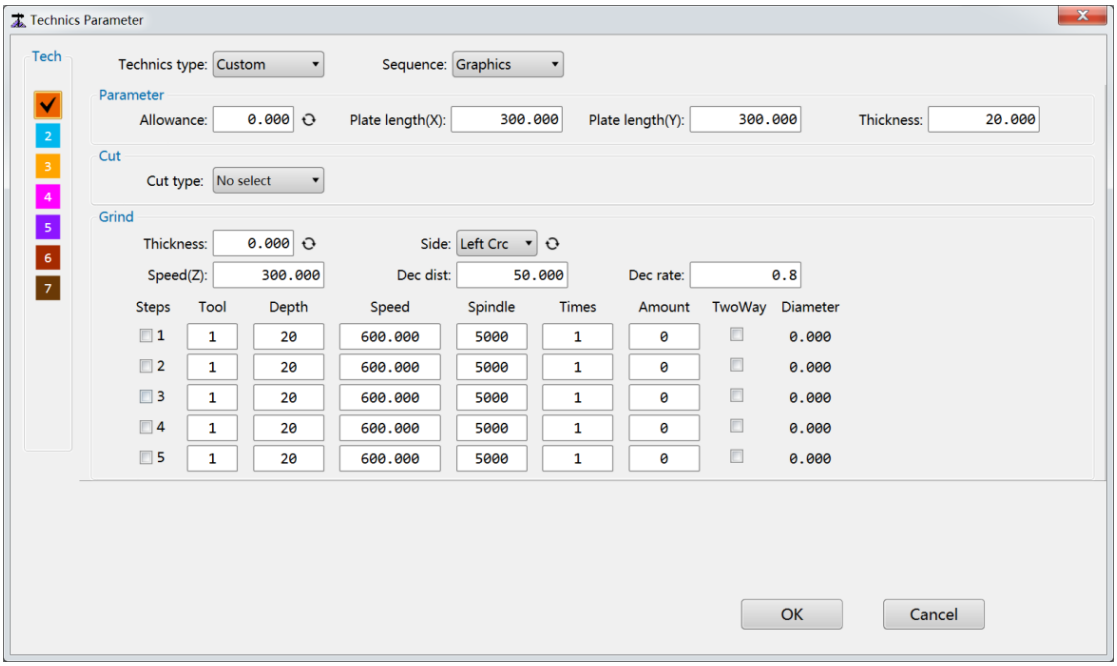
Descriptions of the technics are shown below:

Technic	Description
Pot hole	Only applicable to closed shapes, such as circles, ellipses, rectangles, polygon, etc.
Front water	The lower part of the drawing area is the front water area by default. Only applicable to open shapes, such as lines, arcs, ellipse arcs, polylines.
Back water	The upper part of the drawing area is the back water area by default. Only applicable to open shapes, such as lines, arcs, ellipse arcs, polylines.
Outline	Only applicable to closed shapes, such as circles, ellipses, rectangles, polygon, etc.
Hole opener	Vertical machining at the circle center and not related to the tool radius. Only applicable to circles.
DrainTrough	Applicable to lines and closed areas with the same height.
Custom	Applicable to close and open shapes.

7.1.1 Set Pot Hole Parameters

Follow the steps below to set pot hole parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **Pot hole**.

Note

After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

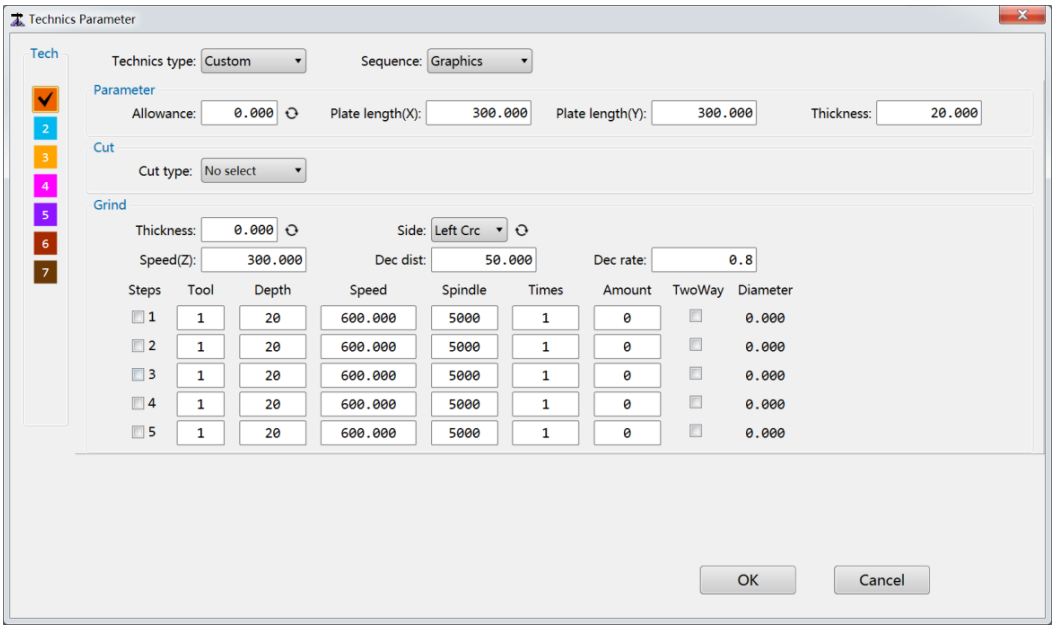
---

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** area. For details, see Technic Parameter Description.
6. Click the **Cut type** drop-down menu, select a cutting type, and set related parameters. For details, see Technic Parameter Description.
   - **No select**
   - **Layered cut**: Cutting layer by layer
   - **Hole cut**: Cutting after drilling a hole at the lead-in line start point
   - **Screw Cutting**: Slow spiral cutting of closed shapes. The Z axis goes down by the screw pitch each turn.
7. Set the parameters in the **Grind** area and tick the **Steps** numbers to increase or decrease grinding steps. At most 5 grinding steps can be added. For details, see Technic Parameter Description.
8. Click **OK**.

7.1.2 Set Front Water Parameters

Follow the steps below to set front water parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **Front water**.

Note

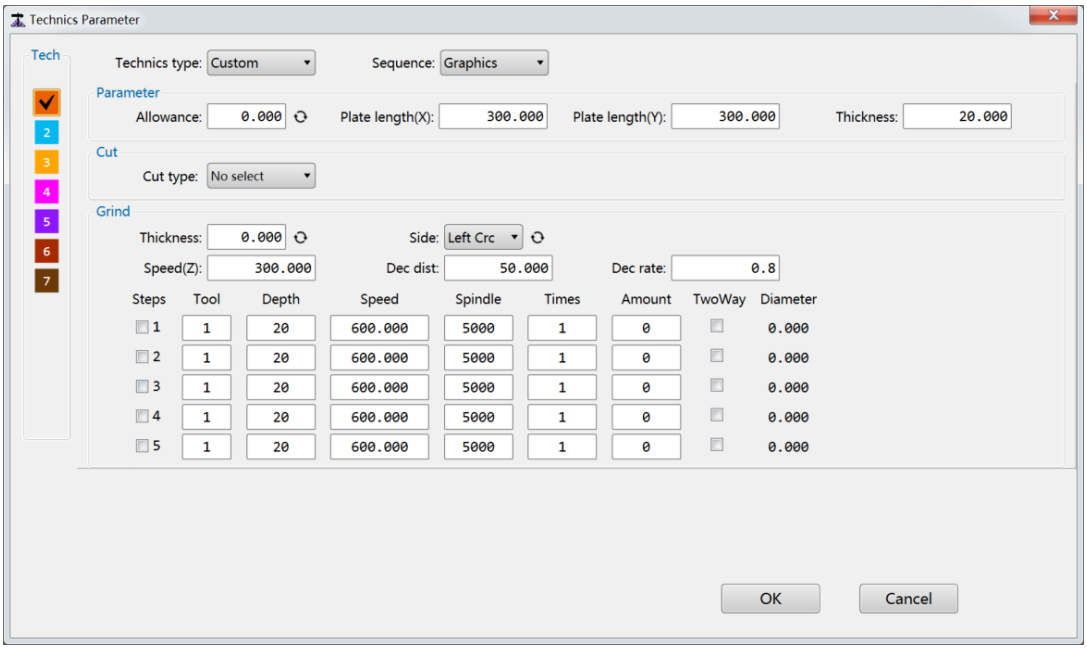
After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** area. For details, see Technic Parameter Description.
6. Click the **Cut type** drop-down menu, select a cutting type, and set related parameters. For details, see Technic Parameter Description.
   - **No select**
   - **Layered cut**: Cutting layer by layer
7. Set the parameters in the **Grind** area and tick the **Steps** numbers to increase or decrease grinding steps. At most 5 grinding steps can be added. For details, see Technic Parameter Description.
8. Click **OK**.

7.1.3 Set Back Water Parameters

Follow the steps below to set back water parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **Back water**.

Note

After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

---

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** area. For details, see Technic Parameter Description.
6. Set the parameters in the **Grind** area and tick the **Steps** numbers to increase or decrease grinding steps. At most 5 grinding steps can be added. For details, see Technic Parameter Description.

Note

If water-retaining bars are used for the back water-retaining area, set **Thickness** in the **Grind** area.

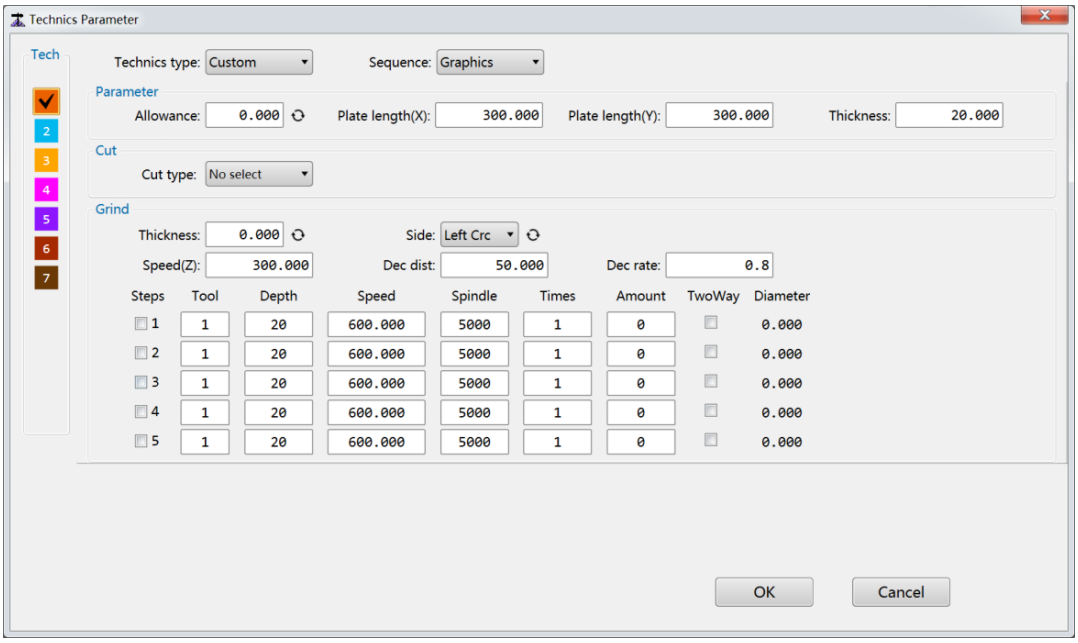
---

7. Click **OK**.

7.1.4 Set Outline Parameters

Follow the steps below to set outline parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **Outline**.

Note

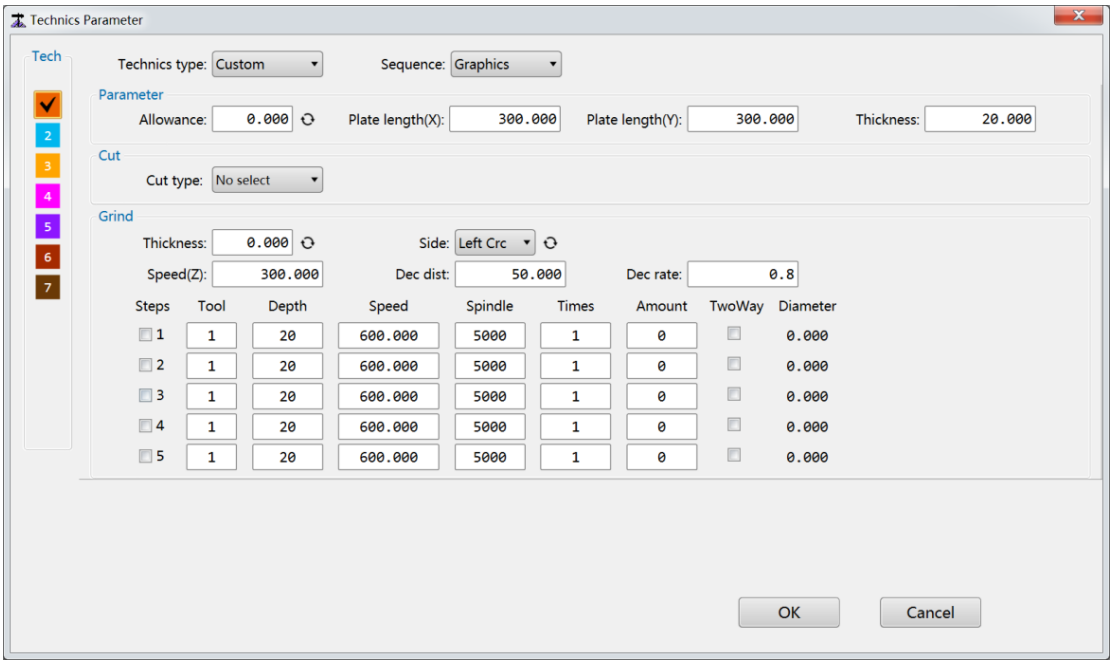
After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** area. For details, see Technic Parameter Description.
6. Click the **Cut type** drop-down menu, select a cutting type, and set related parameters. For details, see Technic Parameter Description.
   - **No select**
   - **Layered cut**: Cutting layer by layer
   - **Screw Cutting**: Slow spiral cutting of closed shapes. The Z axis goes down by the screw pitch each turn.
7. Set the parameters in the **Grind** area and tick the **Steps** numbers to increase or decrease grinding steps. At most 5 grinding steps can be added. For details, see Technic Parameter Description.
8. Click **OK**.

7.1.5 Set Hole Opener Parameters

Follow the steps below to set hole opener parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **Hole opener**.

