Phoenix Double-tool CNC System Users' Manual

Contents

1   System Introduction

1.1   Overview

Phoenix Double-tool CNC System (hereinafter referred to as **Double-tool CNC System**) consists of the following:

Hardware

- NC65C industrial personal computer
- Lambda 21B terminal board

Software

Phoenix double-tool software: see Main Software Page for the introduction of its main software page.

1.2 Hardware

1.2.1 NC65C

NC65C industrial personal computer, also called **NC65C Controller** (hereinafter referred to as **NC65C**). It can be applied to various industries when mated with Lambda terminal board.

The interface layout of **NC65C** is as follows:
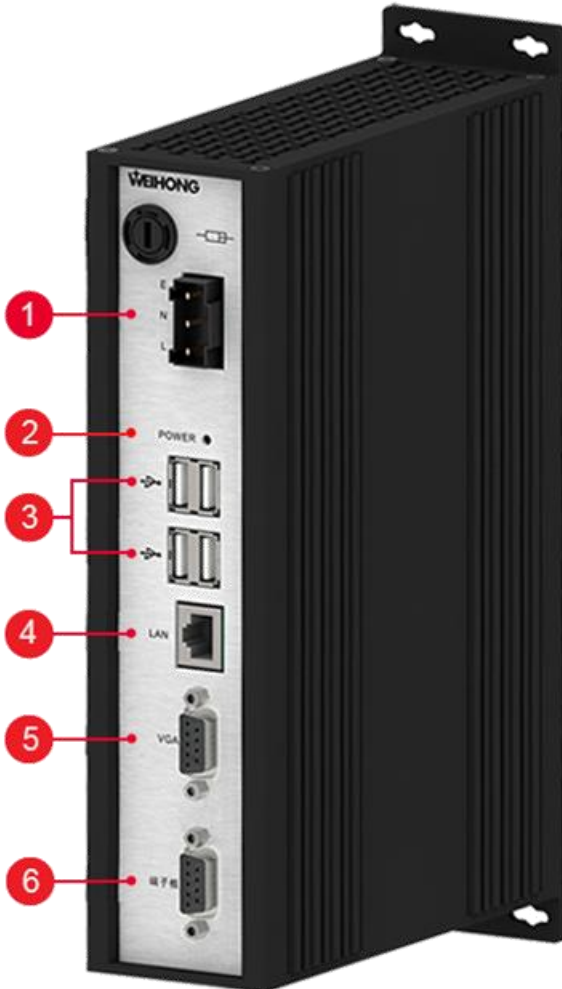
1. LNE (interface of live wire, neutral wire and earth wire): used to connect to 220V power supply.
2. Indicator light: the power light.
3. USB: used to transmit data with a USB flash disk.
4. LAN: the transmission rate is 100Mbps.
5. VGA display screen: used to connect to the display screen.
6. Lambda terminal board: used to connect to the Lambda terminal board.

1.2.2 Lambda 21B

It is used for the bus control system.

The interface layout of **Lambda 21B** is as follows:
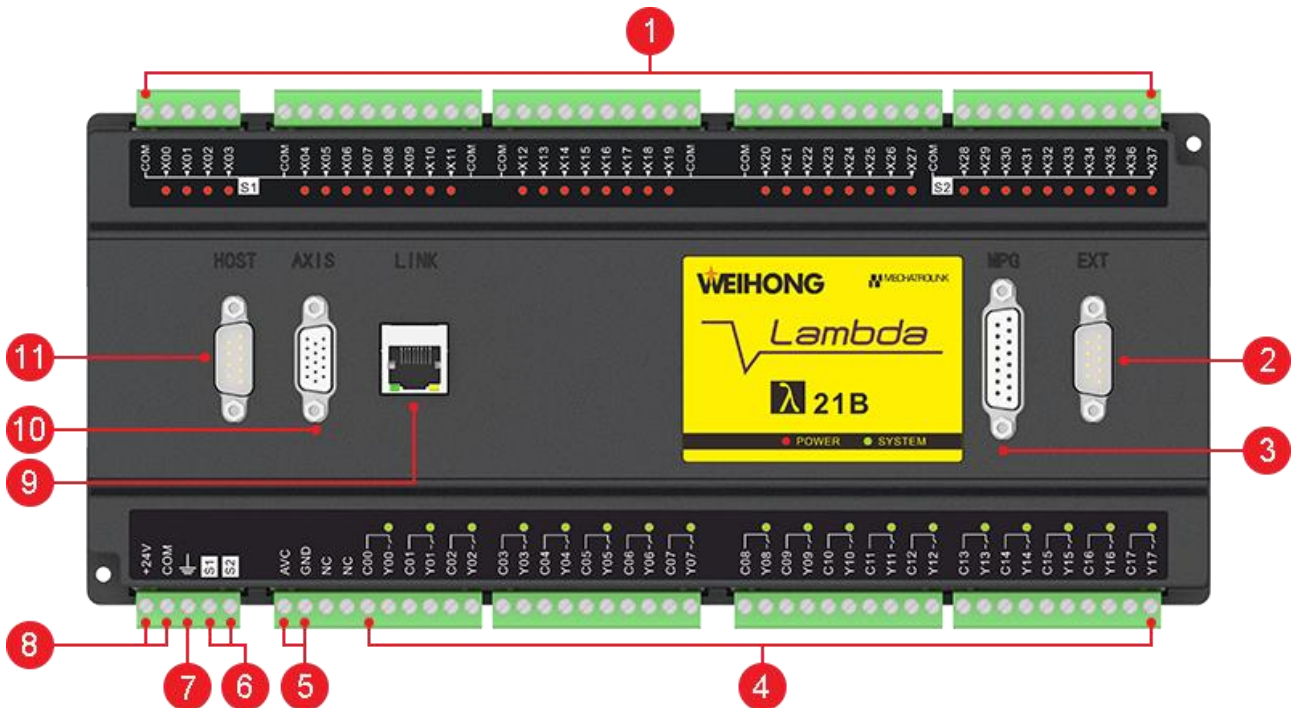
1. General input port: including X00 ~ X37.
2. Extended interface: named EXT. It is the DB9 interface.
3. Handwheel interface: named MPG.
4. General output port: including C00/Y00 ~ C17/Y17.
5. Interfaces for analog voltage outputs: AVC: the interface for analog voltage output; GND: the reference ground for analog voltage.
6. Interfaces for configuring active level: NPN / PNP configurable (S->24V / S->COM)
   S end includes:
   - S1: configure active level for X00 ~ X27.
   - S2: configure active level for X28 ~ X37.
7. Grounding port: use to be grounded to the earth.
8. Power port:
   - +24V: 24V DC positive input
   - COM: 24V DC ground
9. M-II communication port: named LINK. It is the RJ45 interface and supports 16 axes at most.
10. Pulse communication port: named AXIS. It is the DB15 interface and supports the incremental encoder.
11. Host communication port: named HOST. It is the DB9 interface and follows Phoenix bus protocol.

1.3 Main Software Page

According to the operational role and usage scenarios, the software page can be divided into the following:

- Operator page (default page)
  It includes frequently used operations during machining for operators.
  In **Technician** page, to switch to **Operator** page, in the menu bar, click **System →
  Operator Page**.
- Technician page
  It includes rich operations and machine parameters for technicians to debug the machine tool.

  In **Operator** page, to switch to **Technician** page, click [≡] → **Technician**.

1.3.1 Operator Page

Operator page is shown as follows:
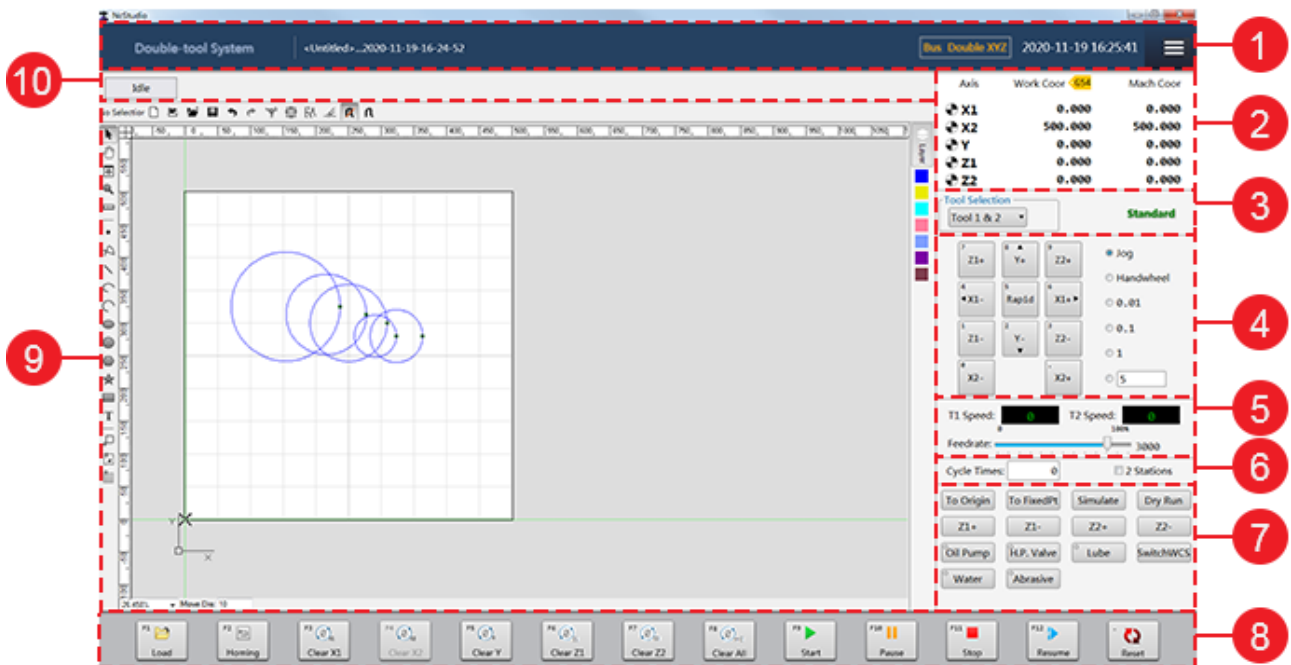
1. Title area
2. Axis coordinate display area
3. Tool selecting and machining mode display area
4. Axis direction and mode selecting area
5. Speed control area
6. Cycle machining setting and station selection area
7. Operation control area
8. Operational buttons
9. Function window
10. CNC status area

1.3.1.1 Title Area

This area consists of the system name, current file name, current configuration, current date and time, and burger button [≡].

Among them, burger button [icon] is used to do the following:

- Set parameters.
- Control valve.
- Switch to **Technician** page.

1.3.1.2 Axis Coordinate Display Area

This area shows the machine coordinates and workpiece coordinates of each axis.

After each axis returns to the machine origin, the sign [icon] will appear in front of each axis in this area.

1.3.1.3 Tool Selecting and Machining Mode Display Area

This area is used to do the following:

- Select tools for machining. See Select the Tool(s) for details.
- Show the current machining mode. See Execute Automatic Machining for how to switch the machining mode.

1.3.1.4 Axis Direction and Mode Selecting Area

This area consists of the following:

- Axis direction buttons: to move each axis towards positive/negative direction.
  Note: The axis number controlled by axis direction buttons differs in the selected tool number.

- Mode buttons: to switch to the following modes:
  - Jog
    - Press an axis direction button. The machine tool keeps running at jogging speed until you release it.
    - Press several axis direction buttons. The selected axes move at the same time at jogging speed until you release them.
    - Press **Rapid** button and an axis direction button at the same time. The machine tool moves at rapid jogging speed until you release them.
  - HW: the machine tool is controlled by the handwheel.
  - Step: click an axis direction. The machine tool moves 0.01mm, 0.1mm, 1mm or the customized step size.
    The default customized step size is 5mm and the customized step should not be too large to avoid damage due to misoperation.
    Note: Please do not click the axis direction button too frequently because the system needs a certain time for response.

1.3.1.5 Speed Control Area

This area consists of the following:

- Current actual speed: to show the current actual speed of tool 1 and tool 2.
- Feedrate override: to adjust the feedrate override.

1.3.1.6 Cycle Machining Setting and Station Selection Area

This area is used to do the following:

- Set the machining times for cycle machining.
  After the set times has reached, the machine tool automatically stops machining.
- Enable double stations for machining.
  With it unchecked, the system uses the current station for machining by default.

1.3.1.7　　Operation Control Area

This area is used to do the following:

- Execute common operations, including <u>returning to the machine origin</u>, returning to the fixed point, <u>simulate machining</u>, <u>execute dry run</u>, <u>execute Z-axis jiggle</u>, and switch the workpiece coordinate system.
- Turn on/off common ports, including the oil pump, high pressure valve, lube valve, water valve, and abrasive valve.

1.3.1.8　　Operational Buttons

The operational buttons include buttons for unloading a machining file, returning to the machine origin, clearing all axes, starting machining, pausing machining, stopping machining, resuming machining and reset machining.

1.3.1.9　　Function Window

This window is used to do the following:

- <u>Draw objects.</u>
- <u>Edit objects.</u>
- <u>Set batch modifying.</u>
- <u>Set technics.</u>
- View the machining track in real time during machining to ensure the proper implementation of the toolpath file.

1.3.1.10　　CNC Status Area

This area is used to check the following:

- Current system status, including **Idle**, **Run**, etc.
- System prompts and alarms.
  You can check system logs in **Log** dialog box by double clicking the blank area.
  See <u>Check System Logs</u> for details.

1.3.2 Technician Page

Technician page is shown as follows:
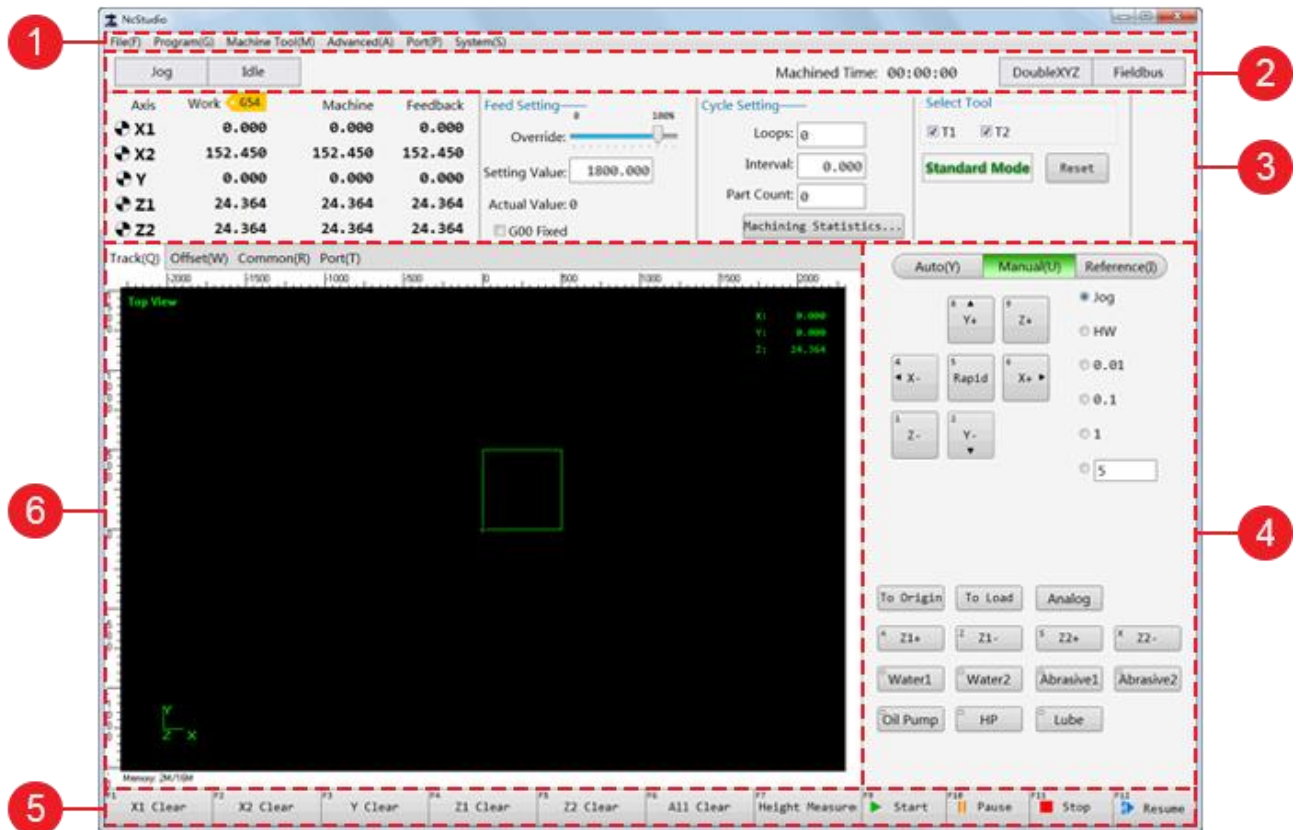
1. Menu area
2. CNC status area
3. Machine control area
4. Mode control area
5. Operational buttons
6. Function windows

1.3.2.1 Menu Area

This area includes the following menus:

- **File**: load a toolpath, generate an installation package, restart the software, restart/close the system, and show the desktop.
- **Program**: check machining statistics.
- **Machine**: execute operations about the machine tool, including start standard machining, simulate machining, return to the machine origin, execute Y1-axis and Y2-axis jiggle, etc.
- **Port**: turn on the water pump, oil pump, high pressure pump, water valve, abrasive valve, and set valve control process.
- **System**: check system logs, set system parameters, set drive parameters, clear the drive alarms, switch to **Operator** page, use remote assistance, etc.

1.3.2.2   CNC Status Area

This area is used to check the following:

- Current system mode, including **Auto**, **Jog**, etc.
- Current system status, including **Idle**, **Run**, etc.
- System prompts and alarms.
  You can check system logs in **Log** dialog box by double clicking the blank area.
  See Check System Logs for details.
- Machined time
- Current software configuration

1.3.2.3   Machine Control Area

This area consists of the following:

- Coordinate display area: to show the current active workpiece coordinate system, workpiece coordinate and machine coordinate of each axis.
- Feed setting area: to set feedrate override, and modify speed.
- Cycle setting area: to set cycle times and interval, check finished cycle times, workpiece count and machining statistics, and enable double stations for machining.
- Tool selection and machining mode display area: to select the tool(s), and display the current machining mode.

1.3.2.4   Mode Control Area

This area consists of the following modes:

- **Auto** mode:
  - Show the content of the loaded toolpath file.
  - Execute common operations, including returning to the workpiece origin, returning to the fixed point, execute dry run, reset machining, and execute Z-axis jiggle.
  - Turn on/off common ports, including oil pump, high pressure valve, lube valve, water valve, and abrasive valve.
- **Manual** mode:
  - Show axis direction buttons and mode buttons. See Axis Direction and Mode Selection Area for details.
  - Execute common operations, including returning to the workpiece origin, returning to the fixed point, execute dry run, and switch the workpiece coordinate system.
  - Turn on/off common ports, including oil pump, high pressure valve, lube valve, water valve, and abrasive valve.
- **Reference** mode:
  - Set the feedback position of the absolute encoder as datum.
  - Adjust the machine coordinates of the software after getting the feedback position of the absolute encoder.

1.3.2.5   Function Windows

Function windows include the following:

- **Track** window: to show the machining track in real time during machining, or dry run.
- **Common Parameters** window: to check, modify and customize common parameters.
- **Port** window: to check and modify the port polarity, conduct a simulation test, and set filter.

2   Commissioning

This section mainly introduces operations to quickly do commissioning for **Double-tool CNC System**.

The process for commissioning is as follows:

```
        Start
          │
          ▼
   Clear I/O port            Adjust the axis
      alarms                    direction
          │                        │
          ▼                        ▼
    Set drive            Confirm machine
    parameters           coordinates of
          │              double tools
          ▼                        │
  Set commissioning                ▼
    parameters           Execute dry run
          │                        │
          └──────────►             ▼
                               End
```
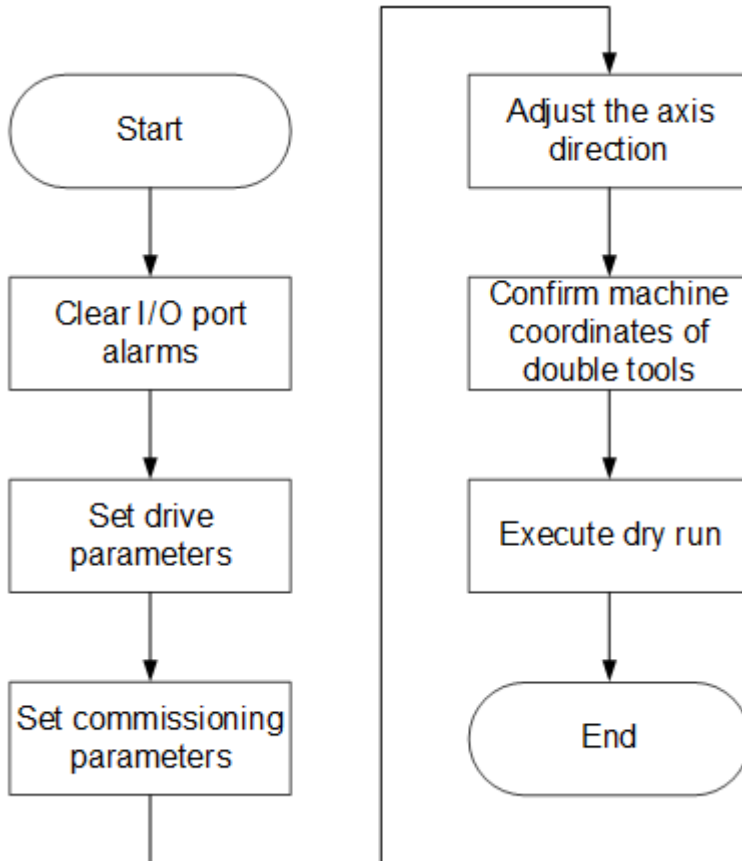
If there is no special instruction, please execute the above operations in **Technician** page (Manufacturer permission is required).

2.1   Clear I/O Port Alarms

It is used to clear I/O port alarms, including E-stop alarm, limit alarms and drive alarms, so as to establish good communication between the system and the servo drive.

2.1.1   Clear the E-stop Alarm

To clear the E-stop alarm, check **E-stop** button:

- If it is pressed, release the button.
- If it is not pressed, modify the polarity of port **E-stop**.
  See Execute I/O Port Related Operations for details.

2.1.2   Clear the Limit Alarms

When the machine tool triggers the limit switch, the system will send a positive/negative limit alarm.

To clear the limit alarms, move the machine tool towards the opposite direction of the limit switch.

2.1.3 Clear the Drive Alarms

To clear the drive alarms, do the following:

1. Check if the wiring of the drive is connected correctly and securely:
   o If it is, proceed to the next step.
   o If it is not, correct and tighten the wiring.
2. Check if the setting value of parameter **Drive Station Address** is the same as the drive station address of the machine tool:
   o If it is, proceed to the next step.
   o If it is not, modify the setting value of the parameter according to the drive station address of the machine tool.
3. **Optional:** If the system is under the bus configuration, to clear the common drive alarms and improve commissioning efficiency, in the menu area, click **Machine → Clear Drive Alarm**.

2.2   Set Drive Parameters

This operation is used to set drive parameters via the software after the system is well connected to the servo drive.

Before setting drive parameters, make sure the system has been connected to the hardware correctly.

To set drive parameters, do the following:

1. In the menu area, click **System → Drive Parameters**. **Drive Parameters** dialog box pops up:
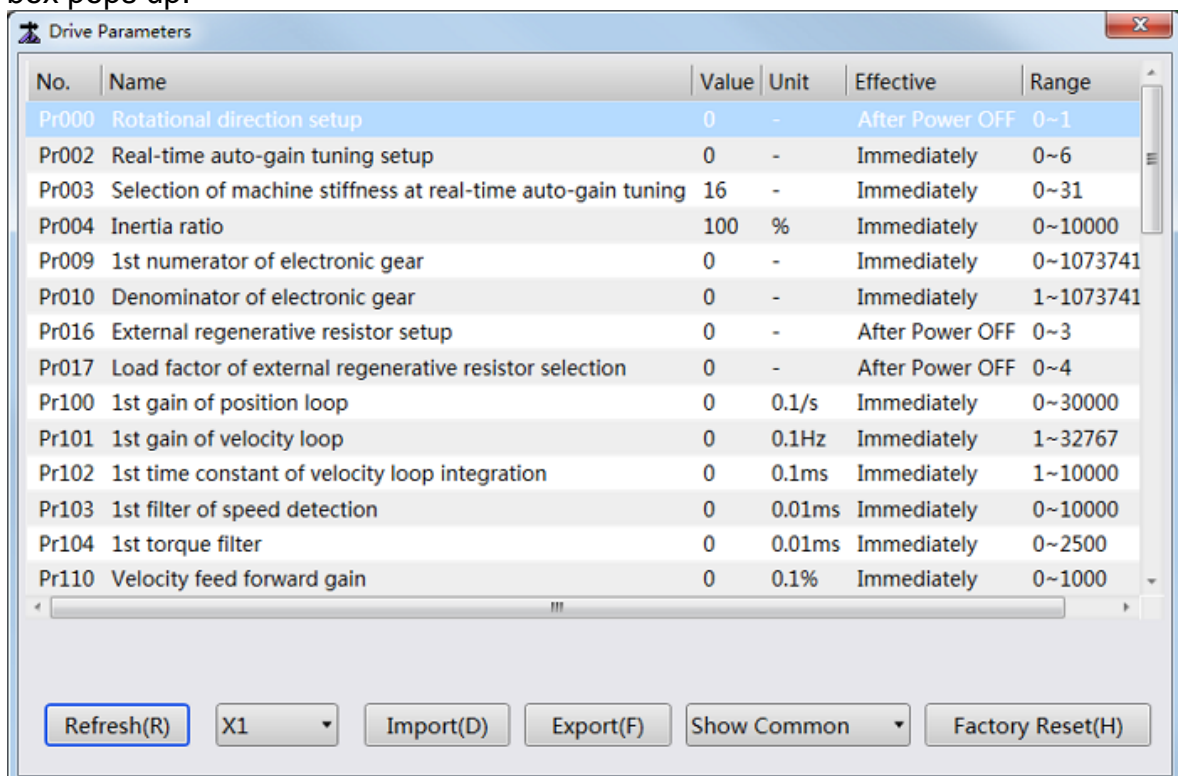
2. Select the target axis.
3. Double click the target parameter, and enter a value.