Note

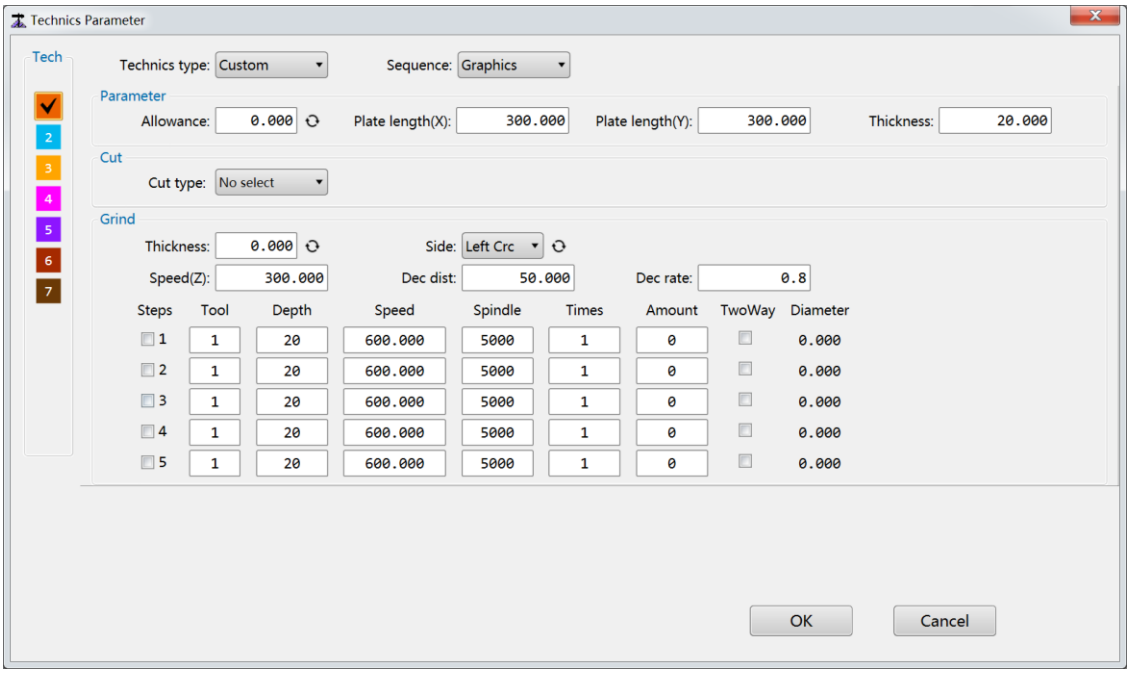
After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** and **Param** areas. For details, see Technic Parameter Description.
6. Click **OK**.

7.1.6 Set Custom Parameters

Follow the steps below to set custom parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **Custom**.

Note

After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

---

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** area. For details, see Technic Parameter Description.
6. Click the **Cut type** drop-down menu, select a cutting type, and set related parameters. For details, see Technic Parameter Description.
   - **No select**
   - **Layered cut**: Cutting layer by layer
   - **Hole cut**: Cutting after drilling a hole at the lead-in line start point
7. Set the parameters in the **Grind** area and tick the **Steps** numbers to increase or decrease grinding steps. At most 5 grinding steps can be added. For details, see Technic Parameter Description.

Note

Click the **Side** drop-down menu in the **Grind** area to select a machining side (**Left Crc** or **Right Crc**) based on the shape machining direction.

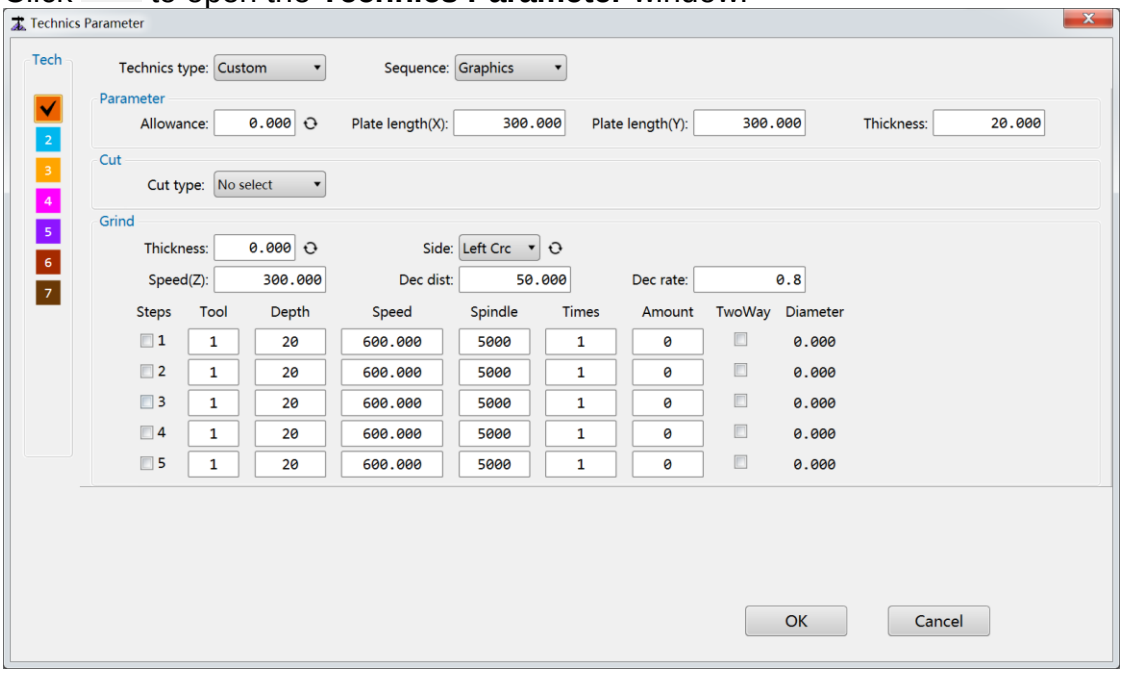
---

8. Click **OK**.

7.1.7 Set DrainTrough Parameters

Follow the steps below to set draintrough parameters:

1. Click  to open the **Technics Parameter** window:



2. In the **Tech** area, select a colored layer for the technic.
3. Click the **Technics type** drop-down menu and select **DrainTrough**.

Note

After a colored layer is selected for the technic, shapes on the layer becomes the selected color.
Because the layers are mainly used to distinguish different technics, it is recommended that you assign different layers to different technics.

---

4. Click the **Sequence** drop-down menu to select machining by **Graphics** or by **Tools**.
5. Set the parameters in the **Parameter** area. For details, see Technic Parameter Description.
6. Click the **Cut type** drop-down menu in the **Cut** area, select a cutting type, and set related parameters. For details, see Technic Parameter Description.
   - **No select**
   - Line: The tool moves along the shape lines.
   - Area: The machining method is normally horizontal milling. It is similar to pocketing and applicable to closed areas, including irregular shapes.
7. Set the parameters in the **Grind** area and tick the **Steps** numbers to increase or decrease grinding steps. At most 5 grinding steps can be added. For details, see Technic Parameter Description.
8. Click **OK**.

7.2 Technic Parameter Description

Allowance

Indicates X-axis and Y-axis radial feeding allowance.

The system will reserve distances accordingly on the machining side.

Plate size parameters

Include **Plate length(X)**, **Plate lengh(Y)**, and **Thickness**. Plate size is immediately shown in the drawing area.

Set the plate size parameters based on the actual size because they affect the positions of the shapes.

Z down tool point height

The distance between the tool and the workpiece surface before lowering the tool.

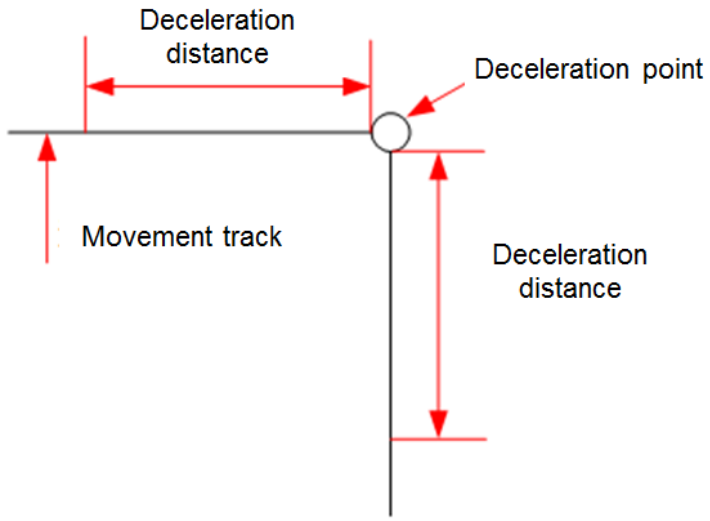
The Z axis travels from its highest point to this height at the G00 speed and travels at the **Z Cut Spd** for each tool after traveling lower than this height.

The cutting point is the green dot in each shape. If a lead line is added, the tool cuts from the lead line start point.

Dec dist and Dec rate

Set the distance before and after the deceleration point during which the tool decelerates at the set decelerating rate.

Actual feed rate within decelerating distance = Feed rate setting * Dec rate



Cut depth

The cutting depth from the Z-axis workpiece origin.

Z Cut Spd

The cutting speed from the Z-axis workpiece origin.

Cut speed

The tool radius is in direct proportion to the machining speed.

Speed

Indicates the grinding speed.

Spindle

Indicates the spindle rotational speed during grinding.

Amount and Times

Amount indicates the total feed amount.

One-time feed amount = Total feed amount/Feed times

Layer Num and Last incise

Layer Num indicates the tool feeding times during layered cutting.

Last incise indicates the amount of last tool feed during layered cutting.

One-time cutting depth for layers other than the bottom layer = (Cut Depth - Last Incise)/(Layer Num - 1)

Two way

Only applicable to open shapes.

If **Two way** is ticked, the system executes the machining path back and forth.

Commonly used in **Front water**, **Back water**, and **Custom** technics.

Thickness

Indicates the back water bar thickness.

Steps

A machining process can include multiple steps.

Within a step, the cutting amount and tool do not change.

Diameter

Indicates the diameter of the tool used in the current step.

Pitch of screw

Indicates the Z-axis feed amount per turn during screw cutting.

7.3 Set Other Technic Parameters

The following parameters can be set for a technic:

- Unfill and Fill
- Scalpel Line
- Set Workpiece Origin
- Set Slow Spot
- Set Machining Sequence
- Set Machining Direction

7.3.1 Unfill and Fill

Unfill is used when the material inside the closed shape needs to be removed. **Fill** is used when the material inside the closed shape needs to be reserved.

Follow the steps below to set unfill/fill parameters:

1. Select the target shape(s).
2. Use one of the following methods to select unfill/fill:
   - In the menu bar, go to **Object** > **Unfill/Fill** > **Fill/Unfill/Auto Set Fill**.

   If **Auto Set Fill** is selected, the system automatically determines whether to use fill or unfill.
   - Click the right mouse button, select **Unfill/Fill**, and select **Unfill** or **Fill**.

7.3.2 Scalpel Line

The scalpel line is used to avoid the tool staying at the machining start point for too long and causing errors or workpiece damage for better machining accuracy.

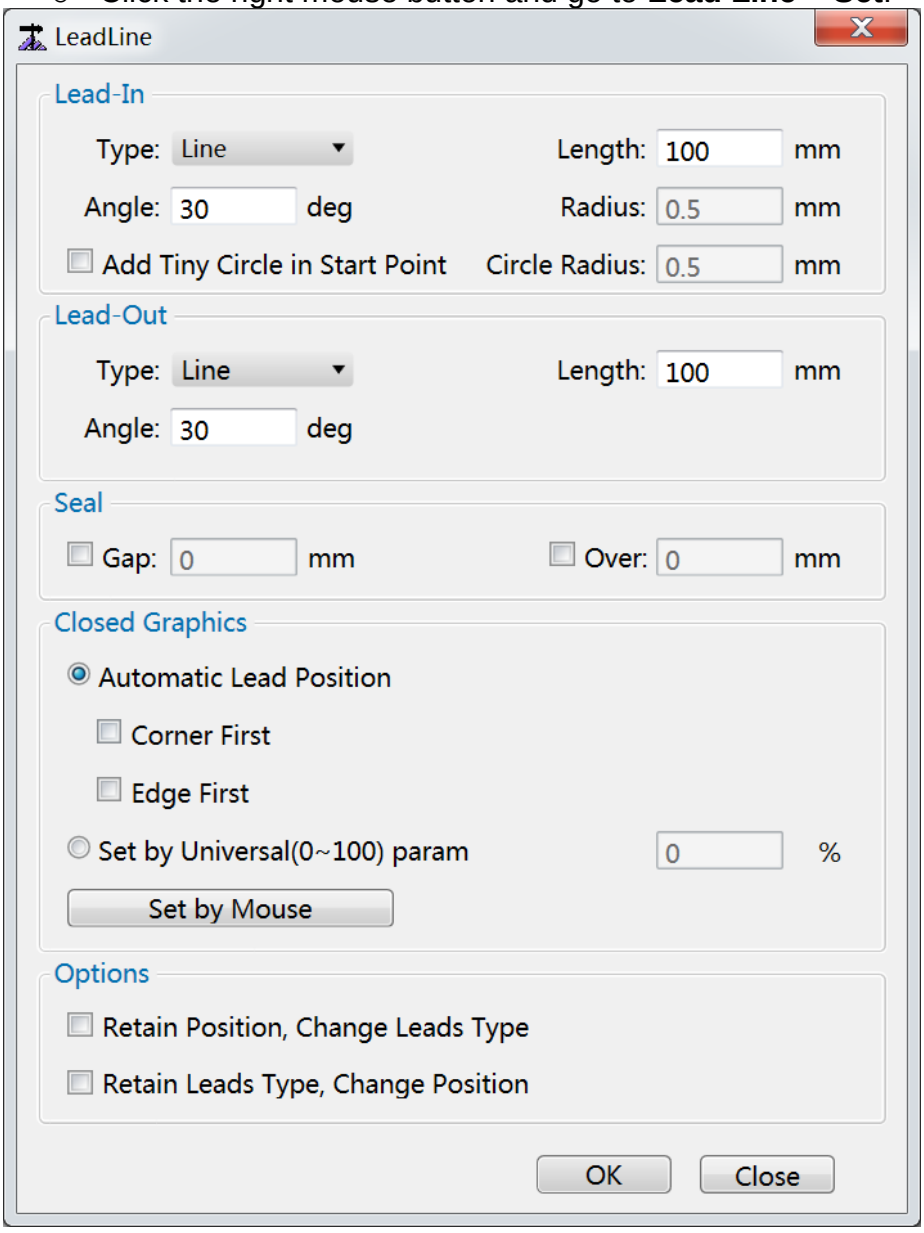
If only closed shapes are selected, functions only applicable to open shapes will be disabled and vice versa.

Scalpel lines include lead-in lines and lead-out lines.

- Lead-in line types: **Line**, **Arc**, and **Hook. Hook** lead-in lines are composed of connected arcs and straight lines.
- Lead-out line types: **Line** and **Arc**.

Follow the steps below to set scalpel line parameters:

1. Select the target shape(s).
2. Use one of the following methods to open the **LeadLine** dialog box:
   - In the **drawing tool bar**, click  .
   - In the menu bar, go to **Object** > **Scalpel Line**.
   - Click the right mouse button and go to **Lead Line** > **Set**.