In **Drive Parameters** dialog box, you can also do the following:

- To refresh values of drive parameters, click **Refresh**.
- To import drive parameters that have been set before, click **Import** and select a file.
- To export drive parameters, click **Export** and select a path.
- To show common/all drive parameters, select **Show Common** / **Show All**.
- To initialize drive parameters, click **Factory Reset**.

2.3 Set Commissioning Parameters

This operation is used to set parameters for commissioning in the bus control system.

To set commissioning parameters, do the following:

1. In the menu area, click **System** → **Parameters**. **Parameter** dialog box pops up:
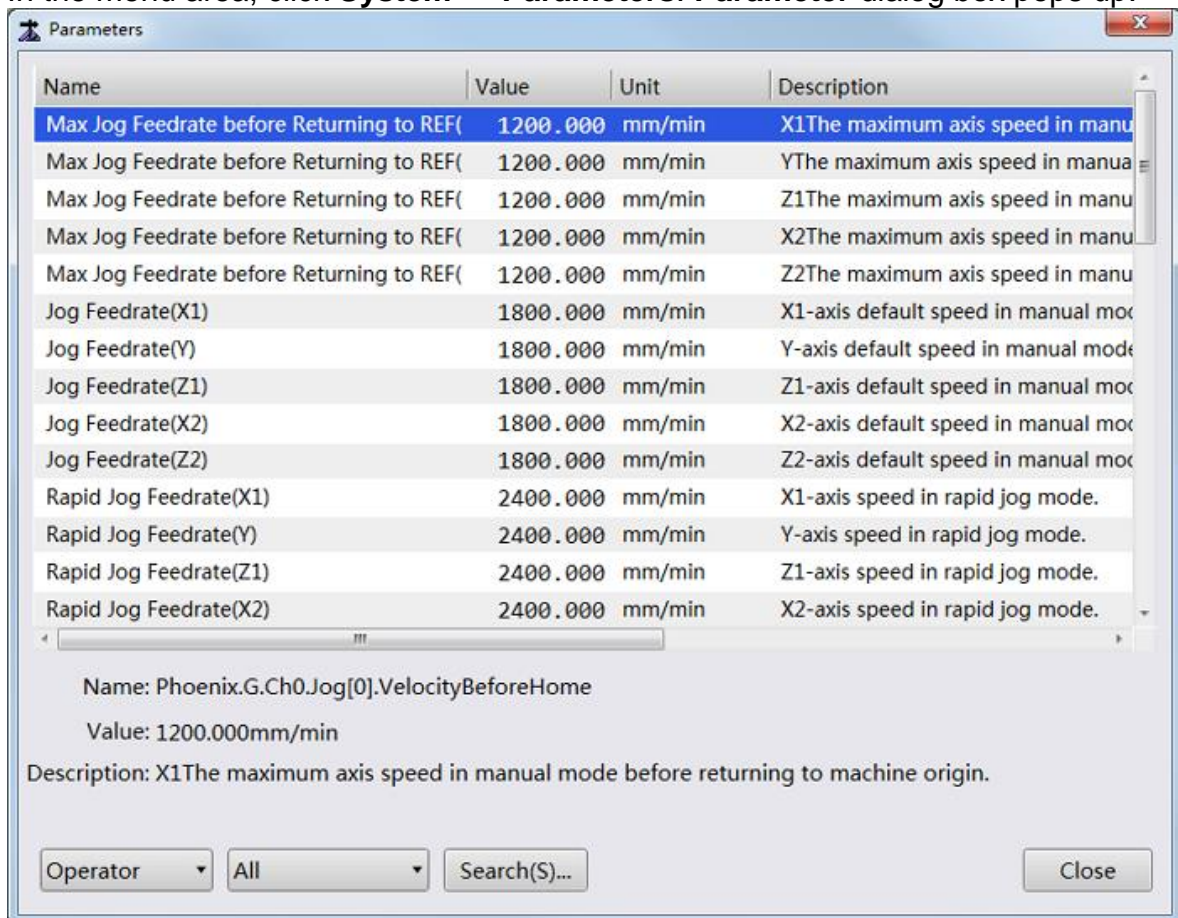
2. Select **Manufacturer** permission in the lower left, search and set the following parameters:
   - **Screw Pitch**: the axial distance between the corresponding points of two adjacent teeth on the threads.
     Its setting should match the actual condition.
   - **Mechanical Reducer Ratio**: the ratio of reducer input speed to output speed.
     It consists of parameter **Numerator of Mechanical Reducer Ratio** and parameter **Denominator of Mechanical Reducer Ratio**. And its setting should match the actual situation:
     Mechanical Reducer Ratio
     = Reducer Input Speed / Reducer Output Speed
     = Teeth No. of Drive Wheel / Teeth No. of Driving Wheel
     = Motor Rotational Speed / Screw Rotational Speed

- **Encoder Digit**: its setting should match the actual situation.
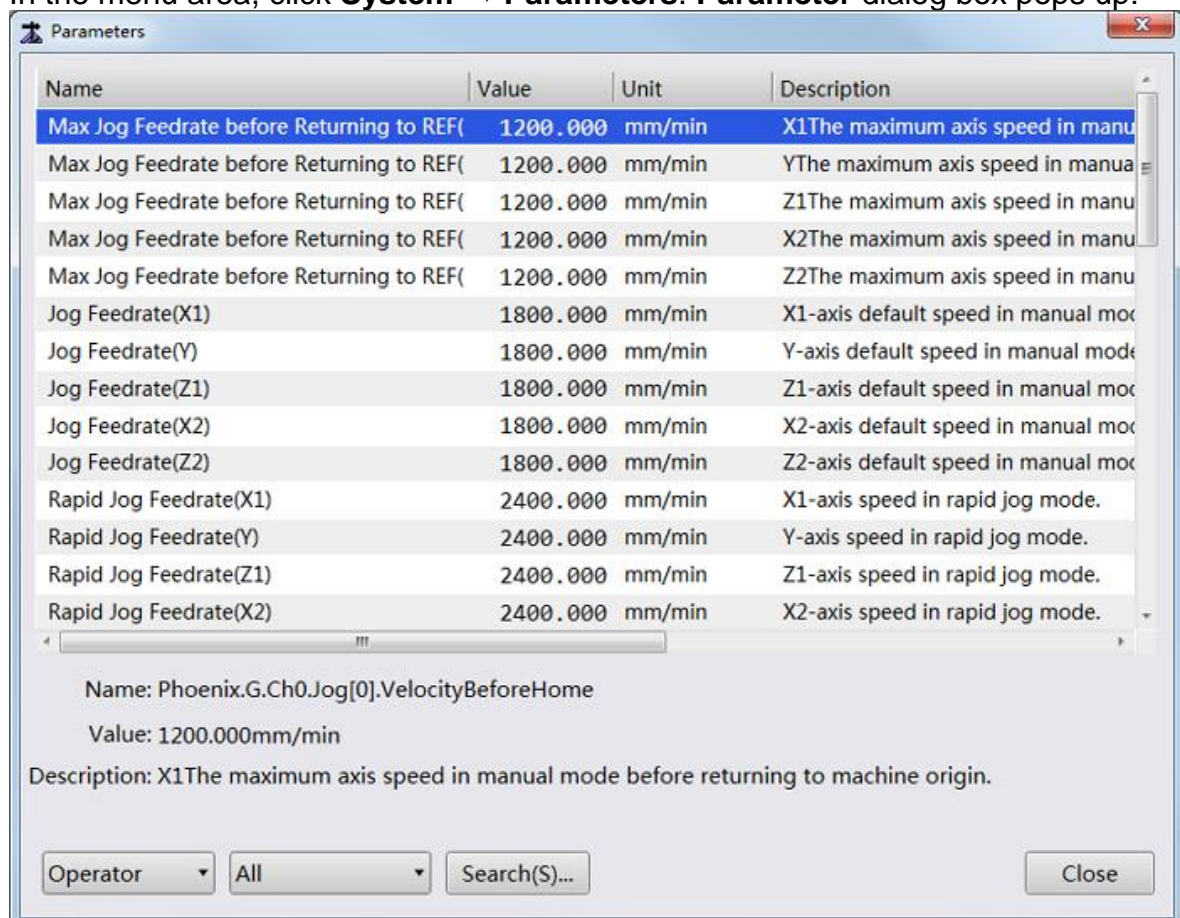- **Electronic Gear Ratio**: the ratio that the servo enlarges or shrinks the received pulse frequency. If it is greater than 1, the servo enlarges the received pulse frequency; if it is less than 1, the servo shrinks the received pulse frequency. It is set to **1:1** by default.
  It consists of parameter **Numerator of Electronic Gear Ratio** and parameter **Denominator of Electronic Gear Ratio**. And its setting should match the actual situation.

2.4 Adjust the Axis Direction

This operation is used to check if the positive direction of each axis is the same with the direction stipulated by **Right Hand Rule**, so as to avoid damage to the machine tool due to incorrect direction.

Taking X-axis as an example, to adjust the axis direction, do the following:

1. In the menu area, click **System → Parameters**. **Parameter** dialog box pops up:

Name	Value	Unit	Description
Max Jog Feedrate before Returning to REF(	1200.000	mm/min	X1The maximum axis speed in manu
Max Jog Feedrate before Returning to REF(	1200.000	mm/min	YThe maximum axis speed in manua
Max Jog Feedrate before Returning to REF(	1200.000	mm/min	Z1The maximum axis speed in manu
Max Jog Feedrate before Returning to REF(	1200.000	mm/min	X2The maximum axis speed in manu
Max Jog Feedrate before Returning to REF(	1200.000	mm/min	Z2The maximum axis speed in manu
Jog Feedrate(X1)	1800.000	mm/min	X1-axis default speed in manual mo
Jog Feedrate(Y)	1800.000	mm/min	Y-axis default speed in manual mode
Jog Feedrate(Z1)	1800.000	mm/min	Z1-axis default speed in manual mo
Jog Feedrate(X2)	1800.000	mm/min	X2-axis default speed in manual mo
Jog Feedrate(Z2)	1800.000	mm/min	Z2-axis default speed in manual mo
Rapid Jog Feedrate(X1)	2400.000	mm/min	X1-axis speed in rapid jog mode.
Rapid Jog Feedrate(Y)	2400.000	mm/min	Y-axis speed in rapid jog mode.
Rapid Jog Feedrate(Z1)	2400.000	mm/min	Z1-axis speed in rapid jog mode.
Rapid Jog Feedrate(X2)	2400.000	mm/min	X2-axis speed in rapid jog mode.

Name: Phoenix.G.Ch0.Jog[0].VelocityBeforeHome

Value: 1200.000mm/min

Description: X1The maximum axis speed in manual mode before returning to machine origin.

Operator | All | Search(S)... | Close

2. Select **Manufacturer** permission in the lower left, search parameter **Axis Direction (X)**, and check its setting value.
3. Judge the positive direction of X-axis according to the **Right Hand Rule**.
4. In **Manual** mode, click **X+** or **X-** button to move X-axis and observe its moving direction.
5. **Optional:** If the actual moving direction is opposite to the judged direction, change the setting value of parameter **Axis Direction (X)** to the opposite value.

2.5   Confirm Machine Coordinates of Double Tools

This operation is used to execute returning to the machine origin, confirm the distance between double tools, and set datum when you use the system at the first time, so as to adjust coordinates before machining.
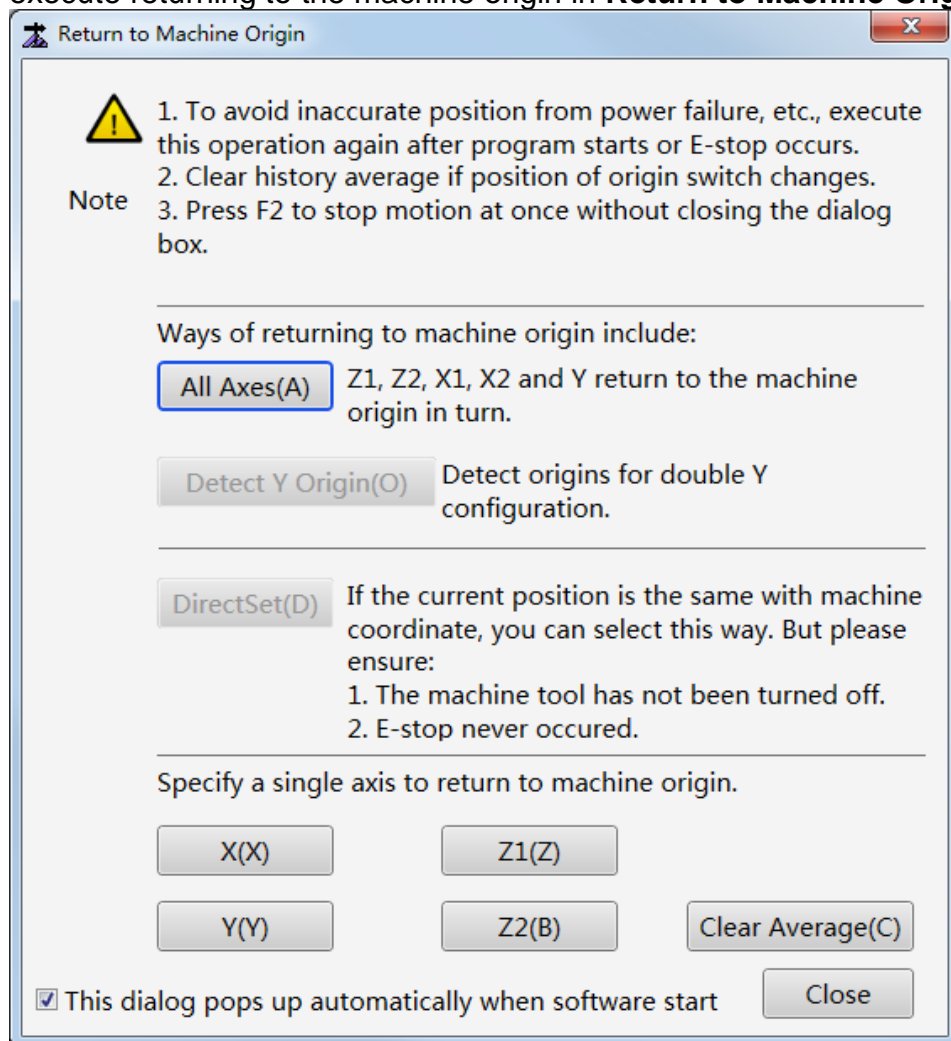
There is no need to set a returning order for all axes during setting datum, and set datum again after recovering from alarms, like power interruption and E-stop, because the system will automatically read datum information.

The double tools share the same machine coordinate system.

To confirm the machine coordinates of double tools, do the following:

1. Switch to **REF** mode in the mode control area.
2. To execute returning to the machine origin, click one of the following operational buttons:
   - Click **All Axes**. The system returns to the machine origin in the order of Z1-axis, Z2-axis, X-axis, and Y-axis.
   - Click **X Back Home** / **Y Back Home** / **Z1 Back Home** / **Z2 Back Home**. The specified axis returns to the machine origin.
     For safety, it is suggested to firstly return Z1-axis and Z2-axis.
   Alternatively, in the menu area, click **Machine → Return to Machine Origin** and execute returning to the machine origin in **Return to Machine Origin** dialog box:

Return to Machine Origin

⚠ **Note**
1. To avoid inaccurate position from power failure, etc., execute this operation again after program starts or E-stop occurs.
2. Clear history average if position of origin switch changes.
3. Press F2 to stop motion at once without closing the dialog box.

Ways of returning to machine origin include:

| All Axes(A) | Z1, Z2, X1, X2 and Y return to the machine origin in turn. |

| Detect Y Origin(O) | Detect origins for double Y configuration. |

| DirectSet(D) | If the current position is the same with machine coordinate, you can select this way. But please ensure:
1. The machine tool has not been turned off.
2. E-stop never occured. |

Specify a single axis to return to machine origin.

| X(X) | Z1(Z) |
| Y(Y) | Z2(B) | Clear Average(C) |

☑ This dialog pops up automatically when software start   Close

If **This dialog pops up automatically when software starts** is checked, the dialog box will pop up automatically when software starts.

3. Measure the distance between tools of X1-axis and X2-axis, and set the measured value as the value of parameter **Home Position(X2)**.
   Note: Parameter **Home Position(X1)** and parameter **Home Position(X2)** should be modified synchronously to confirm that the difference between machine origin of X1-axis and X2-axis keeps the measured value.

4. To set datum for the axis that has returned to the machine origin, do one of the following:
   o In the menu area, click **Machine → Datum Setting**, and select the target axis.
   o In the mode control area, click **Set** button after the target axis.

   After setting datum, the sign ⏣ appears before the related axis.

2.6   Execute Dry Run

This operation is used to know the running area and the speed steadiness of the machine tool before machining.

During executing dry run, the machine tool moves like normal machining without turning on/off valves.

To execute dry run, in the mode control area, switch to **Auto** / **Manual** mode, and click **Dry Run**.

3 Quick Start

This section mainly introduces the operations to quickly use **Double-tool CNC System** for machining.
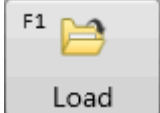
The process of quick start is as follows:
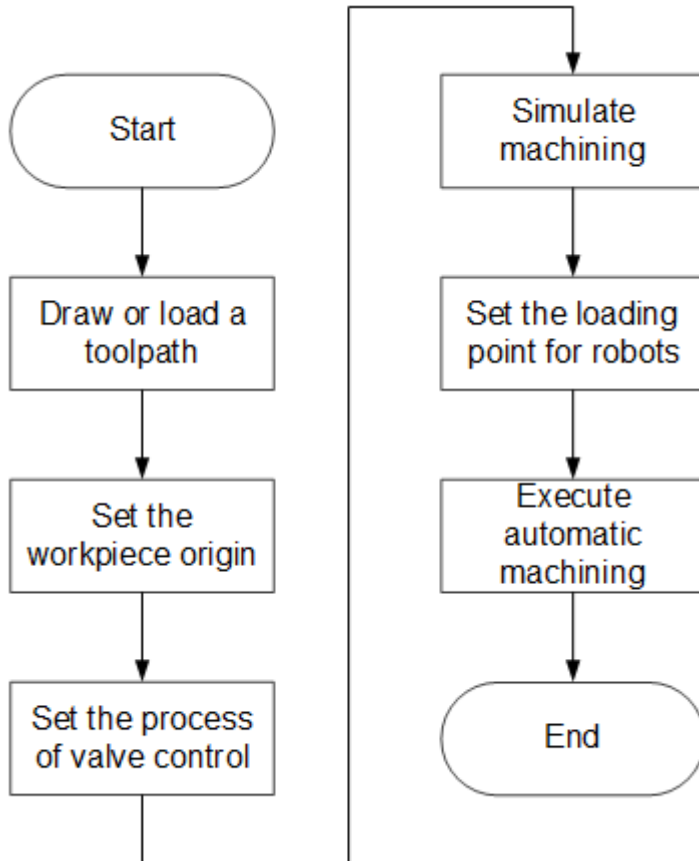
If there is no special instruction, please execute the above operations in **Operator** page.

3.1 Load or Draw a Toolpath

Before machining, you need to load a prepared toolpath or draw a toolpath in the function window.

To load or draw a toolpath, do one of the following:

- To load a toolpath, do one of the following:
  - Drag the target toolpath (supported formats: NCEX / DXF / DWG / PLT / G / NC) into the function window.
  - To load a NCEX file only, do one of the following and select the target tool path file:

    - In the upper part of the function window, click .

    - Click the operational button .
  - To import a DXF / DWG / PLT / G / NC file, in the upper part of the function window, click .

- To draw a toolpath, do one of the following:
  - Manually draw a toolpath in the function window.
  - Call objects in the gallery.

The loaded tool path shows in the function window.

3.2 Set the Workpiece Origin

This operation is used to set the origin of the workpiece coordinate system (WCS) by clearing workpiece coordinates of each axis, so as to define the coordinate origin in the toolpath file before machining.

To set the workpiece origin, do the following:

1. **Optional:** If the target WCS is not the default one, to switch to the target WCS, click **Switch WCS** in the operation control area.
   The system supports WCS **G54** and **G55**. The default WCS is **G54**.
2. Select one of the following:
   - Set X-axis workpiece origin.
   - Set Y-axis workpiece origin.
   - Set Z-axis workpiece origin.

3.2.1 Set X-axis Workpiece Origin

This operation is used to set X-axis workpiece origin by clearing. And it differs in clearing modes.
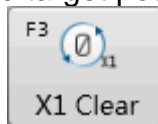
The clearing mode is defined by parameter **Clearing Mode of Double Tools**:

- **0**: clear X-axis by space.
  This mode only supports clearing X1-axis. The system will automatically calculate X2-axis origin position according to the distance between workpiece origins after clearing X1-axis.
- **1**: clear X-axis independently, that is, separately clear X1-axis and X2-axis.
  In this mode, the distance between double tools during machining will keep the value at clearing.

According to the clearing mode, set X-axis workpiece origin:

- If parameter **Clearing Mode of Double Tools** is set to **0**, do the following:
  a. To ensure the distance between workpieces, set parameter **Distance between Y1Y2 Workpiece Origins(G54)** / **Distance between Y1Y2 Workpiece Origins(G55)** according to the actual situation.
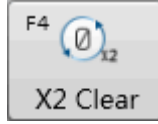 b. To move X1-axis to the target position, click **X1+** / **X1-** in **Manual** mode.
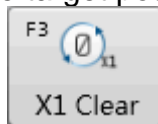
  c. To clear X1-axis, click [X1 Clear]. The system clears workpiece coordinates of X1-axis and automatically calculates X2-axis origin position according to the set distance.

  Note: The distance between double tools during machining keeps the setting value of parameter **Distance between Y1Y2 Workpiece Origins(G54)** / **Distance between Y1Y2 Workpiece Origins(G55)**.

- If parameter **Clearing Mode of Double Tools** is set to **1**, do the following:
    a. To move X1-axis to the target position, click **X1+** / **X1-** in **Manual** mode.
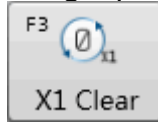
    b. To clear X1-axis, click [X1 Clear] .
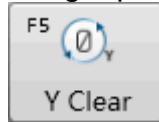   c. To move X2-axis to the target position, click **X2+** / **X2-** in **Manual** mode.

    d. To clear X2-axis, click [X2 Clear] .

  Note: During machining, the distance between double tools keeps the value at clearing.

3.2.2 Set Y-axis Workpiece Origin

To set Y-axis workpiece origin, do the following:

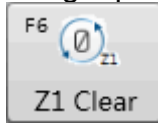1. To move Y-axis to the target position, click **Y+** / **Y-** in **Manual** mode.
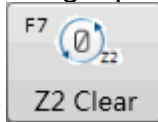
2. To clear Y-axis, click [Y Clear] .

3.2.3 Set Z-axis Workpiece Origin

To set Z-axis workpiece origin, do the following:

1. To move Z1-axis to the target position, click **Z1+** / **Z1-** in **Manual** mode.

2. To clear Z1-axis, click [Z1 Clear] .
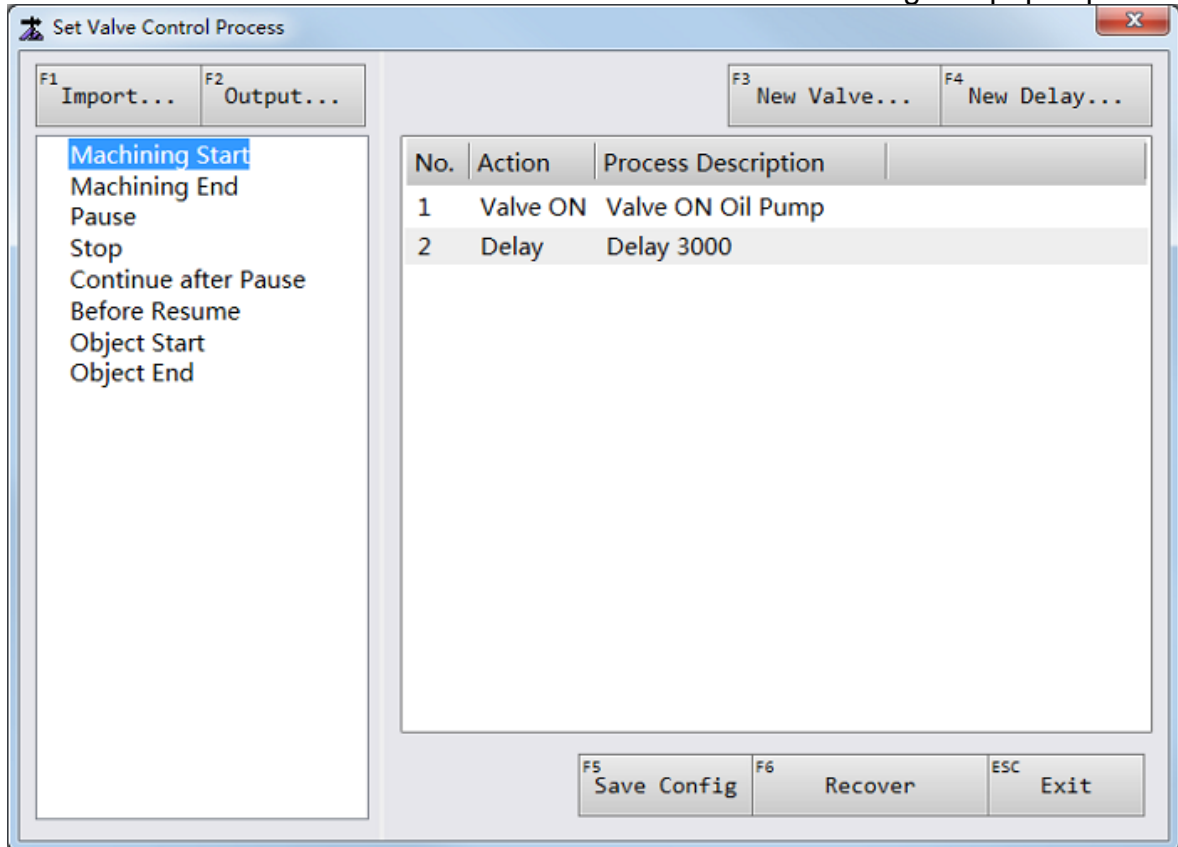3. To move Z2-axis to the target position, click **Z2+** / **Z2-** in **Manual** mode.

4. To clear Z2-axis, click [Z2 Clear] .

3.3   Set the Process of Valve Control

This operation is used to define the turning on/off sequence and delay time for valves in the current machining mode.