3. Set the parameters in the **Lead-In** and **Lead-Out** areas.
   Parameter definition may vary based on the scalpel line type:
   - Line
     - **Angle**: Formed by the scalpel line and the tangent line at the scalpel line and the shape's crossing point.
     - **Length**: The line length.
   - Arc
     - **Angle**: The central angle
     - **Length**: The arc length.
   - Hook
     - **Radius**: The arc radius
     - **Length**: Sum of the arc radius and straight line length.
4. In the **Seal** area, tick **Gap**/**Over** to set the scalpel line seal.
   - **Gap**: Open lead lines.
   - **Over**: Closing lead lines.
5. Use one of the following methods to set the scalpel line position:
   - Select **Automatic Lead Position** and tick **Corner First** or **Edge First**.
     - **Corner First**: Add a scalpel line at corners.
     - **Edge First**: Add a scalpel line on the longest edge.
   - Select **Set by Universal(0~100) param** and enter a percentage value in the field.
   - Click the **Set by Mouse** button. The mouse pointer becomes  . Click on the shape edges to select the scalpel line position. Click the right mouse button or press **Esc** to exit.
6. **Optional**: In the **Options** area, you can tick **Retain Position, Change Leads Type** or **Retain Leads Type, Change Position**.
7. Click **OK**.

Related tasks

- To delete a scalpel line, select the target shape, click the right mouse button, and go to **Clear** > **Lead Line**.
- To modify a scalpel line, select the target shape, click the right mouse button, and go to **Lead Line** > **Start Point**. Follow corresponding steps based on your needs:
  - To modify the position of the scalpel line but not its angle or length, click the left mouse button on the target position.
  - To re-draw a straight lead-in line, click the left mouse button outside the shape and then click the left mouse button on a shape edge.

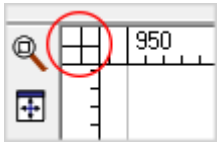
7.3.3 Set Workpiece Origin

The workpiece origin is the zero point of the workpiece coordinate system. It is selected normally under considerations of programming and size conversion convenience, and small machining errors. The position of the workpiece origin can be changed.

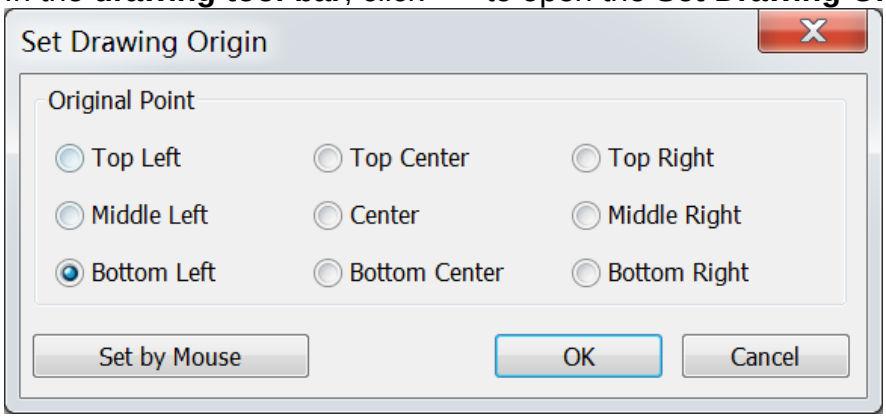
Before starting machining, ensure that the workpiece origin is located on the workpiece.

Follow the steps below to set the workpiece origin:

1. Use one of the following methods to determine the position of the drawing origin:
   - Left-click and hold on the intersection of the horizontal and vertical rulers. Drag it to the target position.



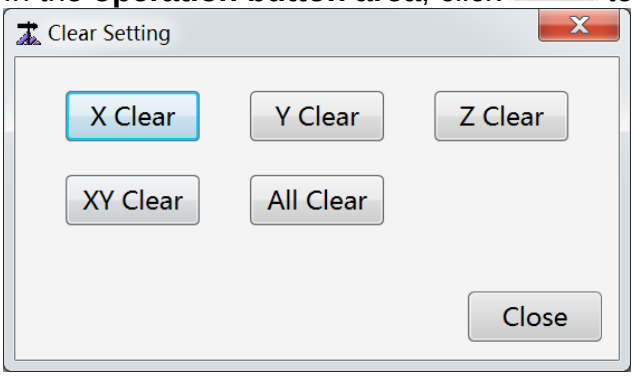
- In the **drawing tool bar**, click  to open the **Set Drawing Origin** dialog box:



Use one of the following methods to set the position of the drawing origin:

- Select from the system options:
  A. Select one from **Top Left**, **Top Center**, **Top Right**, etc.
  B. Click **OK**.
- Select a position by mouse:
  A. Click the **Set by Mouse** button. The mouse pointer becomes  .
  It is recommended that you enable the capture function first before setting the drawing origin position by mouse.
  B. Click the left mouse button on the target position.
  C. Click the right mouse button to exit.

2. In the **machine control area**, click **X+/X-/Y+/Y-** to move the X axis or Y axis to the target position.
3. In the **operation button area**, click  to open the **Clear Setting** dialog box:



4. Click **X Clear**/**Y Clear**/**XY Clear** to set the X-axis, Y-axis, or both X-axis and Y-axis coordinates of the current position to 0.
5. Click **Close**.

7.3.4 Set Slow Spot

This function is used to add slow spots/deceleration points before and after the target position to make the system execute machining at the deceleration rate.

Only applicable to the back water technic.

Follow the steps below to set slow spots:

1. Select the target shape. Use one of the following methods to use the slow spot tool:
   - In the **drawing tool bar**, click .
   - In the menu bar, go to **Object** > **Slow Spot**.
2. Click the left mouse button at the target position to add a slow spot.

If is displayed, a slow spot has been added in the position.

7.3.5 Set Machining Sequence

This function is used to assign the machining sequence for the shapes in the file.

Click or go to **View** > **Show Order** in the menu bar to show the machining sequence of all the shapes in the file.

You can set the machining sequence with the following two methods:

- By tool No.: A tool is changed after it has finished all its path to save tool change time and improve machining efficiency.
  - Ascending by the cutting tool No.: Generates all cutting paths (including the hole opener technic).
  - Ascending by the grinding tool No.: Generates all grinding paths.

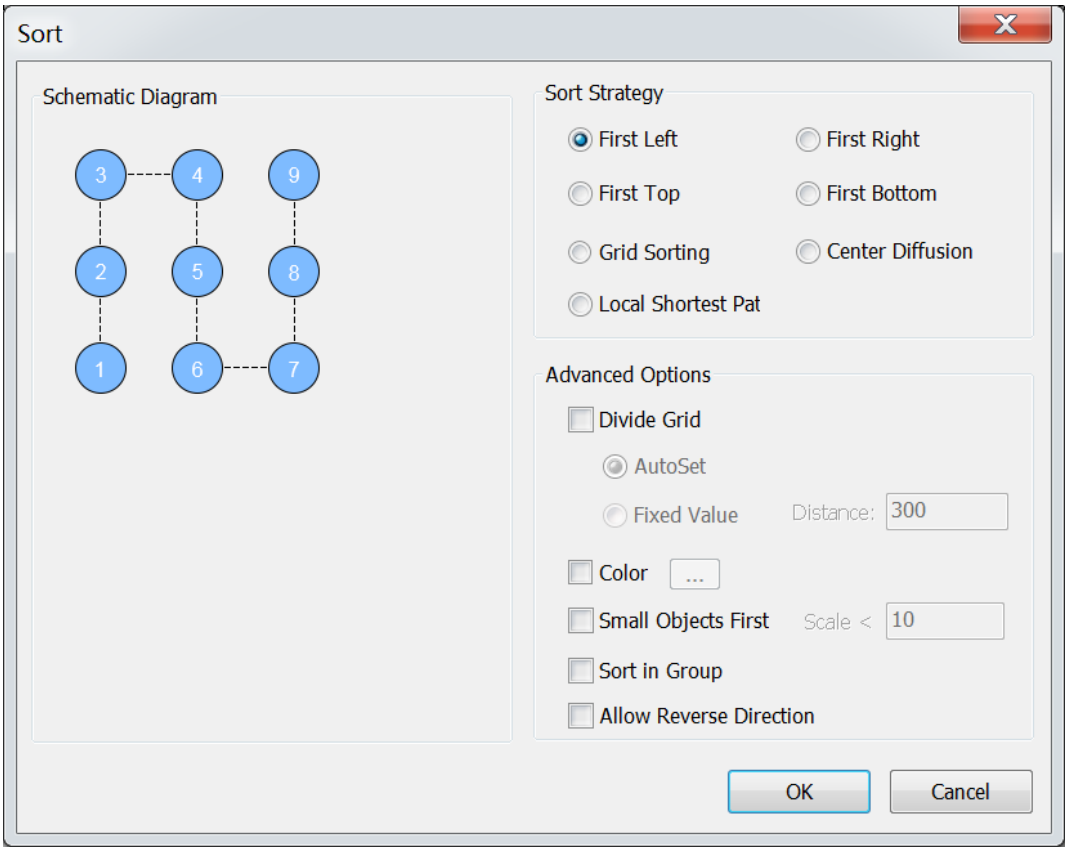
  For details, see description about the parameters in the **Cut** area and **Grind** area in Set Technic Parameters.
- By shape: The system supports the following methods of setting custom machining sequence:
  - Auto Set Order
  - Manual Set Order
  - Specify Order
  - Order List
  - Manual Draw
  - Sort to Top or Sort to Bottom
  - Group Set Order

7.3.5.1 Auto Set Order

The system sets the machining sequence based on the parameter setting.

Follow the steps below to use the auto set function:

1. Select the target shapes.
2. Use one of the following methods to open the **Sort** dialog box:
   - In the menu bar, go to **Object** > **Mach Order** > **Auto Set Order**.
   - Click the right mouse button and go to **Mach Order** > **Auto Sorting**.



3. Select a sort strategy.
4. **Optional**: Set parameters in the **Advanced Options** area.
    - **Divide Grid**: A directional sequencing strategy.
        - **AutoSet**: The system automatically divides the shapes into grids based on the their distance and number.
        - **Fixed Value**: Shapes whose distance from each other is smaller than the **Distance** value will be divided into the same grid.
    - **Color**: The shapes are sequenced based on the layer sequence. Only applicable to nested shapes inside components.
    - **Small Objects First**: Puts shapes whose sizes are smaller than the set value first. Only applicable to nested shapes inside components.
    - **Sort in Group**: Shapes inside different groups may have different machining sequence based on the selected sort strategy.
    - **Allow Reverse Direction**: Allow the shape machining sequence to be reversed.
5. Click **OK**.

7.3.5.2 Manual Set Order

Manually specify the machining sequence for the shapes.

Follow the steps below to use the manual set function:

1. Select the target shapes. Use one of the following methods to enable manual set:
    - In the menu bar, go to **Object** > **Mach Order** > **Manual Set Order**.
    - Click the right mouse button and go to **Mach Order** > **Manual Sorting**.

    The mouse pointer becomes  and the current machining sequence is displayed.
2. Click the shape you want to be processed first.

The mouse pointer becomes  and the number beside the shape becomes 1. The numbers beside other shapes become 2, 3,... respectively in the original sequence.
To reset a shape's sequence, click the right mouse button and select **Previous Order**.

3. Click the rest of the shapes in the desired machining sequence.
4. Use one of the following methods to exit manual sequencing:
   - Click the right mouse button and select **Exit**.
   - Press **Esc**.

7.3.5.3 Specify Order

Manually specify the machining sequence for the selected single shape.

Follow the steps below to use the specifying order function:

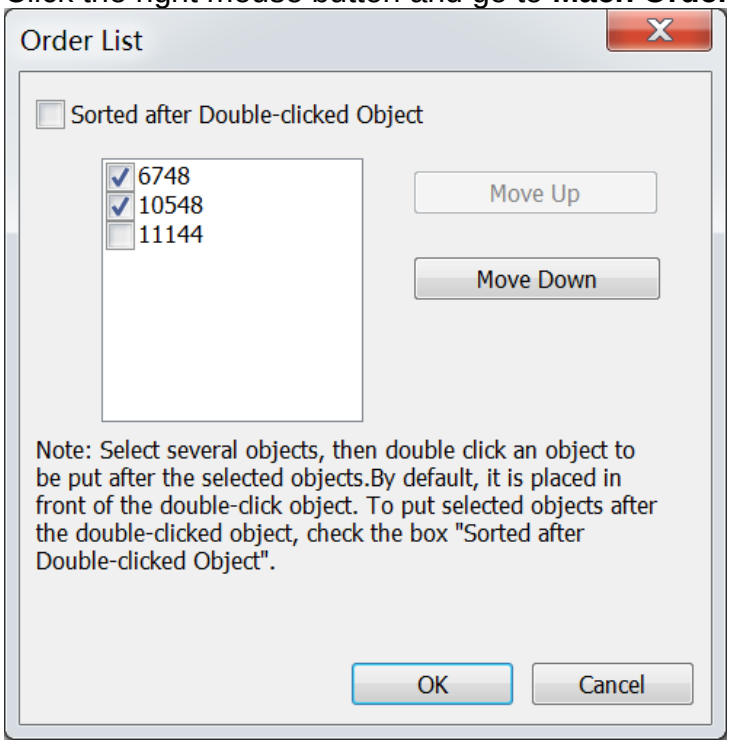
1. Select the target shape.
2. Use one of the following methods to open the **Specify Order Separately** dialog box:
   - In the menu bar, go to **Object** > **Mach Order** > **Specify Order**.
   - Click the right mouse button and go to **Mach Order** > **Specify Order Separately**.
3. Enter the sequence number in the **Number [1,n]** field.
   "N" represents the maximum sequence number in the current file.
4. Click **OK**.

7.3.5.4 Order List

This function is used to set the machining sequence based on the shape number.

Follow the steps below to use the order list function:

1. Select the target shape(s).
2. Use one of the following methods to open the **Order List** dialog box:
   - In the menu bar, go to **Object** > **Mach Order** > **Order List**.
   - Click the right mouse button and go to **Mach Order** > **Order List**.



3. Tick one or multiple shape numbers and use one of the following methods to adjust their sequence:
    - Click **Move Up**/**Move Down**.
    - Double-click an unselected shape number to move it down.
4. Click **OK**.

7.3.5.5 Manual Draw

This function is used to manually exchange the shape sequence numbers.

Follow the steps below to use the manual draw function:

1. Use one of the following methods to enable manual draw:
    - In the menu bar, go to **Object** > **Mach Order** > **Manual Draw**.
    - Click the right mouse button and go to **Mach Order** > **Manual Draw Sort**.

   The shapes are connected by green dotted lines.
2. Move the mouse pointer to a shape edge. Click and hold the mouse left button and drag it to another shape's edge until a green highlight line between the two shapes is displayed. Release the mouse button to change their machining sequence (as indicated by the direction of the arrow on the connecting green dotted line).

7.3.5.6 Sort to Top or Sort to Bottom

This function is used to make the selected single shape be processed first or last.