To set the process of valve control, do the following:

1. Click ▤ → **Valve Control**. **Set Valve Control Process** dialog box pops up:

	Set Valve Control Process	
F1 Import... F2 Output...		F3 New Valve... F4 New Delay...

	No.	Action	Process Description	
Machining Start	1	Valve ON	Valve ON Oil Pump	
Machining End	2	Delay	Delay 3000	
Pause				
Stop				
Continue after Pause				
Before Resume				
Object Start				
Object End				

F5 Save Config   F6   Recover   ESC Exit

2. Select the target process on the left, and double click the target action on the right. **Edit Valve Process** / **Edit Delay Process** dialog box pops up:

Edit Valve Process

Actions: ○ Valve ON   ● Valve OFF

Valve Type: [High Pressure Valve ▾]

[OK]   [Cancel]
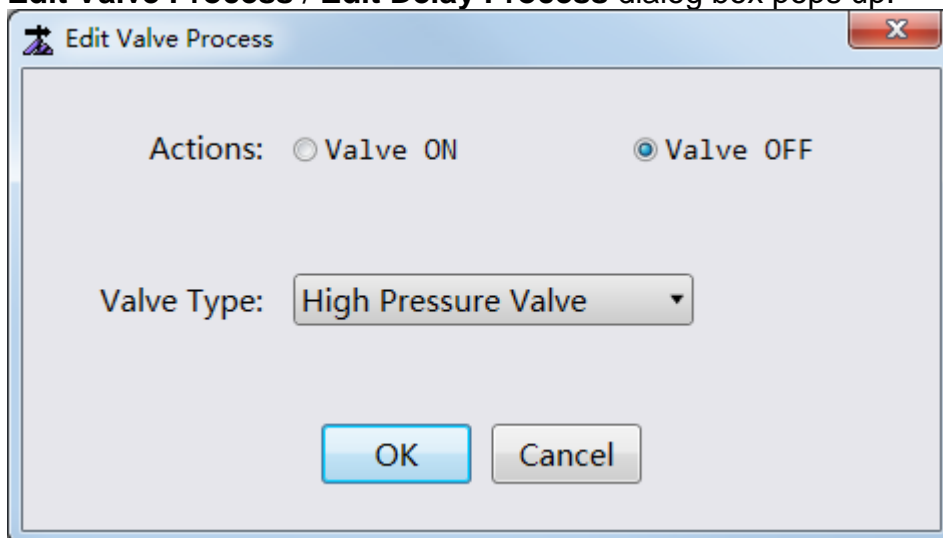
3. Modify the action type and valve type/ delay time.
4. **Optional:** To undo modifications and restore the default, click **Recover**.
5. To save modifications, click **Save Config**.

In **Set Valve Control Process** dialog box, you can also do the following:

- To create a new valve/delay process, click **New Valve** / **New Delay** in the upper right.
- To import or export a valve/delay process, click **Import** / **Output** in the upper left.

3.4   Simulate Machining

This operation is used to see the movement of the machine tool in advance, so as to avoid damage to the machine tool due to programming mistakes in the toolpath.

It provides a fast but lifelike simulation machining environment. During simulation, no actual machining occurs, and only the moving track of the tool at a high speed shows in the function window.

To simulate machining, do the following:

1. To enter into the simulation mode, click **Simulate** in the operation control area.

2. In **Auto** mode, click [F9 ▶ Start]. The system starts to simulate machining.

The simulation track shows in the function window.

3.5   Set the Loading Point for Robots

Double-tool CNC System uses robots to load/unload material. This operation is used to set the correct loading point, so as to confirm the safety during loading/unloading material with robots.

The loading point is the same as the fixed point in the system.

To set the loading point for robots, click [≡] → **Parameters** in the upper right, find and set parameter **Fixed Point Position**.

After setting the loading point for robots, click **To FixedPt** in the operation control area. The tools move to the loading point, and the robot starts to load material.

3.6   Execute Automatic Machining

This operation is used to start machining.

To execute automatic machining, use one of the following modes:

- Standard machining mode: to automatically execute machining from the beginning to the end.
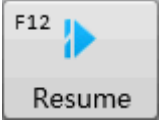- Remote machining mode: to automatically execute machining with the assembly line of double stations.

3.6.1 Use the Standard Machining Mode

To execute machining in the standard machining mode, click [F9 ▶ Start] in **Auto** mode. The system automatically starts machining.

During machining, you can do the following to control machining:

- To stop machining, click [F11 Stop].

- To pause machining, click [F10 Pause].

- To resume machining from the exact interrupted position when breakpoint resume or E-stop occurs, click [F12 Resume].

3.6.2 Use the Remote Machining Mode

To use the remote machining mode, do the following:

1. To enable the remote machining mode, click [≡] → **Parameters** in the upper right, find parameter **Enable Assembly Line for Machining** and set it to **Yes**.
2. **Optional:** In cycle machining setting and station selection area, set the cycle times and check **2 Stations** to enable double stations.
   With **2 Stations** unchecked, if left and right stations are both available, the system will use left station for machining by default.

3. In **Auto** mode, click [F9 Start]. The system automatically executes machining as follows (taking firstly starting machining on the left station as an example):

L: left station; R: right station.

4. **Optional:** To manually unload material, click **To Load** in the operation control mode after unloading material. The system automatically informs the robots to load material again and repeats the above process.

If the machining process ends abnormally and you need to continue machining, click

[F12 Resume] to continue machining.

Note: The system does not exit the remote machining mode during executing breakpoint resume.

To exit the remote machining mode, in **Idle** status, click [Reset].

4   Object Operations

This section introduces the drawing tools that are used to draw objects in the function window for machining and operations for better effects of object editing.

If there is no special instruction, please execute object operations in **Operator** page.

4.1   Object Drawing

The drawing tools include the following:

- [icon] : dot

- [icon] : polyline

- [icon] : line

- [icon] : elliptic arc

- [icon] : arc

- [icon] : ellipse

- [icon] : circle

- [icon] : polygon

- [icon] : star

- [icon] : rectangle

- [icon] : text

- [icon] : gallery

After drawing, do the following:

1. To exit drawing, right click or press **Esc**.
2. To adjust the object, select the target object and do one of the following:
   - To modify the size and position of the object, directly input the values.
   - To modify the size of the object, drag the rectangular points around the object.

4.1.1 Draw a Polyline

A polyline is a chain of lines and arcs that are joined together into one object. It includes line and tangent arc. The tangent arc corresponds to the arc.

To draw a polyline, do the following:

1. Click [icon].
2. Left click to specify two points to draw a line.

3. **Optional:** To switch to the arc mode, right click on the function window and select **Tangent Arc**.
   The arc is tangent to the last line/arc.
   If you need to switch back to the line mode, right click on the function window and select **Line**.
4. Left click to specify another point.
5. Right click and do one of the following:
   o To draw the end point of the polyline, select **Confirm**. The polyline is an unclosed object.
   o To connect the current point to the start point, click **Close**. The polyline is a closed object.
   o To cancel the previous operations and exit drawing a polyline, click **Cancel**.

After drawing a polyline, to change an unclosed polyline into a closed one, select it and check **Close** in the top of the function window. (**Note:** This operation is irreversible.)

4.1.2 Draw an Elliptic Arc

To draw an elliptic arc, do the following:

1. Click .
2. Draw an ellipse.
   See Draw an Ellipse for details.
3. Left click to separately specify the start point and the end point.

4.1.3 Draw an Arc

To draw an arc, do the following:

1. Click .
2. Left click to specify the center point.
3. Left click to specify the start point.
   The distance between the start point and the center point is the radius.
4. Left click to specify the end point.

4.1.4 Draw an Ellipse

To draw an ellipse, do the following:

1. Click .
2. Left click to specify the center point.
3. Left click to separately specify two points.
   The distances between the center point and the two points are separately the major axis and minor axis of the ellipse.

4.1.5 Draw a Circle

To draw a circle, do the following:

1. Click .
2. Left click to specify the center point.
3. Left click to specify a point.
   The distance between this point and the center point is the radius.

4.1.6 Draw a Polygon

To draw a polygon, do the following:

1. Click .
2. Set parameter **Edges** in the top and press **Enter** on the keyboard.
3. Left click twice to separately specify the center point and a vertex.

4.1.7 Draw a Star

To draw a star, do the following:

1. Click .
2. Set parameter **Vertexes** in the top of the function window and press **Enter** on the keyboard.
3. Left click twice to separately specify the center point and a vertex.

4.1.8 Enter Text

It is mainly used to draw text in an advertisement board.

To enter text, do the following:

1. Click .
2. To specify a text box, click and drag the mouse.
3. Enter text in the input box.
    During this process, you can press **Ctrl + Enter** for line feed.
4. Press **Enter**.

4.1.9 Call Objects in the Gallery

It is used to select the basic objects from the gallery.

To call objects in the gallery, do the following:

1. Click [icon]. **Gallery** dialog box pops up:
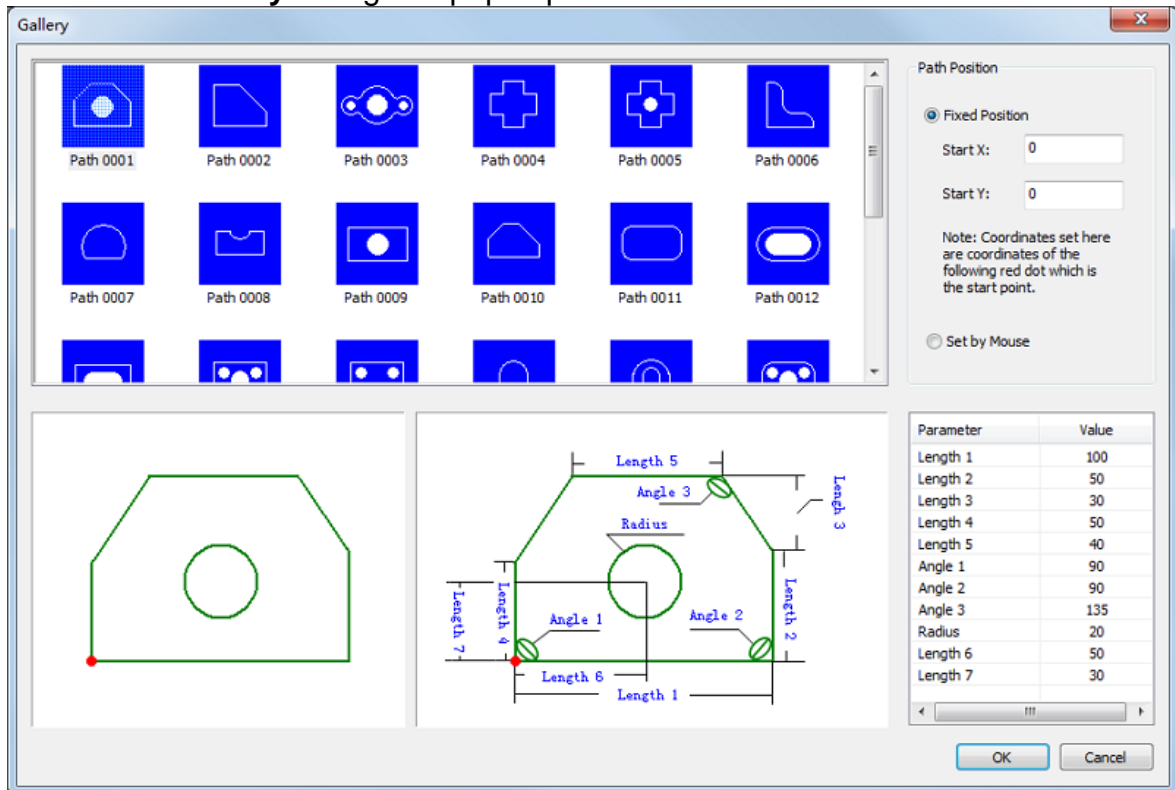
2. Select the target object.
3. Set the size and position of the object.

4.2 Auxiliary Editing

You can do the following for better effects of object editing:

- Select objects
- Pan the view
- Fit to window
- Zoom in the selected area
- Measure distance
- View objects
- Set catch options

4.2.1 Select Objects

It is used to select objects for editing.

To select objects, do one of the following:

1. In the left of the function window, to call **Select Object** command, click   .
2. Do one of the following:
    o   To select a single object, left click the target object on the function window.
    o   To select several objects, press **Ctrl** and left click the target objects one by one.
    o   To select objects contained in the selected area, hold the left mouse button to drag from the upper left to the lower right.
        Objects intersected with the area will not be selected.
    o   To select objects intersected with or contained in the selected area, hold the left mouse button to drag from the lower right to the upper left.

4.2.2 Pan the View

It is used to reposition the function window by selecting a base point and a second point for the new position, and used for a better view of objects without changing the actual sizes and coordinates.

To pan the view, do one of the following:

*   Press the mouse wheel and drag the mouse at the same time to move the view to the target position.

*   Click   to call **Pan** command, and hold the left mouse button to move the view to the target position.

To stop panning the view, right click or press **Esc**.

4.2.3 Fit to Window

It is used to resize objects and make them fit to the function window on the premise of showing all objects, and used for a better view of objects without changing the actual sizes and coordinates.

To fit to window, do one of the following:

*   In the left of the function window, click   .
*   On the numeric keypad, press *.

4.2.4 Zoom in the Selected Area

It is used to zoom in the selected part of the object to the size of the function window, and used for a better view of objects without changing the actual sizes and coordinates.

To zoom in the selected area, do the following:

1. To call **Zoom by Rect** command, click   .
2. Left click on the function window, drag the mouse to form a rectangle and left click again. The area formed by the rectangle zooms in.

4.2.5 Measure Distance

It is used to measure distance, X-axis / Y-axis offset and the vector angle between two points.

To measure distance, do the following:

1. To call **Measure** command, in the left of the function window, click ⬜.
2. Left click to select the start point for measuring, and move the cursor to select the end point.

The measurement result shows near the end point.

4.2.6 Set Catch Options

It is used to precisely locate feature points and connect objects during drawing an object.

Once the cursor moves close to the feature points, the system can easily locate to the points.

To set catch options, do the following:

1. To open **Catch Options** dialog box, in the top of the function window, click ⬜:
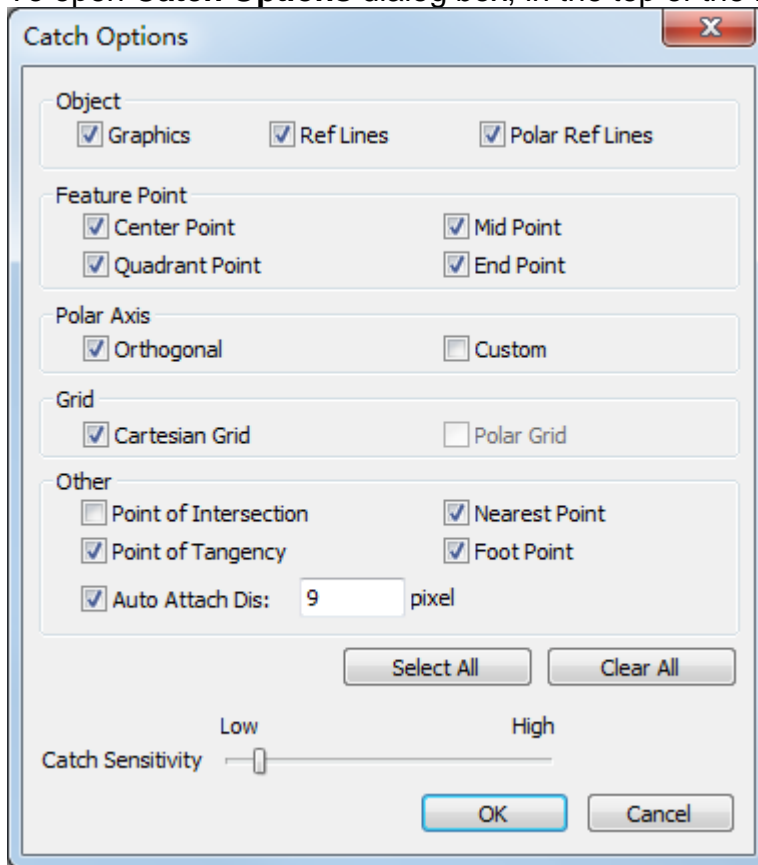
2. Check the feature items according to the shape of the object.
3. Adjust the catch sensitivity.
   It is easier to get the feature points with higher catch sensitivity.

4. To enable catching function, in the top of the function window, click ⬜.

4.3 Batch Modifying

It is used to modify the target objects and objects with the same shape and size at the same time.

To modify objects in batch, select the target object, and do the following:

1. To call batch modifying command, right click on the function window, select **Batch Modifying**.
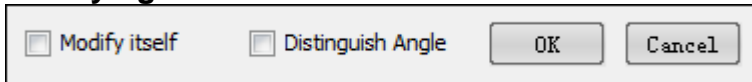
2. **Optional:** Do one of the following based on your needs:
   o To modify the target object only, check **Modify Itself**:
     With it unchecked, the system modifies objects with the same shape and size.
   o Not to modify objects with the same shape and size but with different rotation angles, check **Distinguish Angle**.

5 Technics

This section introduces technics you can set in **Double-tool CNC System**.

If there is no special instruction, please set technics in **Operator** page.

5.1 Add a Chamfer

It is used to add chamfers to objects whose angles are less than 180°, so as to improve cutting effect on corners of thick material.

To add a chamfer, select the target object, and do the following:

1. To open **Chamfer** dialog box, right click on the function window, and select **Chamfer**:
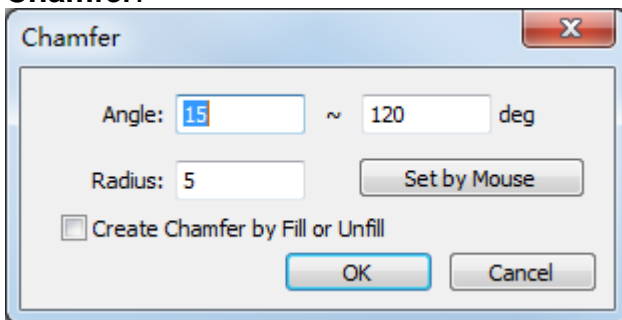
2. Set parameter **Angle** and **Radius**.
3. **Optional:** To automatically add chamfers for closed objects according to the attribute of fill/unfill, check **Create Chamfer by Fill or Unfill**.

The system automatically adds a chamfer at the angle that meets the demand.

Taking setting parameter **Angle** to 45°~90° and checking **Create Chamfer by Fill or Unfill** as an example, the result is as follows:

Unfill   Fill
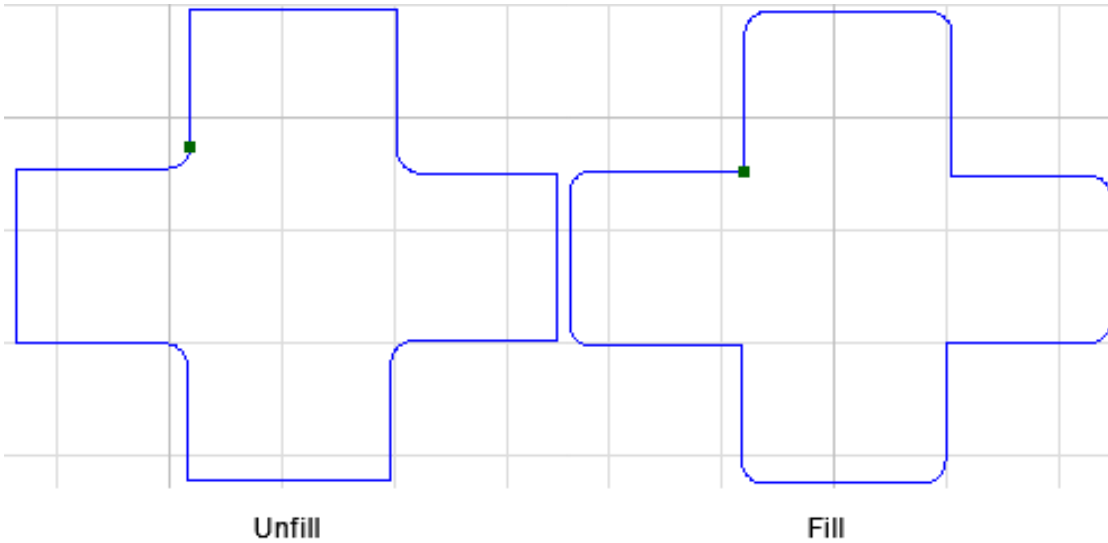
5.2   Set Unfill or Fill

It is used to reserve a certain part of closed objects. For objects with **Unfill** attribute, the system reserves the outer part; for objects with **Fill** attribute, the system reserves the inner part.

To set unfill or fill, select closed objects, and do the following:

1. Right click on the function window, and click **Unfill/Fill**.
2. Select one of the following:
    o   To manually set the selected objects as unfill/fill, select **Unfill** / **Fill**.
    o   To set the selected objects as unfill by default, select **Auto Setting**.

5.3   Set a Lead Line

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

The type of lead line differs in the leading direction:

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

To set a lead line, select the target object, and do the following:

1. To open **Set** dialog box, right click on the function window, select **Lead Line → Set**.
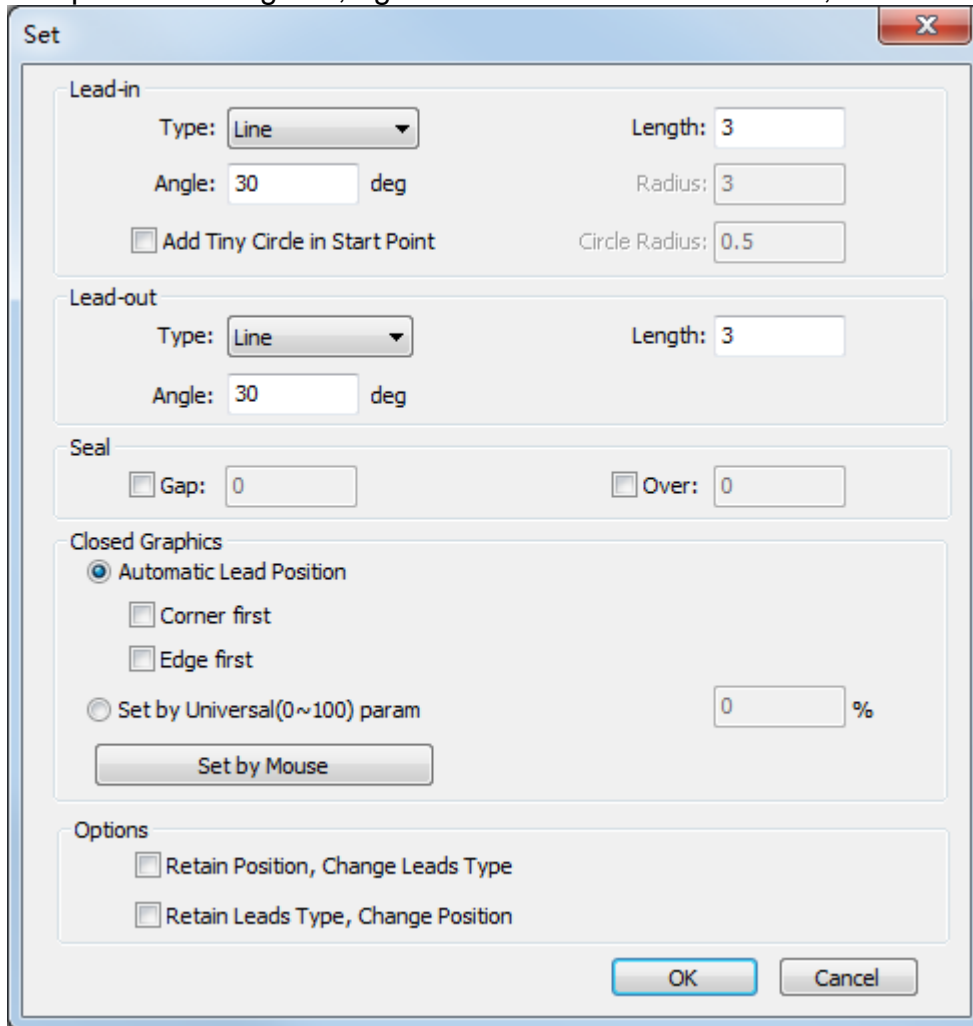
2. Select a type for the lead line according to the cutting technics.
3. Set related parameters of the lead line.
   See **Parameters** for details.
4. **Optional:** If the target object is a closed object, in **Seal** area, select one of the following:
   o **Gap**: The lead line is unclosed and the cutting head will not cut through at the sealing position.
   o **Over**: The lead line is closed and the cutting head will cut at the sealing position.
5. Set the position of the lead line:
   o If you check **Automatic Lead Position**, do one of the following:
     ▪ To automatically set the lead line at the position with a larger angle in the object, check **Corner first**.
     ▪ To automatically set the lead line at the longest edge of the object, check **Edge first**.
   o If you check **Set by Universal (0~100) param**. The position of the lead line is the set ratio times the total length.
   o If you check **Set by Mouse**, click on the edge of the object after the cursor turns into to specify the position of the lead line.

6. In **Options** Area, select one of the following:
   o **Retain Position, Change Leads Type**
   o **Retain Leads Type, Change Position**
7. **Optional:** Manually modify the lead line:
   a. To call **Start Point** command, Right click on the function window, select **Lead Line → Set Start Point**.
   b. Do one of the following:
      ▪ To modify the position of the lead-in line, left click on the edge of the object.
      ▪ To draw a line-type lead-in line from outside to the edge on the object, left click on the outside and the edge of the object.