Follow the steps below to use the sort to top/bottom function:

1. Select the target shape.
2. Use one of the following methods to use the function:
    - In the menu bar, go to **Object** > **Mach Order** > **Sort to Top**/**Sort to Bottom**.
    - Click the right mouse button and go to **Mach Order** > **Sort To the Top**/**Sort To the Bottom**.

The machining sequence of the selected shape becomes the first/last.

7.3.5.7 Group Set Order

This function is used to set the machining sequence for shapes inside a group.

Prerequisite:

The target shapes are in the same group.

Follow the steps below to use the function:

1. Select the target shapes.
2. Use one of the following methods to open the **Sort** dialog box:
    - In the menu bar, go to **Object** > **Mach Order** > **Group Set Order**.
    - Click the right mouse button and go to **Mach Order** > **Group Sort**.

For details, see operations in Auto Set Order.

7.3.5.8 Set Machining Direction

This function is used to display or change the machining direction of the path. Two methods are available:

- **Reverse**: Reverses the current machining direction.
- **Set**: Smart change of the machining direction of the closed shapes

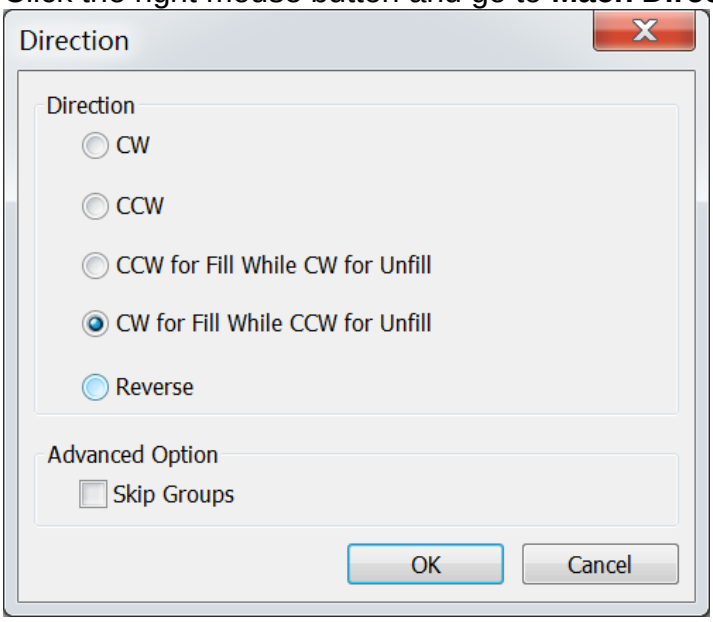
Follow the steps below to change the machining direction:

1. Select the target shape(s).
2. Use one of the following methods to show the machining direction:

- In the menu bar, go to **View** > **Show Dir**.
- In the **drawing tool bar**, click  .

3. Use one of the following methods to change the machining direction:
   - To reverse the machining direction:
     - In the menu bar, go to **Object** > **Mach Direction** > **Reverse**, or
     - Click the right mouse button and go to **Mach Direction** > **Reverse**.
   - To select a machining direction:

     A. Use one of the following methods to open the **Direction** dialog box:
     - In the menu bar, go to **Object** > **Mach Direction** > **Set**.
     - Click the right mouse button and go to **Mach Direction** > **Set**.



B. Select a machining direction among the system options.
C. **Optional**: To keep the machining direction of the shapes inside groups unchanged, tick **Skip Groups**.
D. Click **OK**.

8 Go to Mechanical Origin

The mechanical origin is the origin of the mechanical coordinate system of the machine. The mechanical coordinate system of a piece of machine is fixed and determined before out of factory.

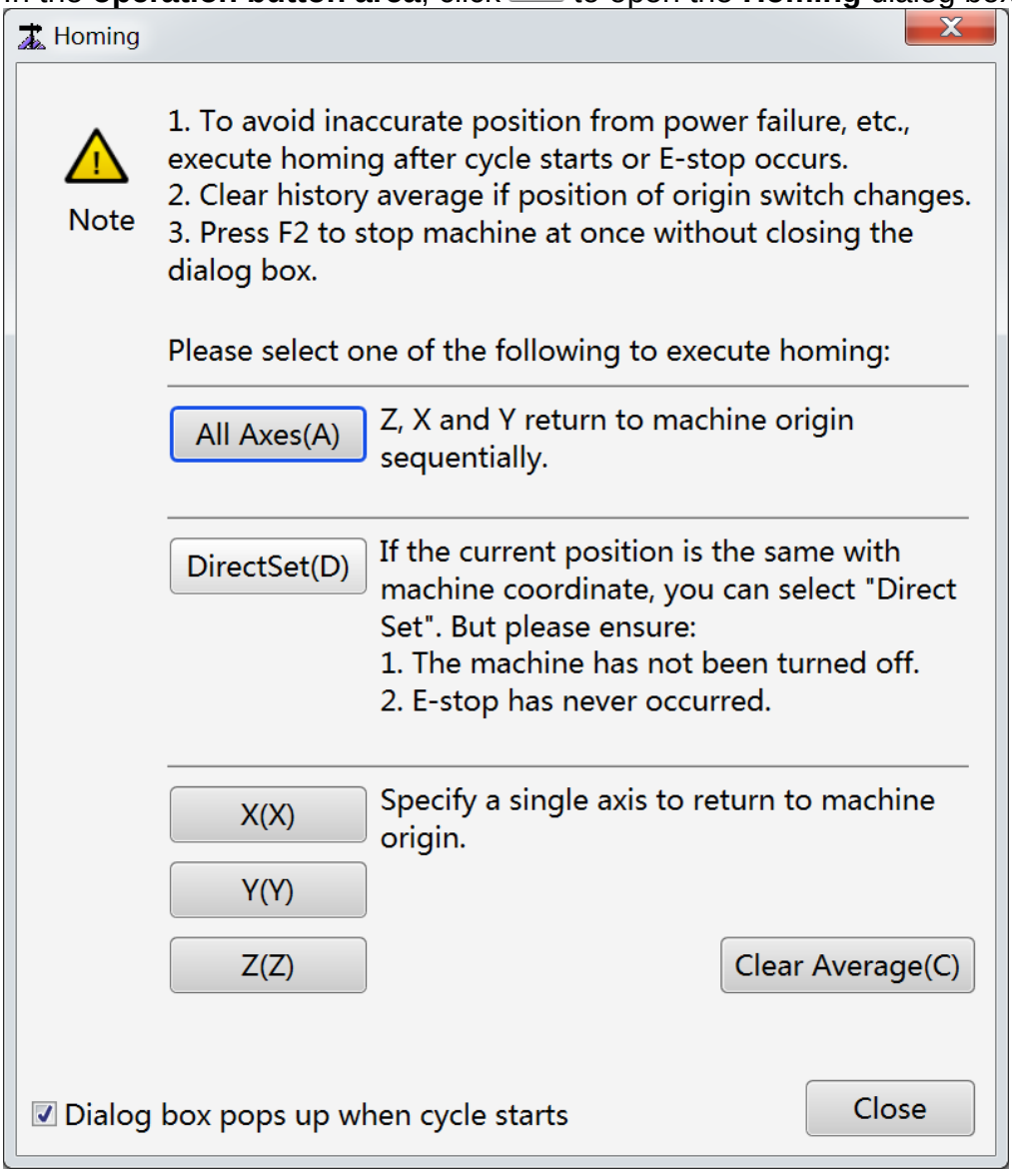
The purpose of making the axes go to the mechanical origin is to synchronize the system mechanical coordinate system with the machine mechanical coordinate system. The axes must go to the mechanical origin first before machining.

Prerequisite:

There are no servo motor alarms.

Follow the steps below to make an axis or all axes go to the mechanical origin:

1. In the **operation button area**, click  to open the **Homing** dialog box:



This dialog box is opened by default when the software is started. Unselect **Dialog box pops up when cycle starts** to stop showing it automatically when the software is started.

2. Use one of the following methods to make an axis or all axes go to the mechanical origin:
   - Click **All Axes** to make all the axes go to the mechanical origin in the sequence of Z axis, X axis, and Y axis.
   - Click **X**, **Y**, or **Z** to make the X axis, Y axis, or Z axis go to the mechanical origin.

     For safety considerations, it is recommended that you make the Z axis go to the mechanical origin first.

   - Click **DirectSet** to set the current point as the mechanical origin.

     Applicable when you are sure that the machine coordinates are correct (The mechanical coordinates of the current point are the same as the machine's actual mechanical coordinates, and the software was not closed nor does the machine have servo motor alarms or other exception that can cause machine position errors).

After an axis has gone to the mechanical origin successfully,  will be displayed before it in the **machine control area**.

9 Set Tool Parameters

You can use the software to manage the tools effectively to improve machining efficiency.

Tool parameters include

- Set Tool Compensation Parameters
- Set Wear Compensation Parameters
- Set Tool Offset Parameters

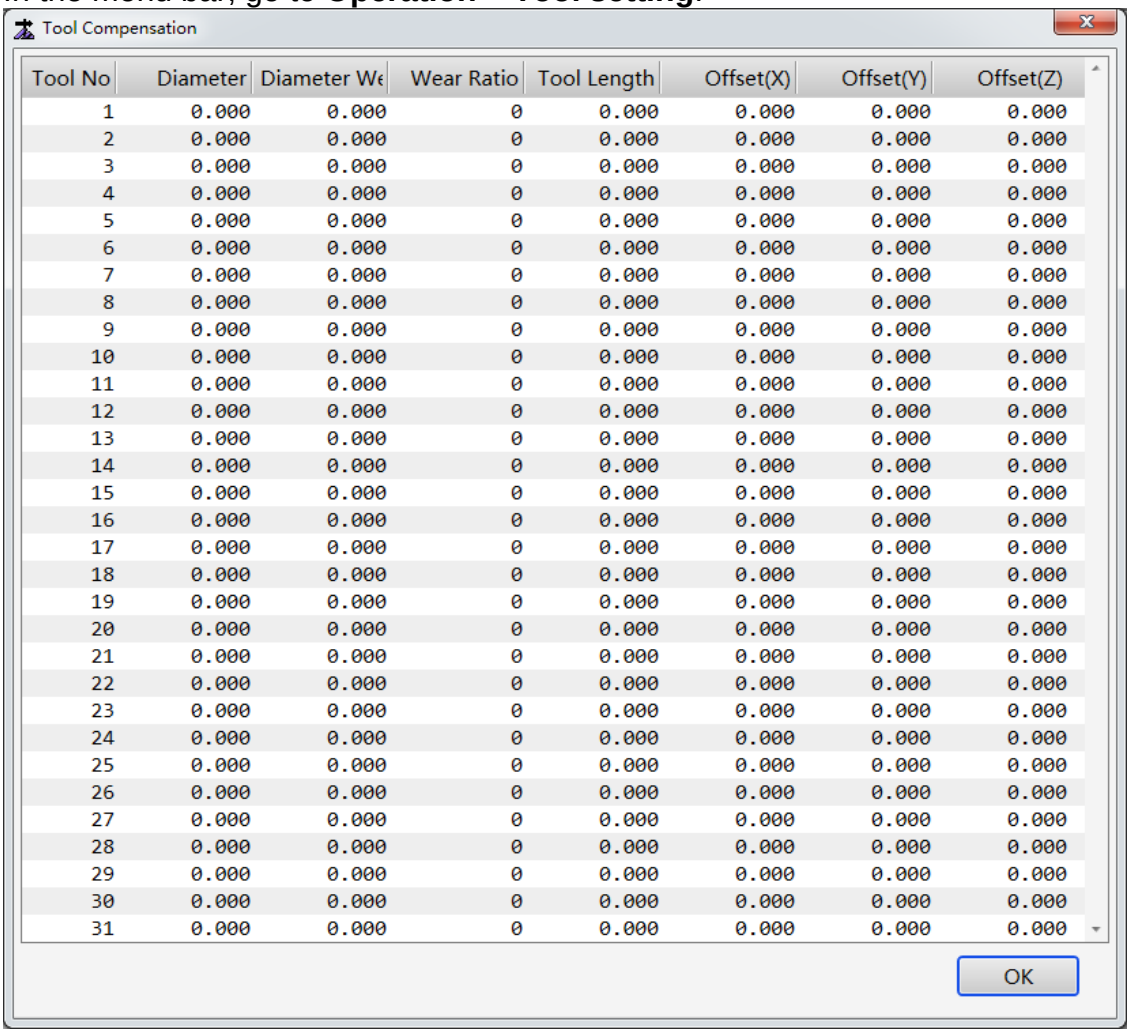
9.1 Set Tool Compensation Parameters

Used to compensate the difference between the actual tool position and the programmed tool position.

After the tool compensation parameters are set, if a tool is replaced, you only need to change the tool compensation value instead of modifying the machining program.

Follow the steps below to set tool compensation parameters:

1. In the menu bar, go to **System** > **Global Parameters**. In the **Param** window, click the first drop-down menu in the lower-left corner and select **Manufacturer**. Set the parameters **Enable Tool Length Compensation** and **Enable Tool Radius Compensation** to **Yes**.
2. In the menu bar, go to **Operation** > **Tool setting**.



Tool No	Diameter	Diameter We	Wear Ratio	Tool Length	Offset(X)	Offset(Y)	Offset(Z)
1	0.000	0.000	0	0.000	0.000	0.000	0.000
2	0.000	0.000	0	0.000	0.000	0.000	0.000
3	0.000	0.000	0	0.000	0.000	0.000	0.000
4	0.000	0.000	0	0.000	0.000	0.000	0.000
5	0.000	0.000	0	0.000	0.000	0.000	0.000
6	0.000	0.000	0	0.000	0.000	0.000	0.000
7	0.000	0.000	0	0.000	0.000	0.000	0.000
8	0.000	0.000	0	0.000	0.000	0.000	0.000
9	0.000	0.000	0	0.000	0.000	0.000	0.000
10	0.000	0.000	0	0.000	0.000	0.000	0.000
11	0.000	0.000	0	0.000	0.000	0.000	0.000
12	0.000	0.000	0	0.000	0.000	0.000	0.000
13	0.000	0.000	0	0.000	0.000	0.000	0.000
14	0.000	0.000	0	0.000	0.000	0.000	0.000
15	0.000	0.000	0	0.000	0.000	0.000	0.000
16	0.000	0.000	0	0.000	0.000	0.000	0.000
17	0.000	0.000	0	0.000	0.000	0.000	0.000
18	0.000	0.000	0	0.000	0.000	0.000	0.000
19	0.000	0.000	0	0.000	0.000	0.000	0.000
20	0.000	0.000	0	0.000	0.000	0.000	0.000
21	0.000	0.000	0	0.000	0.000	0.000	0.000
22	0.000	0.000	0	0.000	0.000	0.000	0.000
23	0.000	0.000	0	0.000	0.000	0.000	0.000
24	0.000	0.000	0	0.000	0.000	0.000	0.000
25	0.000	0.000	0	0.000	0.000	0.000	0.000
26	0.000	0.000	0	0.000	0.000	0.000	0.000
27	0.000	0.000	0	0.000	0.000	0.000	0.000
28	0.000	0.000	0	0.000	0.000	0.000	0.000
29	0.000	0.000	0	0.000	0.000	0.000	0.000
30	0.000	0.000	0	0.000	0.000	0.000	0.000
31	0.000	0.000	0	0.000	0.000	0.000	0.000

3. In the **Tool Compensation** window, double-click the target field. Enter a value in the input box. Descriptions of the parameters are shown below:

Parameter	Description
Tool No.	The tool sequential number. At most 31 tools are supported.
Diameter	Used for radius compensation.
Diameter Wear	Used for worn diameter compensation.
Wear Ratio	Indicates the worn diameter amount every time the tool processes 1-meter materials. Applicable to real-time wear compensation.
Offset(X)/(Y)/(Z)	Indicates the tool offset value. Manually enter a value in the field or use a tool sensor.

4. Click **OK**.

9.2 Set Wear Compensation Parameters

If the wear compensation function is enabled, the system makes real-time adjustment of the tool feed amount based on the wear amount.

Wear compensation is not needed during cutting.

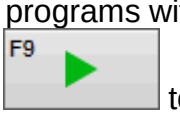
If the wear compensation function is enabled, the tool radius will have a certain deviation. You need to check to see if such deviation causes any intervention.

Follow the steps below to set wear compensation parameters:

1. In the menu bar, go to **System** > **Global Parameters**. In the **Param** window, change the status of **Enable Tool Radius Compensation** and **Enable wear compensation** based on the description below:

Enable Tool Radius Compensation	Enable Wear Compensation	Description
No	Yes	Real-time tool wear compensation is disabled.
Yes	No	Real-time tool wear compensation is disabled. Tool compensation for general cases is used.
Yes	Yes	If tool diameter compensation commands **G41**/**G42** is not included in the machining program, real-time tool wear compensation will not be enabled.

   If the wear compensation conditions are not met, a prompt indicating that the tool diameter is too short will be displayed.
2. Set the parameter **Enable Tool Length Compensation** based on the actual situation.

3. Load programs with **G41**/**G42**/**G40** tool diameter compensation commands.
4. Click 

to start machining.
    - After the **G41**/**G42** commands are processed, real-time tool diameter wear compensation is enabled.
    - After the **G40** command is processed, real-time tool wear compensation is disabled.
5. Click  or  to save the file in the original path or another path.
   The system determines if wear compensation will be enabled based on the wear ratio. If the wear ratio is not 0, wear compensation will be enabled.

9.3 Set Tool Offset Parameters

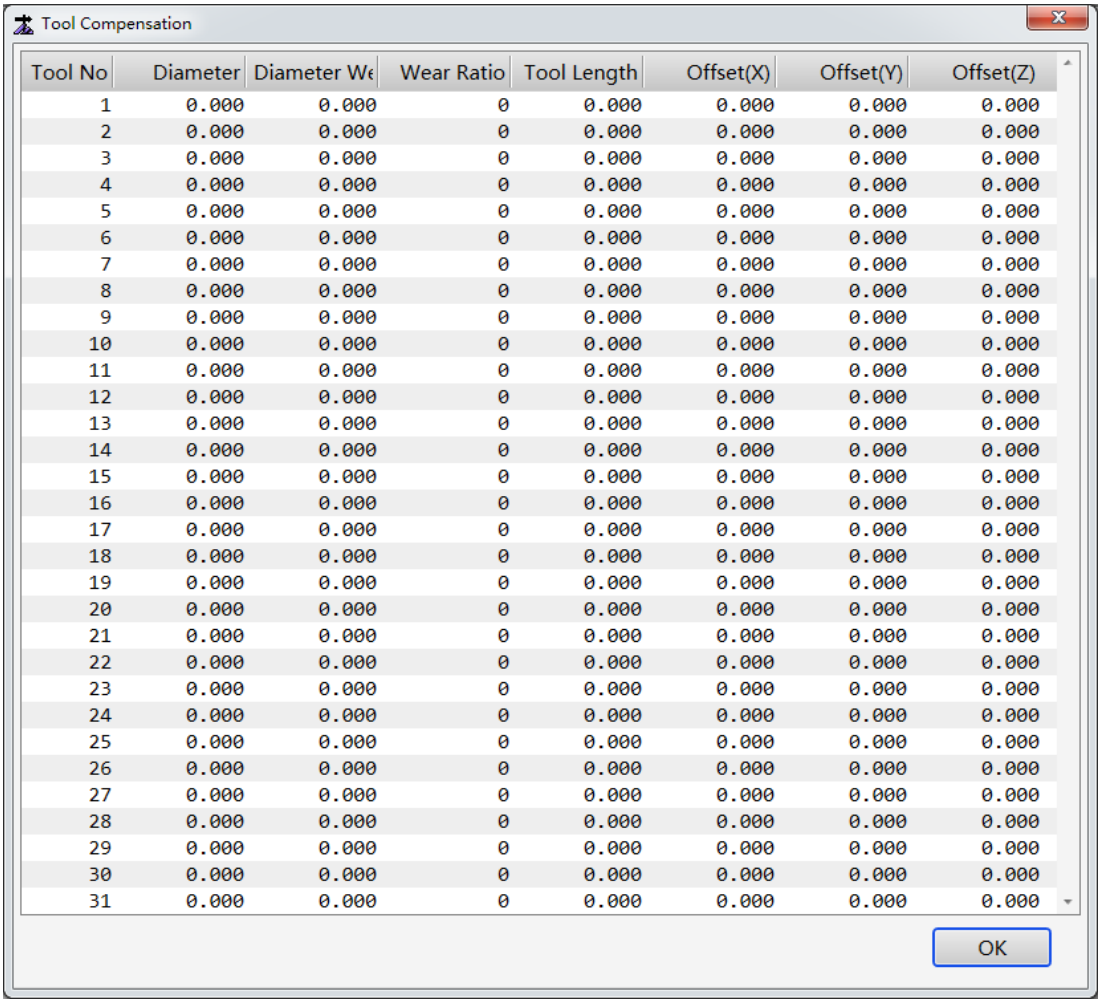
Used to make sure that the tools are lowered to the correct height during machining.

You can set the tool offset parameters with the following methods:

- Enter the Z-axis offset value.
- Use the current point Z-axis mechanical coordinate as the Z-axis offset.
- Use the tool calibration result as the Z-axis offset.

Follow the steps below to **enter the offset value**:

1. In the menu bar, go to **Operation** > **Tool setting** to open the **Tool Compensation** window:

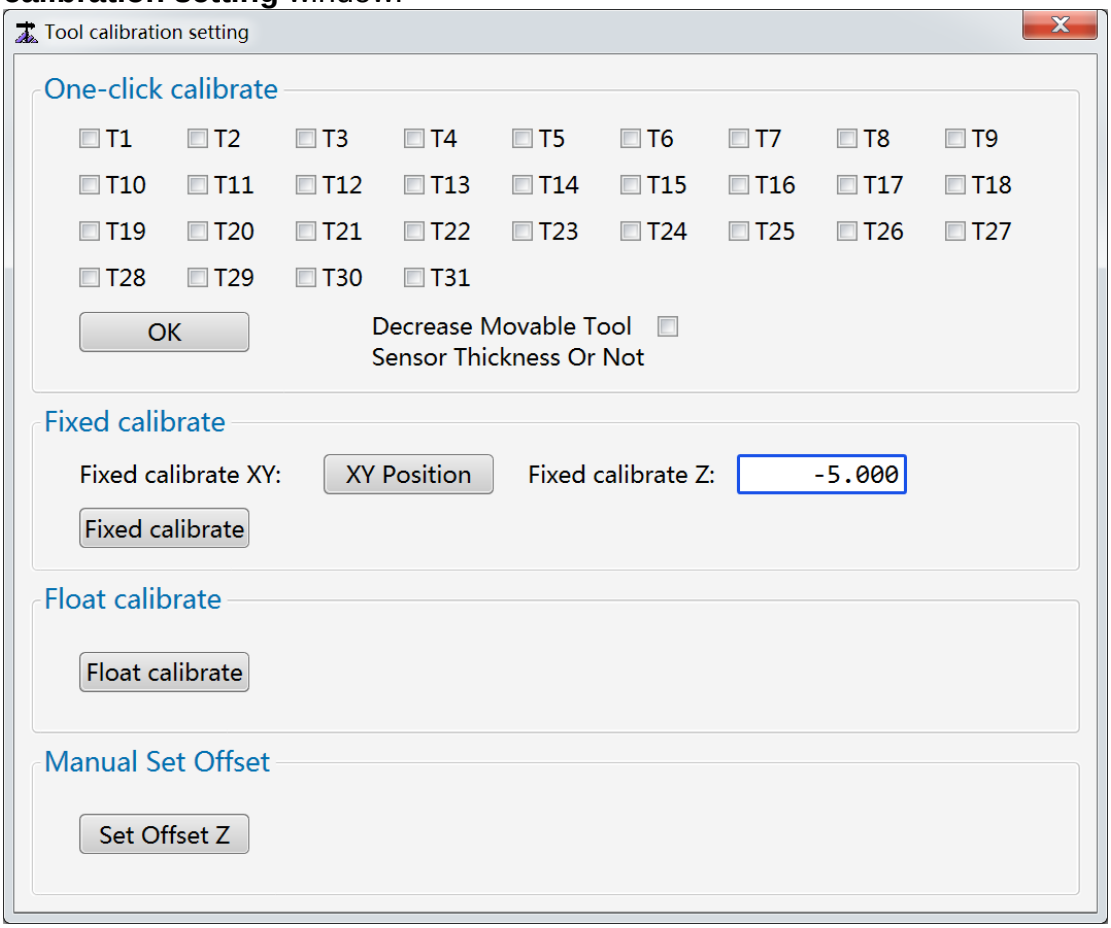


Tool No	Diameter	Diameter We	Wear Ratio	Tool Length	Offset(X)	Offset(Y)	Offset(Z)
1	0.000	0.000	0	0.000	0.000	0.000	0.000
2	0.000	0.000	0	0.000	0.000	0.000	0.000
3	0.000	0.000	0	0.000	0.000	0.000	0.000
4	0.000	0.000	0	0.000	0.000	0.000	0.000
5	0.000	0.000	0	0.000	0.000	0.000	0.000
6	0.000	0.000	0	0.000	0.000	0.000	0.000
7	0.000	0.000	0	0.000	0.000	0.000	0.000
8	0.000	0.000	0	0.000	0.000	0.000	0.000
9	0.000	0.000	0	0.000	0.000	0.000	0.000
10	0.000	0.000	0	0.000	0.000	0.000	0.000
11	0.000	0.000	0	0.000	0.000	0.000	0.000
12	0.000	0.000	0	0.000	0.000	0.000	0.000
13	0.000	0.000	0	0.000	0.000	0.000	0.000
14	0.000	0.000	0	0.000	0.000	0.000	0.000
15	0.000	0.000	0	0.000	0.000	0.000	0.000
16	0.000	0.000	0	0.000	0.000	0.000	0.000
17	0.000	0.000	0	0.000	0.000	0.000	0.000
18	0.000	0.000	0	0.000	0.000	0.000	0.000
19	0.000	0.000	0	0.000	0.000	0.000	0.000
20	0.000	0.000	0	0.000	0.000	0.000	0.000
21	0.000	0.000	0	0.000	0.000	0.000	0.000
22	0.000	0.000	0	0.000	0.000	0.000	0.000
23	0.000	0.000	0	0.000	0.000	0.000	0.000
24	0.000	0.000	0	0.000	0.000	0.000	0.000
25	0.000	0.000	0	0.000	0.000	0.000	0.000
26	0.000	0.000	0	0.000	0.000	0.000	0.000
27	0.000	0.000	0	0.000	0.000	0.000	0.000
28	0.000	0.000	0	0.000	0.000	0.000	0.000
29	0.000	0.000	0	0.000	0.000	0.000	0.000
30	0.000	0.000	0	0.000	0.000	0.000	0.000
31	0.000	0.000	0	0.000	0.000	0.000	0.000

2. Double-click the **Offset(Z)** field of the target tool and enter the Z-axis offset value in the input box.
3. Click **OK**.

Follow the steps below to **use the current Z-axis mechanical coordinate as the Z-axis offset**:

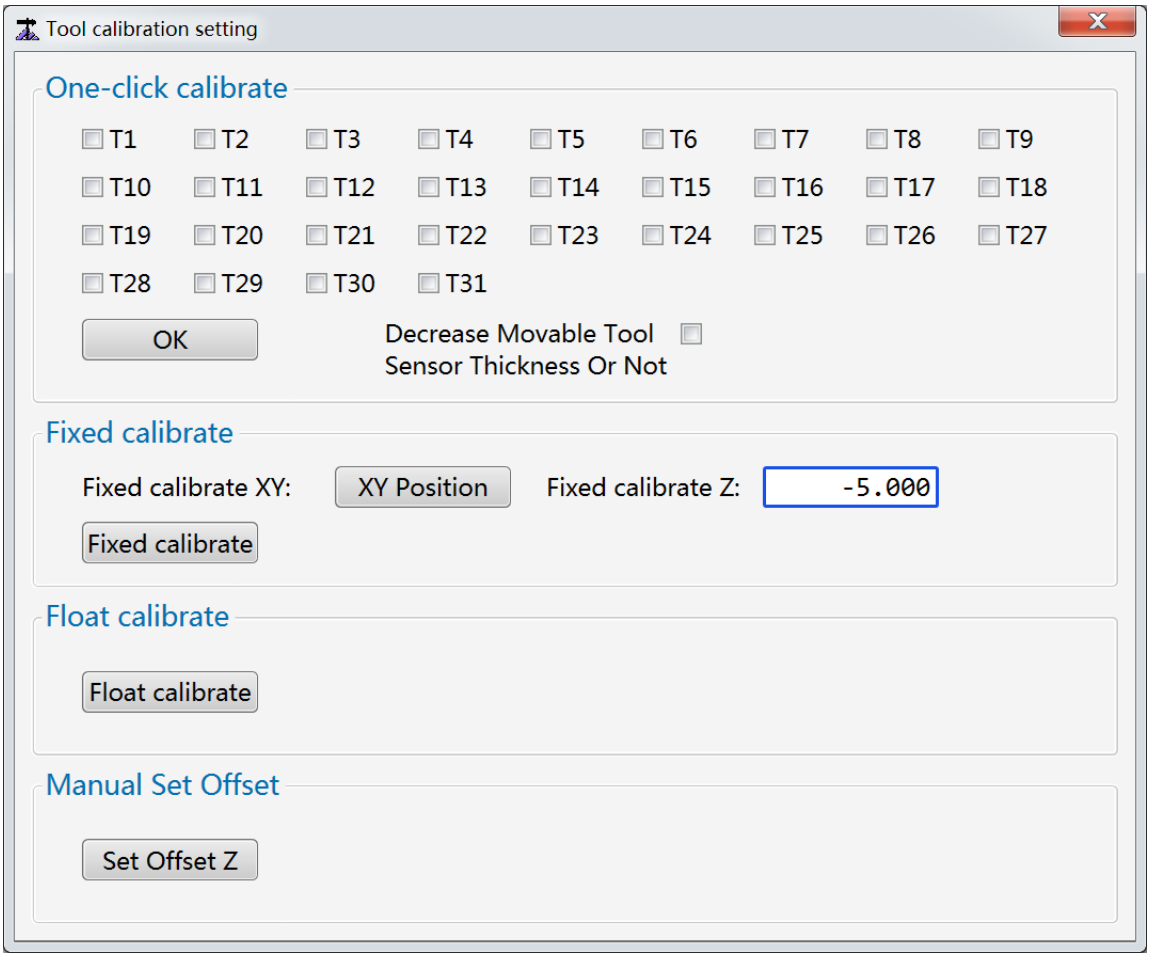
1. In the menu bar, go to **Operation** > **Tool calibration setting** to open the **Tool calibration setting** window:



2. Click **Set Offset Z** in the **Manual Set Offset** area to set the Z-axis offset of the current tool to the current Z-axis mechanical coordinate.