The result is as follows:

Line
Angle: 60deg
Length: 5mm

Hook
Radius: 3mm
Length: 5mm

Arc
Angle: 60deg
Length: 5mm

Line
Angle: 60deg
Length: 5mm
Hole radius: 1mm

Hook
Radius: 3mm
Length: 5mm
Hole radius: 1mm
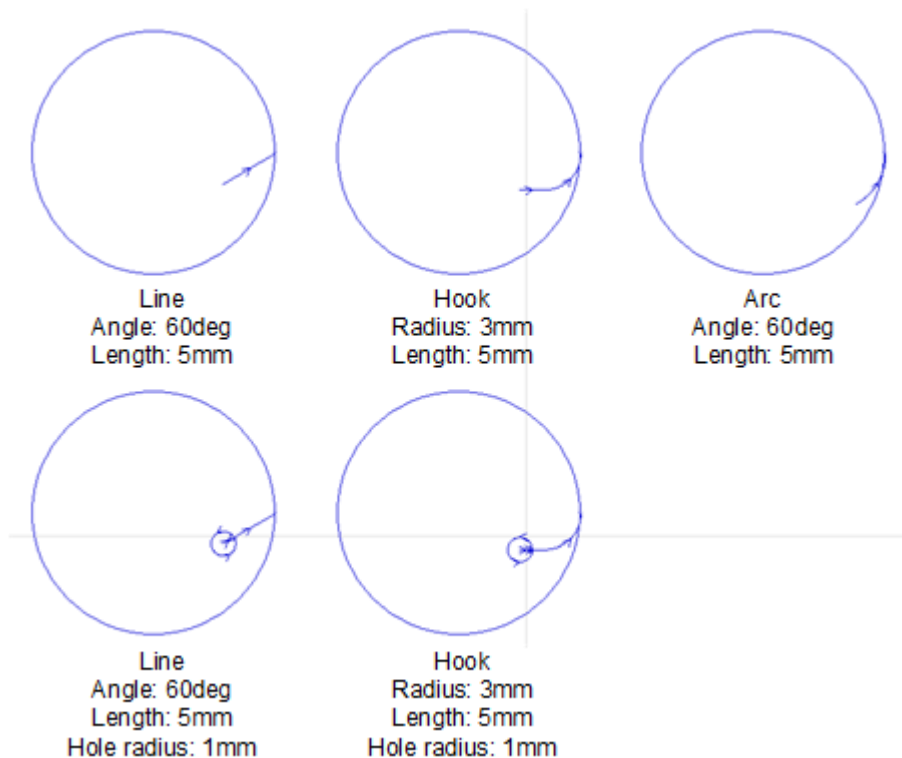
Parameters

- **Angle**: For a line-type lead line, it refers to the angle between the lead line and the tangent line of the intersection; for an arc-type lead line, it refers to the central angle.
- **Length**: For a line-type/arc-type lead line, it refers to the length of the line/arc; for a hook-type lead line, it refers to the sum of the radius of the arc and the length of the line.
- **Radius**: For a hook-type lead line, it refers to the radius of the arc.
- **Add Tiny Circle in Start Point**: Add a tiny circle at the start point of the lead-in line, so as to solve the problem that the accumulation of slag influences cutting effect during piercing a thick tube.
- **Circle Radius**: The radius of the hole at the start point of the lead line.

5.4   Set Kerf Compensation

There exists deviation between the sizes of actual cutting parts and theoretical ones, which is caused by the kerf during cutting. The deviation leads to a smaller size in the outer contour and a larger size in the inner contour. This operation is used to compensate the deviation.

The type of kerf compensation includes the following:

- All shrink: shrink the cutting area for all selected parts.
- All expand: expand the cutting area for all selected parts.
- Unfill: shrink; fill: expand: shrink the cutting area for parts with unfill attribute, and expand the cutting area for parts with fill attribute.

To compensate the kerf, select the closed objects, and do the following:

1. To open **Kerf Compensation** dialog box, right click on the function window, and select **Kerf Compensation**:
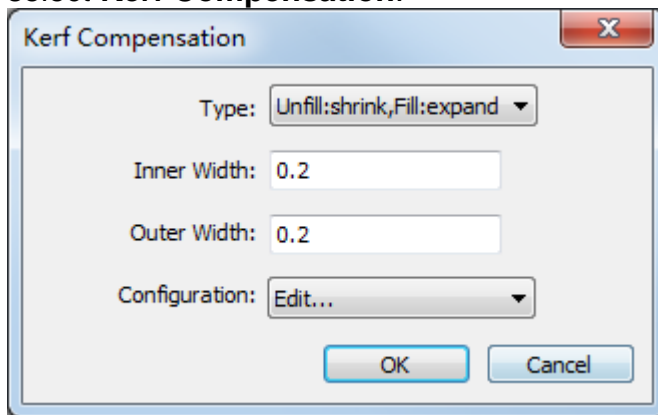
2. Select a compensation type.
3. Set the inner width and outer width.
4. **Optional:** To save the commonly used inner width and outer width for later use, do the following:
    a. In the drop-down box of **Configuration**, select **Edit**. **Configuration** dialog box pops up:
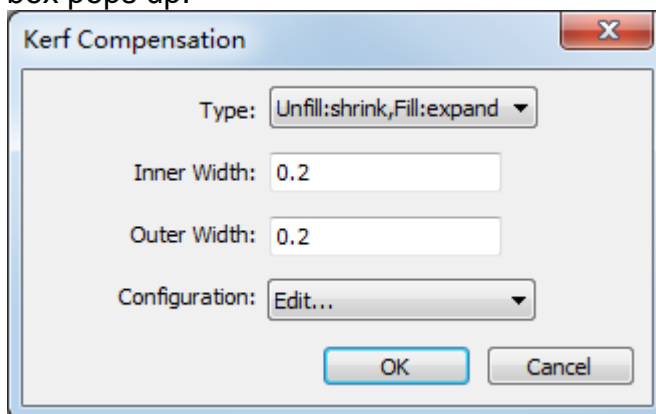
    b. Click **Add**, set a name in **Description** column, set the inner width in **Inner Width** column, and set the outer width in **Outer Width** column.
    c. For later use, in the drop-down box of **Configuration**, select the set name. The system automatically fills in the inner width and outer width.

5.5   Change the Machining Order

It is used to change the machining order. After drawing several objects, the default machining order is the drawing order.

Before changing the machining order, to show the machining order, in the top of the function window, click [icon].

To change the machining order, do one of the following:

- Automatically change the machining order
- Manually change the machining order
- Specify the machining order for a single object
- Change the machining order in a list
- Change the machining order by manual drawing
- Sort to the top or bottom

5.5.1   Automatically Change the Machining Order

The system automatically sorts objects according to the selected sorting strategy.

To automatically change the machining order, select the target objects, and do the following:

1. To open **Sort**, right click on the function window, and select **Machining Order → Auto Sort**:
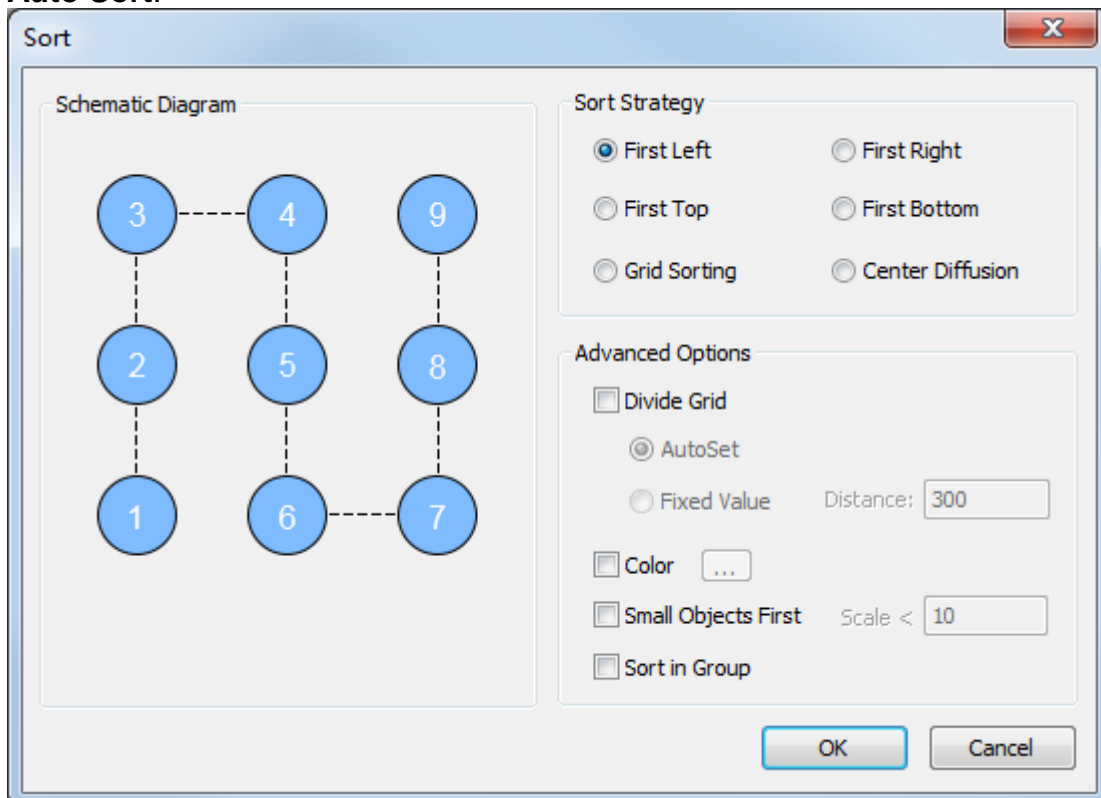
2. In **Sort Strategy** area, select a sorting strategy.

3. In **Advanced Options**, check advanced option(s):
   - **Divide Grid**
     - **AutoSet**: to automatically divide objects into the same group based on the distance and count.
     - **Fixed Value**: the objects whose distance is less than the set value will be divided into the same group.
   - **Color**: to sort objects in the part according to the layer order. It only applicable to the part with nested relation.
   - **Small Objects First**: to sort objects whose sizes are less than the set value. It only applicable to part interior with nested relation.
   - **Sort in Group**: to sort objects in groups according to the sort strategy.

To separately set the order for the objects in a group, right click on the function window, click **Machining Order → Sort within Group**, and select a sorting strategy and advanced option(s) in **Sort** dialog box:
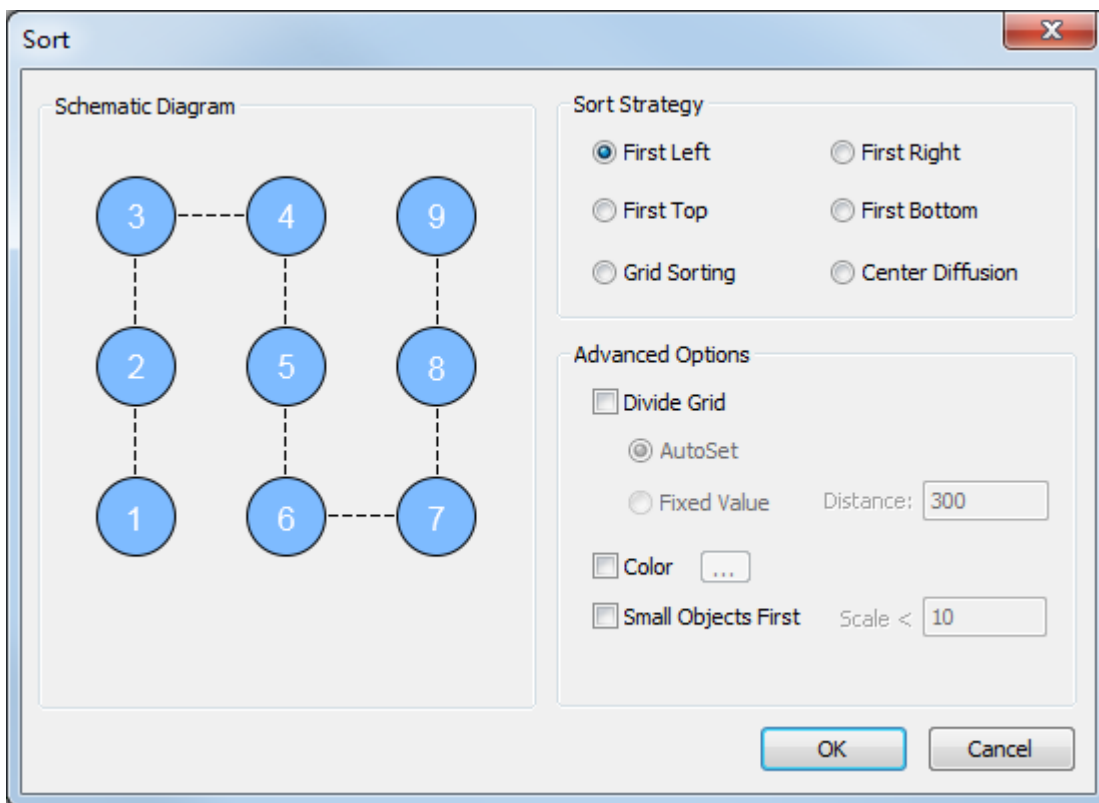
5.5.2 Manually Change the Machining Order

It is used to manually specify the machining direction for a single object or several objects.

To manually change the machining order, do the following:

1. To call **Manual Sorting** command, do one of the following:
   - Select the target objects, right click on the function window, and select **Machining Order → Manual Sort**.
   - Right click on the function window, and select **Manual Sort**.

   The cursor turns into ①, and the system automatically shows the machining order.

2. Select the first target object.

The cursor turns into ②. The machining order on the object turns into 1, and the order of other objects turns into 2,3 ... in sequence on the basis of the previous order.
To reset the last order number, right click and select **Previous Order**.
3. Repeat step 2 and click the objects in order.

If you need to exit manual setting tool, do one of the following:

- Right click and select **Exit**.
- Press **Esc**.

5.5.3 Specify the Machining Order for a Single Object

It is used to specify the machining order for a single object.

To specify the machining order for a single object, select a single object, and do the following:

1. To open **Specify Order Separately** dialog box, on the function window, right click and select **Machining Order** → **Specify Order**:
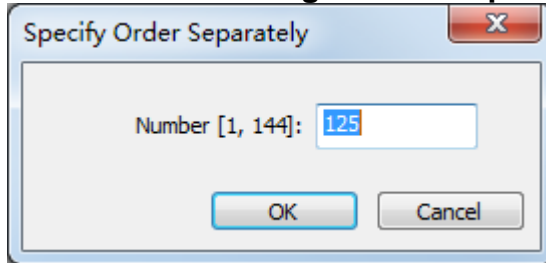
2. To specify an order, enter an order in **Number [1, n]** input box.
   n is the maximum machining order in the toolpath.