Follow the steps below to **use the tool calibration result as the Z-axis offset**:

1. In the menu bar, go to **Operation** > **Tool calibration setting** to open the **Tool calibration setting** window:



2. **Optional**: If **Decrease Movable Tool Sensor Thickness Or Not** is ticked, the Z-axis offset value will be the calibration result minus the tool sensor thickness.
3. Execute tool calibration based on your needs:
   - To execute tool change and calibration for multiple tools:
     A. Tick the target tool numbers in the **One-click calibrate** area.
     B. Click **OK**. The system will write the calibration results of the selected tools into their Z-axis tool offset.
   - To execute tool calibration at a fixed point on the machine for a single tool:
     A. Control the X axis and Y axis to move above the tool sensor. Click **XY Position**.
     B. Enter a value in the **Fixed calibrate Z** field.
     C. Click **Fixed calibrate**.
     The system will write the calibration result of the selected tool into its Z-axis tool offset.

10 Set Edge Finding and Centering Parameters

Used to calculate the plate angle relative to the mechanical coordinate system and rotate the workpiece coordinate system in the program to create a new one accordingly. The purpose is to avoid inaccurate positioning of the plate due to large plate size or weight and improve plate usage rate.

Two methods are available:

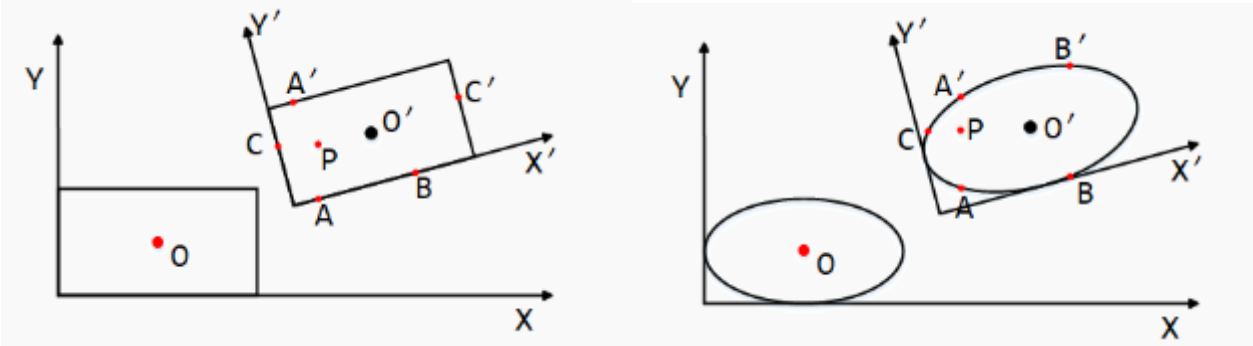
- Automatic Edge Finding and Centering
- Manual Edge Finding and Centering

10.1 Automatic Edge Finding and Centering

The system supports automatic edge finding and centering for rectangles and ellipses.

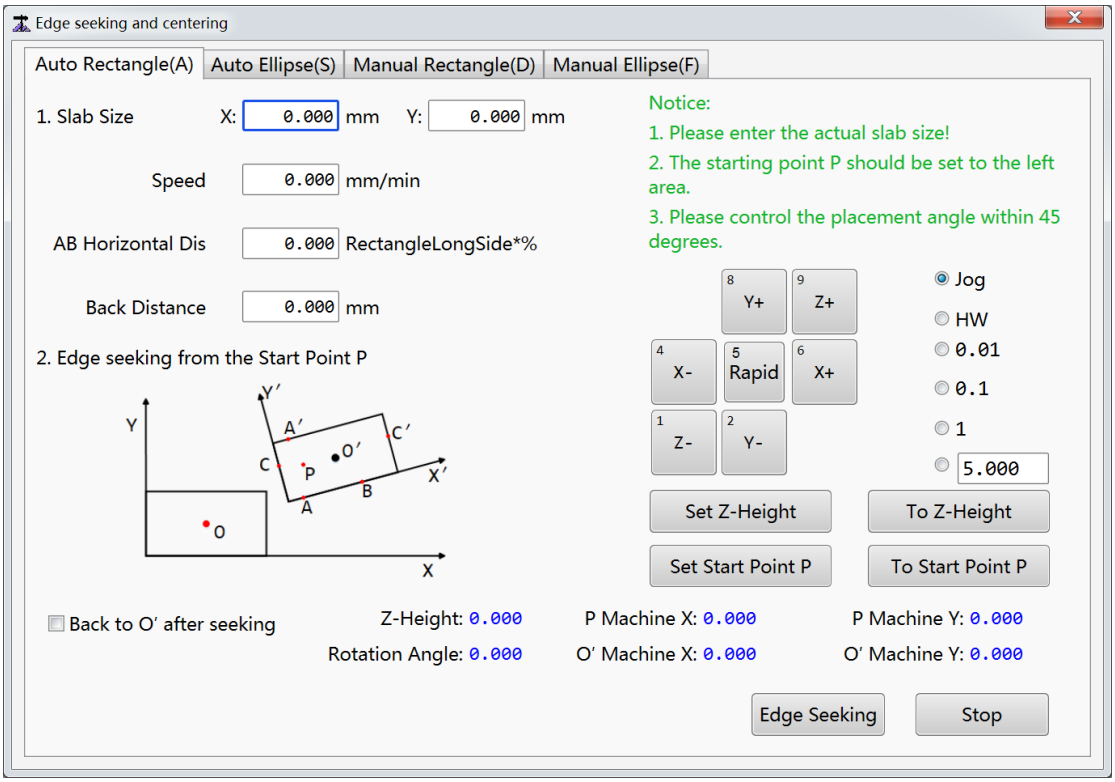
The sensor moves (along with the spindle) during automatic edge finding and centering as shown below:

1. It moves down from the start point P to point A.
2. It receives the edge finding signal, retreats for a certain distance, and moves horizontally to point B.
3. It receives signal at point B and goes back to point P. The rotation angle can be determined.
4. It moves left to point C and down to point A.
5. It moves from point A to point A′. The Y-axis mechanical coordinate of point O′ Y can be determined.
6. It moves from point A′ to point C, and then to point C′. The X-axis mechanical coordinate of point O′ X can be determined. After the edge finding process is complete, the system automatically starts machining.



Follow the steps below to use the automatic edge finding and centering function:

1. In the **drawing tool bar**, click  to open the **Edge seeking and centering** window:



2. Set the parameters in the **Auto Rectangle** and **Auto Ellipse** pages based on the shape. The description of the parameters is shown below:

Parameter	Description
X	The plate X-axis length.
Y	The plate Y-axis length.
Speed	The edge finding feed speed.
AB Horizontal Dis	Horizontal distance between point A and point B. The value should be smaller than the **X** field value
Back Distance	The retreating distance after receiving the edging finding signal at point A.
Set Z-Height and **Set Start Point P**	Click the axis direction buttons to move the axes. Move the Z axis to the safe height. It is recommended to set point P left to the center point at an angle smaller than 45 degrees.

3. Click **Edge Seeking** to start edge finding simulation.
4. **Optional**: To make the cutter return to point O′ after automatic edge finding, tick **Back to O′ after seeking**.
5. Click  to close the **Edge seeking and centering** window.

Related tasks:

To start machining directly after automatic edge finding:

1. In the **port button area**, click **RectangleSeek/Ellipse Seek**.

2. In the **machining operation area**, click 

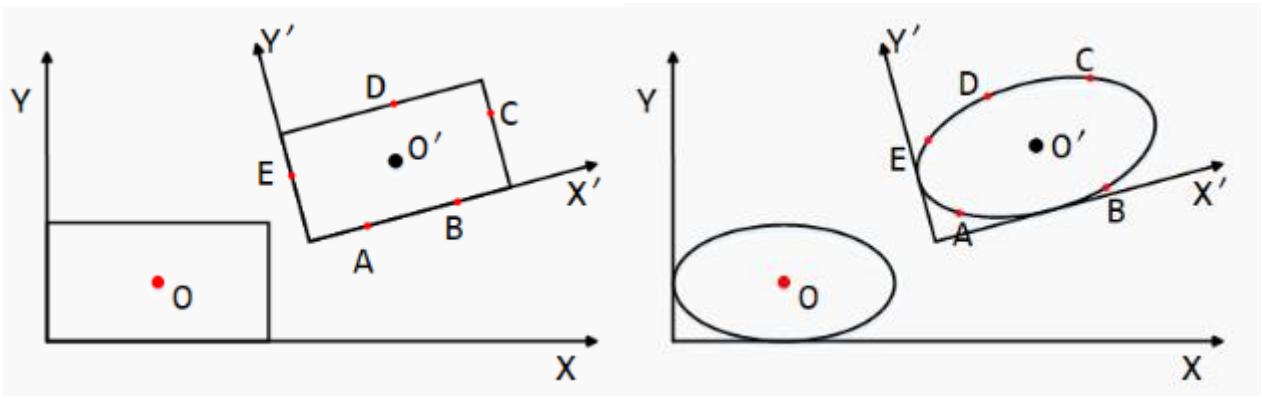
.

10.2 Manual Edge Finding and Centering

The system supports manual edge finding and centering for rectangles and ellipses.

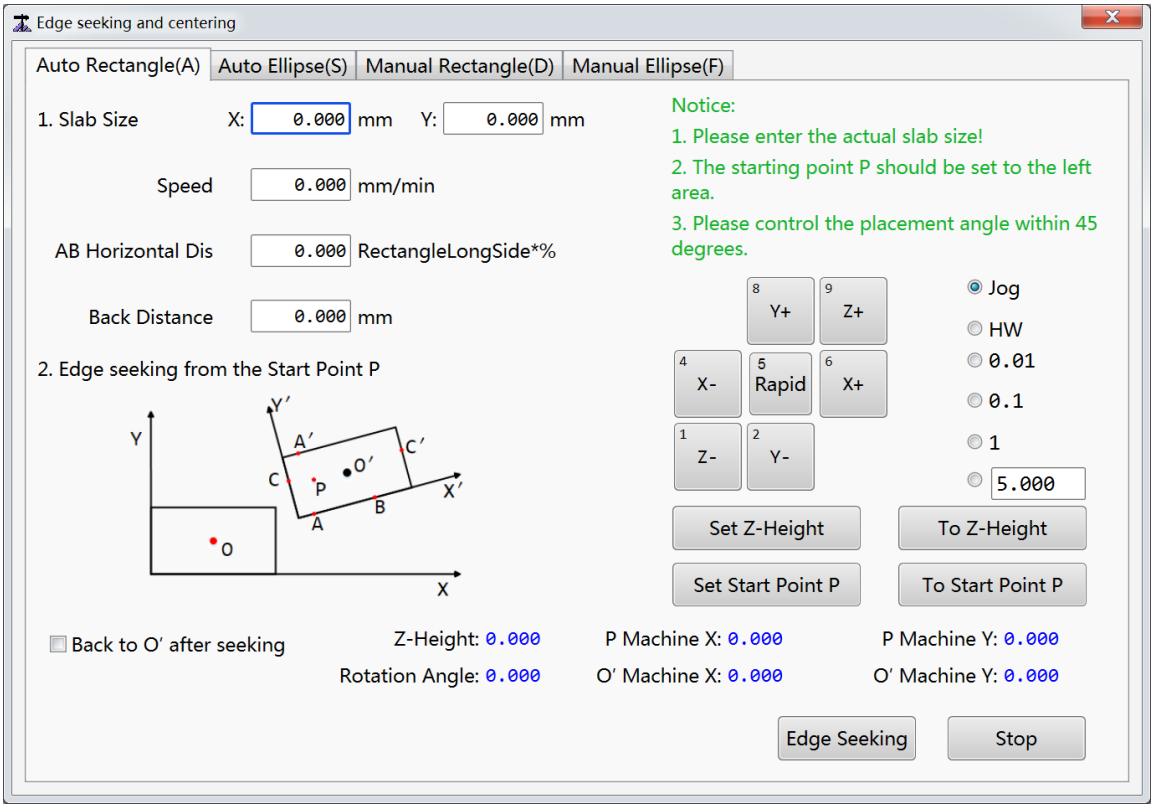
The principle for manual edge finding and centering is shown below:

Mark point A, B, C, D, and E on the work piece. Manually control the cutter to move to the five points to record their coordinates into the software. Click **Calculate** to make the system calculate the rotation angle and the mechanical coordinates of point O′.

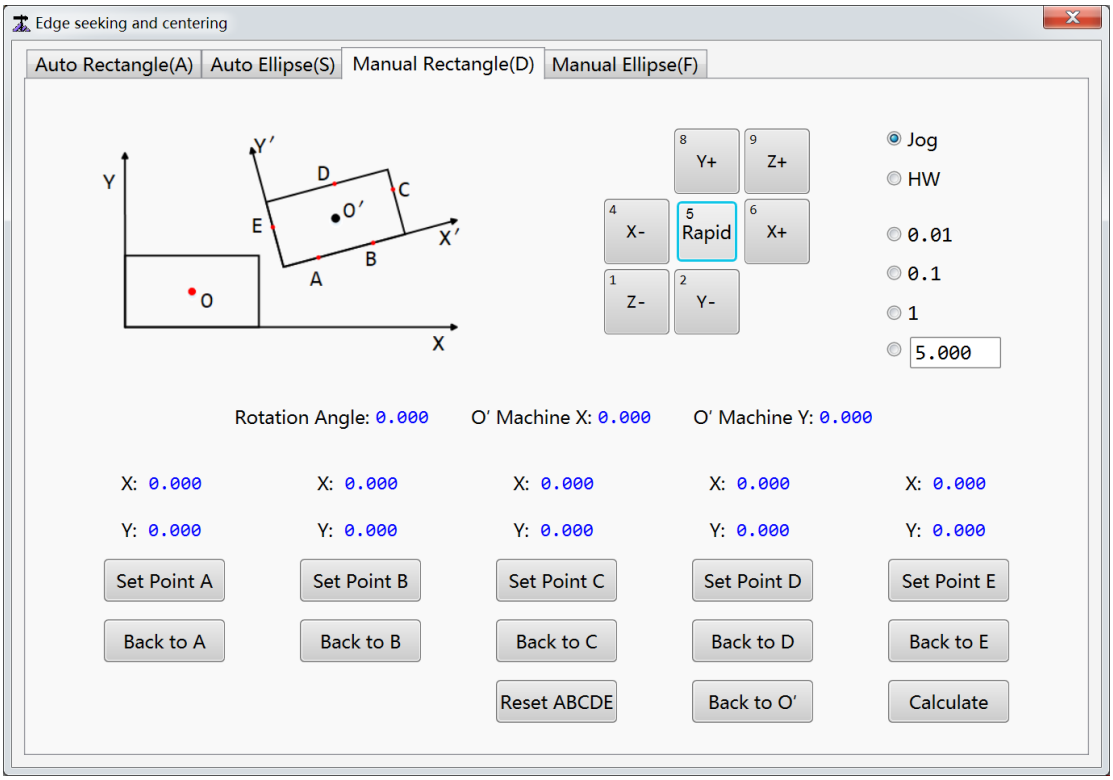


Follow the steps below to set manual edge finding and centering parameters:

1. In the **drawing tool bar**, click  to open the **Edge seeking and centering** dialog box:



2. Click the **Manual Rectangle** or **Manual Ellipse** tab based on the shape. The **Manual Rectangle** screen is shown below as an example:



3. Click the X or Y direction buttons (**X+**, **X-**, **Y+**, **Y-**) to move the  in the drawing area to the target position for point A/B/C/D/E.
4. Click the corresponding **Set Point A/Set Point B/Set Point C/Set Point D/Set Point E** button.
5. Click **Calculate**. The rotation angle and X-axis and Y-axis mechanical coordinates of point O′ are displayed in the window.
6. **Optional**: To make the cutter go to the position of point O′, click **Back to O′**.
7. Click  to close the **Edge seeking and centering** window.

Related tasks:

To start machining directly after manual edge finding:

1. In the **port button area**, click **Manual Seek**.
2. In the **machining operation area**, click 

.

11 Machining Methods and Statistics

The system supports multiple machining methods and viewing of machining statistics:

- Simulation
- Machining Wizard
- Selective Machining
- Advanced Machining
- Machining by Commands
- View Machining Statistics

11.1 Simulation

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

Follow the steps below to use simulation:

1. In the **operation button area**, click **Simulation** to make it highlighted in green.
2. In the **machining operation area**, click 

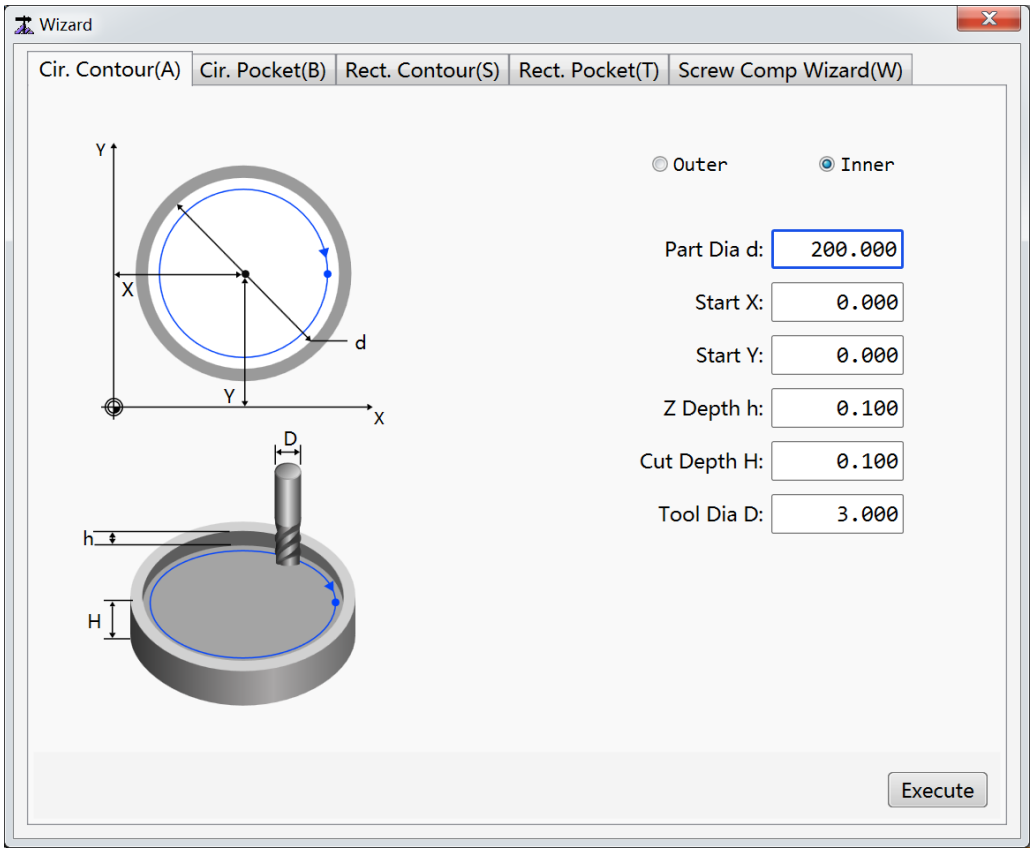
to start simulation.

11.2 Machining Wizard

The machining wizard is used to generate machining path automatically. The system supports different shapes in the **Cir. Contour**, **Cir. Pocket**, **Rect. Contour**, **Rect. Pocket**, and **Screw Comp Wizard** screen.

Follow the steps below to use machining wizard:

1. Follow corresponding steps based on the shape.
    - For circle edges, circle pockets, rectangle edges, rectangle pockets, and screws:
      A. In the menu bar, go to **Program** > **Machining Wizard** to open the **Wizard** window:

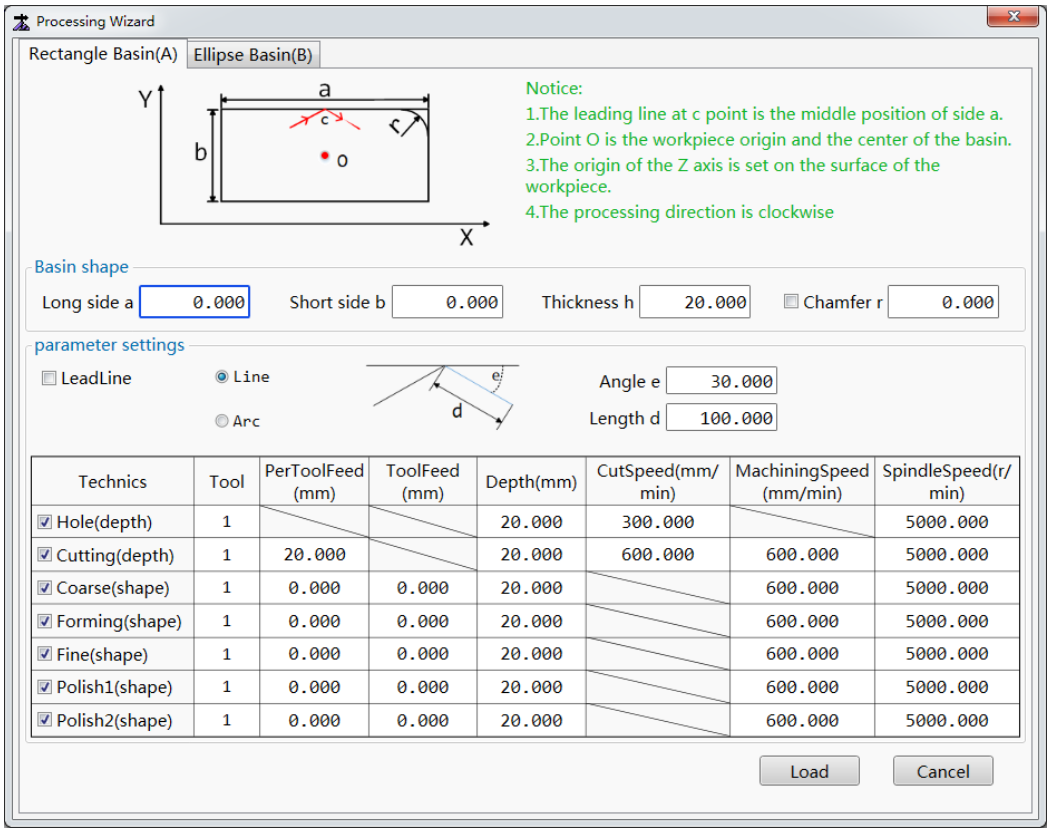


B. Click the **Cir. Contour**, **Cir. Pocket**, **Rect. Contour**, **Rect. Pocket**, or **Screw Comp Wizard** tab based on your needs. Set related parameters as indicated by the left figure.

C. Click **Execute**. The system starts machining.

- For rectangle basins and ellipse basins:

  A. In the **drawing tool bar**, click  to open the **Processing Wizard** window:



B. Click the **Rectangle Basin** or **Ellipse Basin** tab based on your needs. Set related parameters as indicated by the figure. Descriptions of the parameters are shown below:

- Parameters in the **Basin shape** area:

Parameter	Description
Long side a	The length of the workpiece.
Short side b	The width of the workpiece.
Thickness h	The height/thickness of the workpiece.
Chamfer r	The radius of the workpiece corner (when r=0, the workpiece is a rectangle).

- Lead line parameters:

Parameter	Description
Line	Straight lead lines.
Arc	Arc lead lines.
Angle e and **Length d**	The lead line angle and length.

- Technic parameters
  - Tick desired technic items (**Hole**, **Cutting**, **Coarse**, etc.)

- Set the tool number, tool feed, depth, cut speed, machining speed and spindle speed parameters.
- The tool is fed by the **PerToolFeed** amount every time until it reaches the **Depth** value.
- The **PerToolFeed** values for **Coarse**, **Forming**, **Fine**, and **Polish** technics are the tool horizontal feed amount, which are different from that for the **Cutting** technic.

2. After the parameters are set, click **Load** to save the file to the local path. The machining path will be displayed in the drawing area.
3. In the **machining operation area**, click 

to start machining.

Note

Machining path: The cutter goes to the middle point of edge a from the lead-in line, goes along the machining path clockwise once and then goes out from the middle point of edge a along the lead-out line.

11.3 Selective Machining

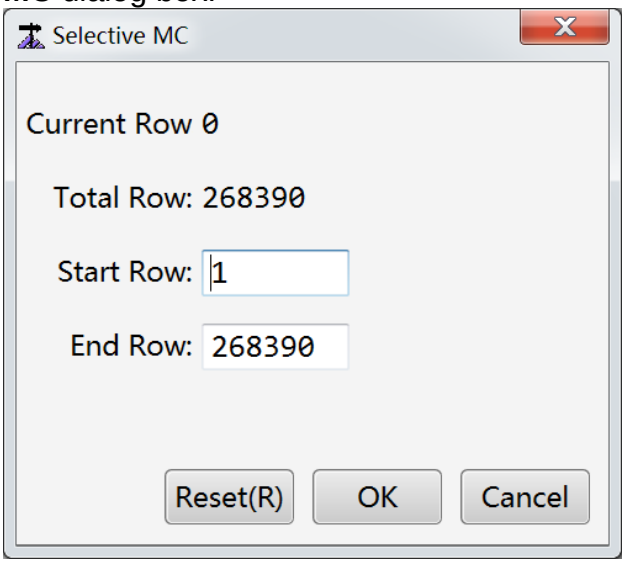
This function is used to select the program lines you want to be executed. The system only executes the selected lines.

Prerequisite:

The target program file is loaded into the software.

Follow the steps below to use the selective machining function:

1. In the menu bar, go to **Advanced** > **Selective Machining** to open the **Selective MC** dialog box:



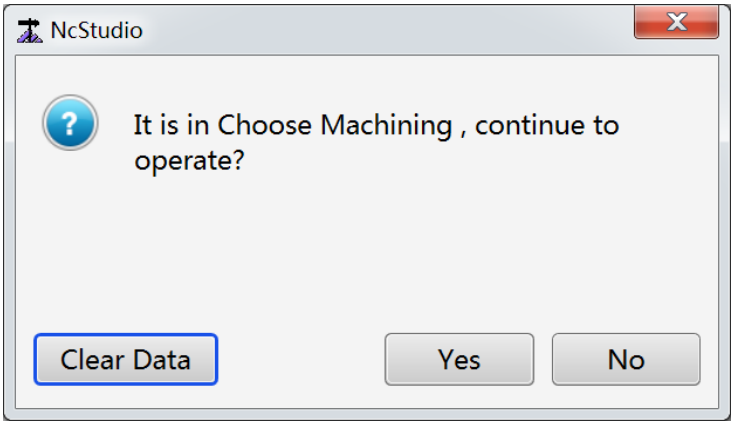
2. Enter values in the **Start Row** and **End Row** fields.
3. **Optional**: To change the starting row and ending row number, click **Reset**. The **Start Row** and **End Row** will be restored to their default values.

Note

The default starting row is the first row and the default ending row is the last row.

4. Click **OK**.
5. In the **machining operation area**, click 

. A prompt will be displayed, as shown below:



6. Click **Yes**.

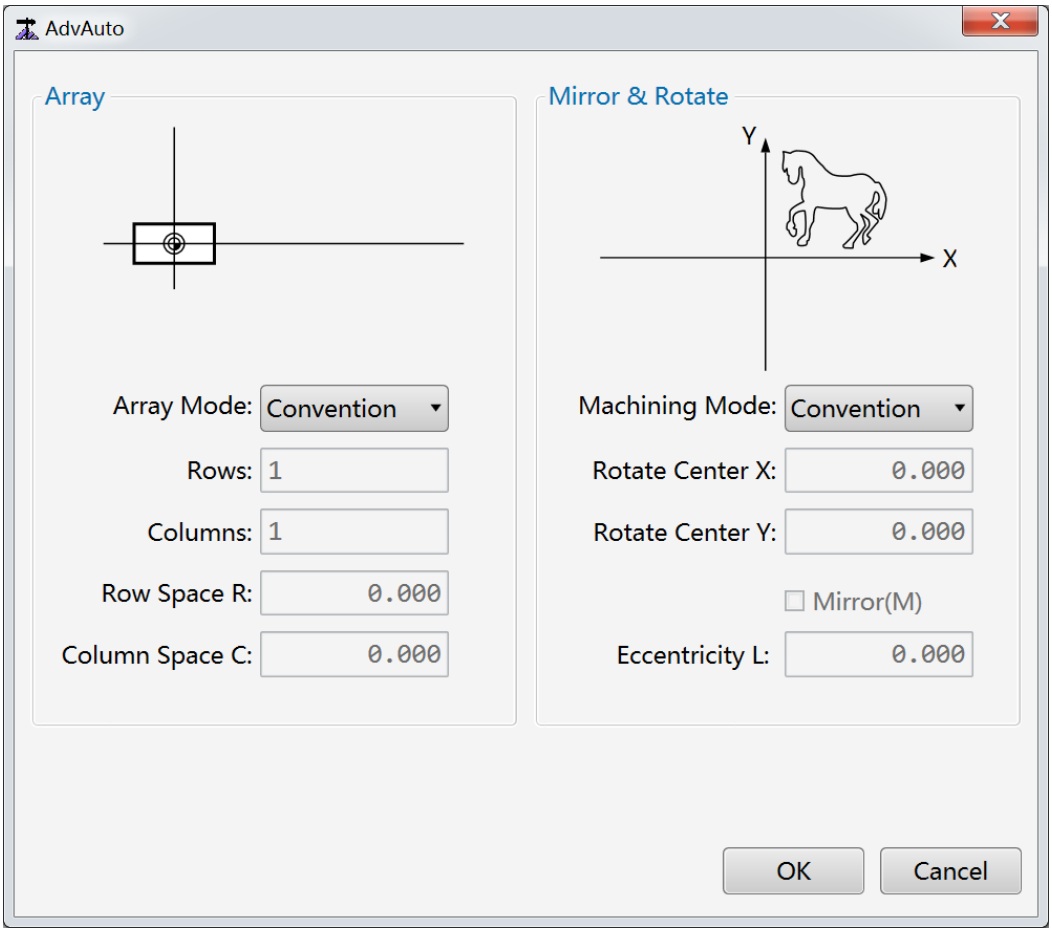
11.4 Advanced Machining

This function is used to create shape array and rotate or mirror a shape.

- A shape array is created by copying a shape along the rectangular matrix.
- Rotating a shape is to rotate it around a certain point by a certain degree.
- Mirroring a shape is to create its symmetrical shape over the X axis or Y axis.

Follow the steps below to use the advanced machining functions:

1. In the menu bar, go to **Advanced** > **AdvAuto** to open the **AdvAuto** window:



2. Follow corresponding steps based on your needs.
   - Array

   A. In the **Array** area, click the **Array Mode** drop-down menu. Select **Rectangular Array**. B. Set the **Rows**, **Columns**, **Row Space R**, and **Column Space C** parameters.
   - Rotation

   A. In the **Mirror & Rotate** area, click the **Machining Mode** drop-down menu. Select **CW 90°**, **CW 180°**, or **CCW 90°**. B. Enter the X and Y coordinates of the rotation center in the **Rotate Center X** and **Rotate Center Y** fields.
   - Mirroring

   A. In the **Mirror & Rotate** area, click the **Machining Mode** drop-down menu. Select **Mirroring by X** or **Mirroring by Y**. B. Tick **Mirror**. C. **Optional**: To adjust the position of the mirrored shape, adjust the **Eccentricity L** value.

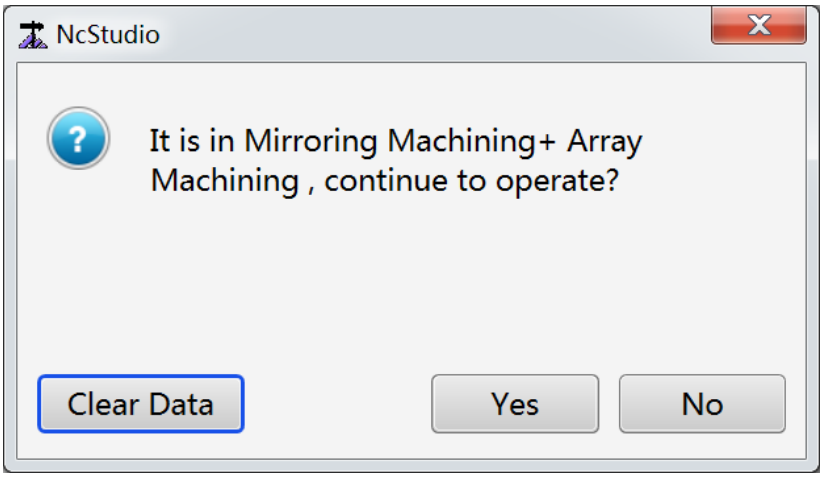
   Note

   The default array mode or machining mode is **Convention**, which means no array, rotation, or mirroring.
   Array settings and rotation/mirroring settings can be applied at the same time.

3. Click **OK**.

4. In the **machining operation area**, click 

. A prompt will be displayed, as shown below:



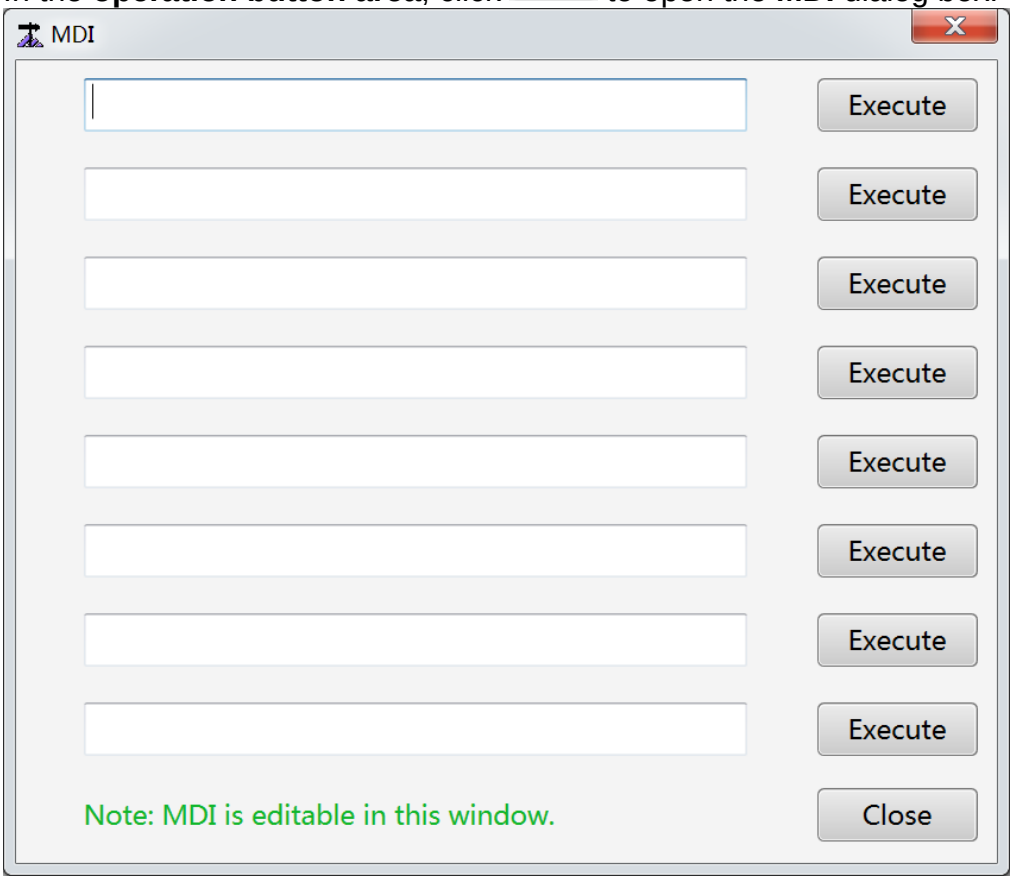
5. Click **Yes** to start machining.

11.5 Machining by Commands

The system supports entry and execution of at most 8 simple commands to achieve quick movement or simple machining tasks.

Follow the steps below to enter and execute machining commands:

1. In the **operation button area**, click  to open the **MDI** dialog box:



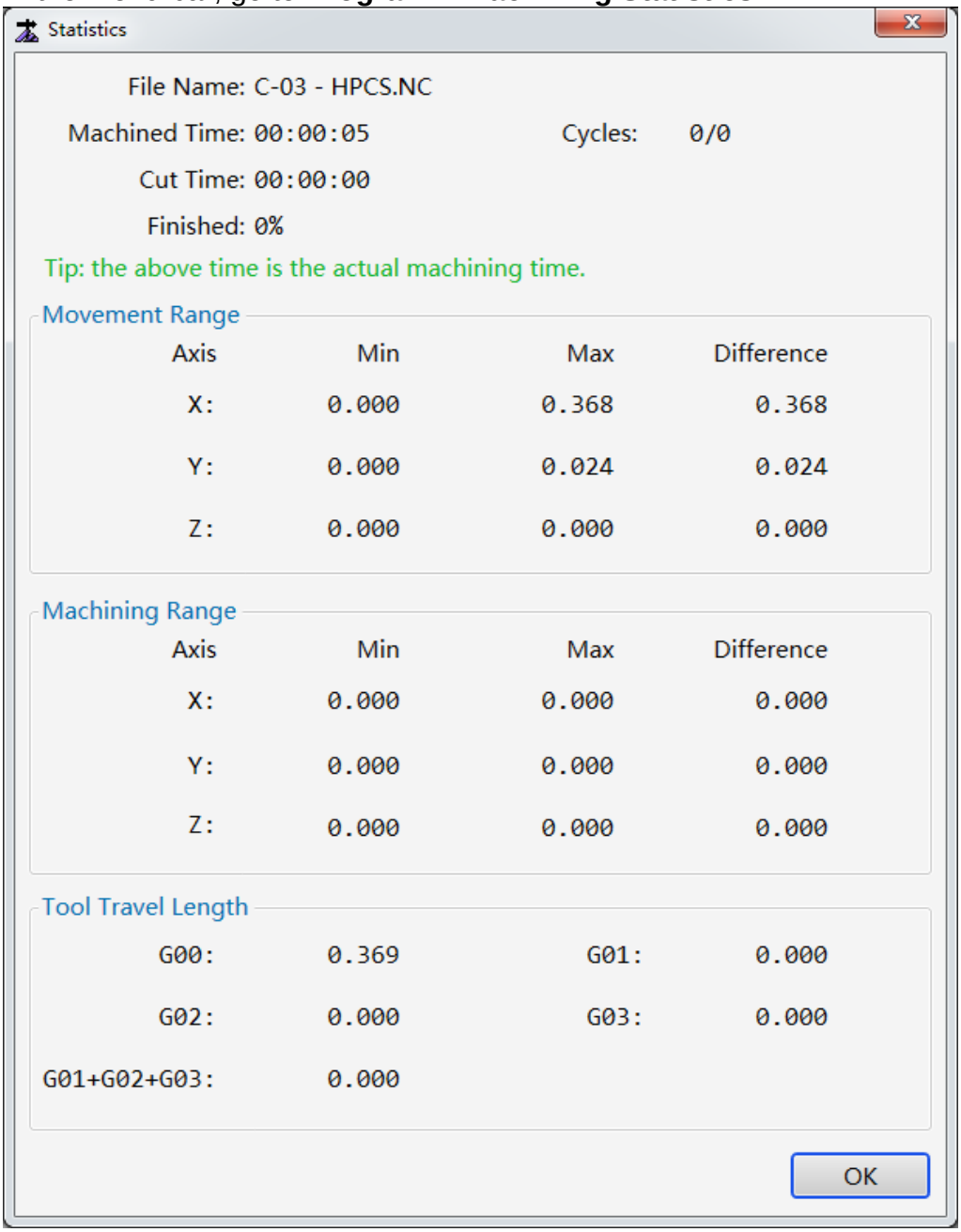
2. Enter commands in the fields. Use ";" for line switching.
3. Click **Execute** behind the command box. The system executes the commands in the box.

11.6 Machining Statistics

You can check the machining statistics automatically collected to create better machining plans and avoid machine collision.

Follow the steps below to check machining statistics:

1. In the menu bar, go to **Program** > **Machining Statistics**.



2. In the **Statistics** window, you can check the machining file name, machining duration, cutting duration, cycle number, machining progress, movement range, machining range, and tool travel length.

12 Case Example

12.1 Quick Start

Follow the steps below to start machining with the **NcStudio Phoenix Stone Machining Center** software:

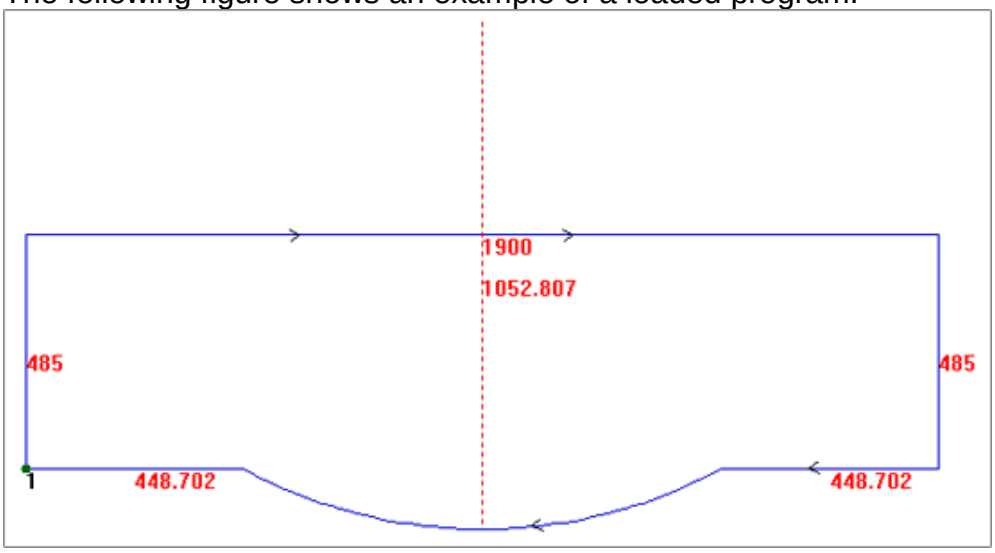
1. Load the machining program.
2. Make the axes go to the mechanical origin.
3. Run machining simulation.
4. In the **machining operation area**, click 

to start machining.

12.2 Cutting Workpiece Outline

Follow the steps below to cut the outline of a workpiece:

1. Load a program file or draw the shape in the software drawing area. For details, see Load Machining Files or Draw Shapes.
   The default layer is layer 0. The color of shapes on it is deep blue.
   The following figure shows an example of a loaded program:



2. Select the shape. Click **TechSet** in the **layers bar**. In the **Technics Parameter** window, click the **Technics type** drop-down menu and select **Outline**. Set the other parameters for the outline technic. For details, see Set Outline Parameters.
3. **Optional**: Set other technic parameters based on your needs:
   - Scalpel Line
   - Set Workpiece Origin
   - Set Slow Spot
   - Set Machining Sequence
   - Set Machining Direction
4. Go to Mechanical Origin
5. Set tool compensation and offset parameters based on your needs:
   - Set Tool Compensation Parameters
   - Set Wear Compensation Parameters
   - Set Tool Offset Parameters

6. **Optional**: Set edge seeking and centering parameters. For details, see Set Edge Finding and Centering Parameters. After setting edge seeking and centering parameters, click the corresponding button (**RectangleSeek**, **Ellipse Seek**, or **Manual Seek**) in the **port button area**.
7. Execute machining simulation to inspect the axis movement.
8. In the **machining operation area**, click 

to start machining.

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.