5.5.4 Change the Machining Order in a List

Each object has a number. This operation is used to manually change the machining order according to the number.

To change the machining order in a list, do the following:

1. Select an object.
2. To open **Order List** dialog box, on the function window, right click and select **Machining Order → Order List**:
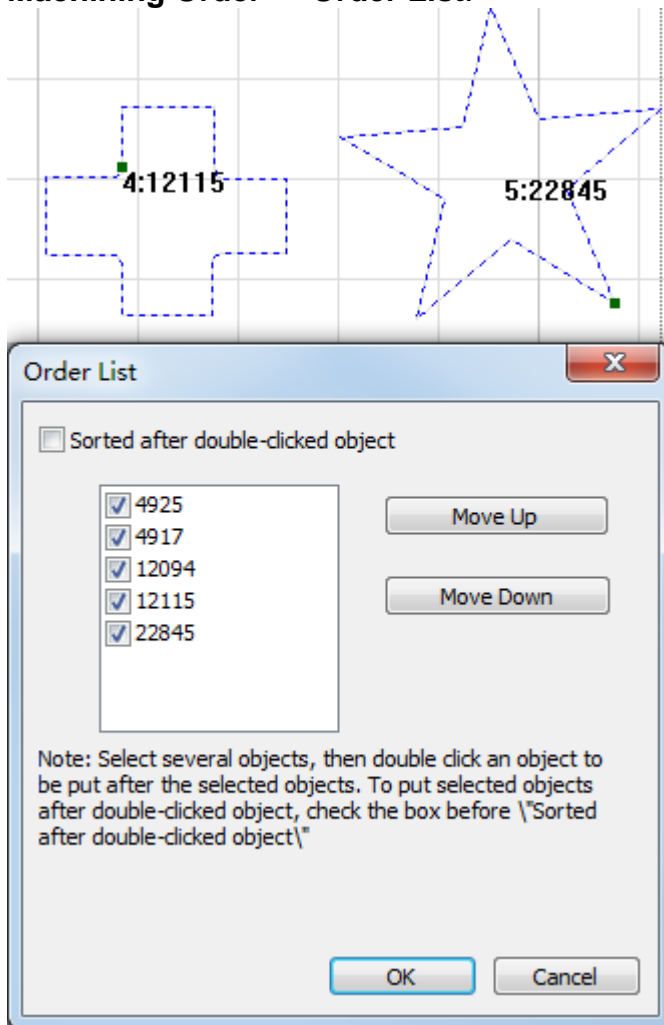
3. Check the object, and do one of the following to sort the machining order:
   - Click **Move Up**/**Move Down** to move the object.
   - Double click another object to move the checked object to the clicked position.

5.5.5 Change the Machining Order by Manual Drawing

It is used to manually change the machining order of two objects, and then change the whole machining order.

To change the machining order by manual drawing, select the target objects, and do the following:

1. To call **Manual Draw Sorting** command, do one of the following:
   - Right click on the function window, click **Machining Order → Draw Connection Line**.
   - Select no object, right click on the function window, click **Draw Connection Line**.

2. Select a single object. The object is highlighted.
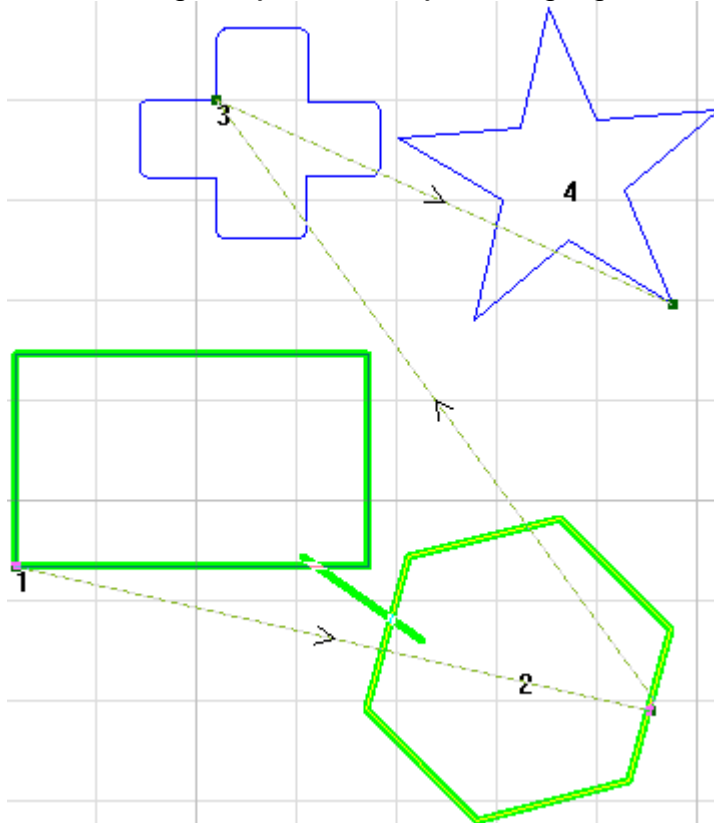
3. Hold the left button, draw a line from the object, connect it to another object (the target object), and release the mouse until the target object is highlighted.

The machining order of the target object is behind the selected object at the first time.

After changing the machining order by manual drawing, to exit **Manual Draw Sorting** command, right click and select **Exit**, or press **Esc**.

5.5.6 Sort to the Top or Bottom

It is used to turn the machining order of a single object to the top or bottom.

To sort to the top or bottom, select a single object, Right click on the function window, and select **Machining Order → Sort to Top** / **Sort to Bottom**.

5.6 Change the Machining Direction

It is used to change the machining direction for all closed objects. After drawing an object, the default machining direction is counterclockwise.

Before changing the machining direction, to show the machining direction, in the top of the

function window, click .

To change the machining direction, select the target object(s), and do the following:

1. To open **Mach Direction** dialog box, right click on the function window, select **Machining Direction → Set**:
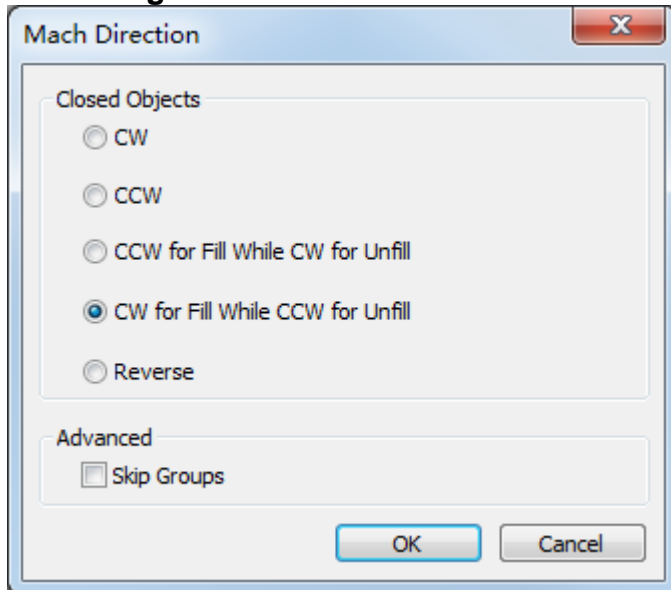
2. Select the machining direction based on your needs.
   Alternatively, if you just need to reverse the machining direction, do one of the following:
   o Right click on the function window, select **Machining Direction → Reverse**.
   o In **Mach Direction** dialog box, select **Reverse**.
3. **Optional:** To skip the machining direction of groups, in **Advanced** area, check **Skip Groups**.

5.7　Set Punching Parameters

Waterjet cutting uses high-speed and high-energy water current to cut material. Before feeding motion, the water current should penetrate the material. For hard and thick material, this cutting method can help to improve the machining efficiency and save material in a great extent.

To set punching parameters, select the target object, and do the following:

1. To open **Punch** dialog box, in the left of the function window, click   :
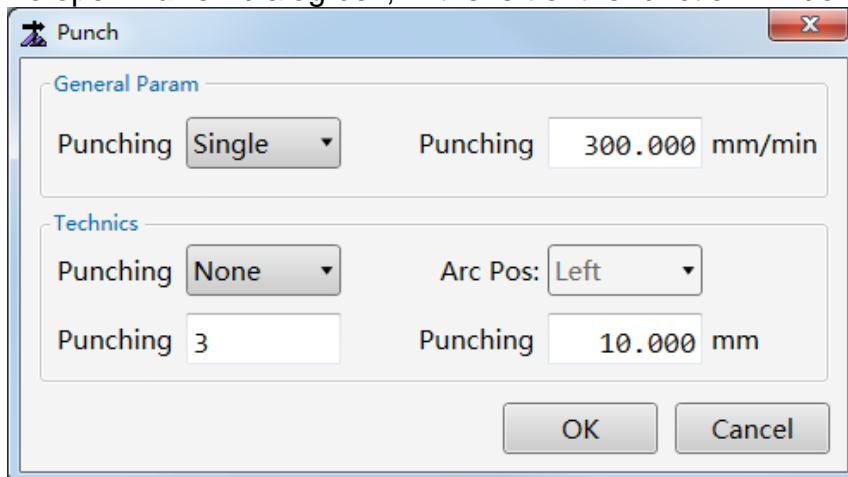
2. In **General Param** area, set the following parameters:
   - **Strategy**: it includes the following:
     - Single punching: during cutting multiple objects, each object is punched and cut before cutting the next one.
     - Batch punching (only support NCE file): during cutting multiple objects, all objects are punched first before cutting them.
   - **Speed**: The speed for punching.
3. In **Technics** area, set the following technics:
   - **Type**
     - Arc punching: circle is punched at the start point of the lead line.
     - Swing punching: straight-line is punched at the start point of the lead line.
   - **Arc Pos**
     - Left: the circle is on the left of the start point of the lead line.
     - Middle: the circle is at the middle of the start point of the lead line.
     - Right: the circle is on the right of the start point of the lead line.
   - **Times**: the machining times of circle for arc punching, and the swinging times of straight-line for swing punching.
   - **Distance**: the punching diameter for arc punching, and punching interval for swing punching.

5.8　Set Cutting Speed

It is used to set the cutting speed for the target layer so as to set the cutting speed for objects in the layer.

In the left of the function window, click [Layer] and set the cutting speed for the target layer in **Layer Setting** dialog box.

5.9　Set the Berth Point

A point on the target object will coincide with the workpiece origin according to the set berth point. For example, if the berth point is set as bottom left, the upper right corner of the target object will coincide with the workpiece origin.

It is used with the operation of setting the workpiece origin.

To set the berth point, do the following:

1. To open **Set Berth Point** dialog box, in the left of the function window, click [icon]:
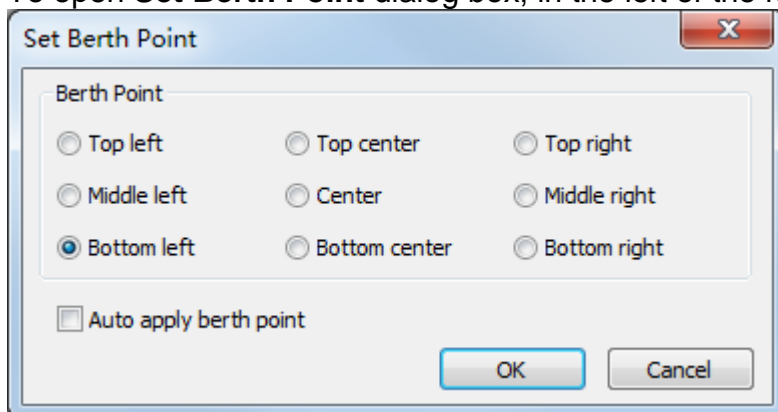
2. Select the berth position.

3. **Optional:** To automatically make the object move to the workpiece origin by the set berth point when the workpiece origin is set again, check **Auto apply berth point**.

5.10 Do Clearing

It is used to clear some set technics, including lead line, kerf compensation and punching.

To do clearing, select the target object, right click on the function window, click **Clear →
Lead Line** / **Kerf Compensation** / **Punch**.

6 Special Operations

This section mainly introduces special operations in **Double-tool CNC System**.

Special operations mainly include the following:

- Select the tool(s)
- Execute Z-axis jiggle
- Execute Y1-axis and Y2-axis jiggle
- Use remote assistance
- Set safety light curtains
- Start Machining from the positioning point
- Optimize acceleration/deceleration at corners

6.1 Select the Tool(s)

It is used to choose the tool(s), which is convenient to execute adjustment and use another tool to continue cutting in case of something wrong with one tool.

Before selecting the tool(s), set parameter **Min Distance between X1 and X2** and **Max Distance between X1 and X2** to protect tools from collision.

To select the tool(s), select the tool(s) in **Tool Selection** area in **Operator** page or **Technician** page:

- Select tool 1 or tool 2
    - During manual movement, only the selected tool moves.
    - During automatic machining, the double tools move synchronously, but the unselected tool does not do cutting actions and its Z-axis does not move for safety.
- Select tool 1 and tool 2
    - During manual movement, the double tools move at the same time.
    - During automatic machining, the system synchronizes the workpiece coordinates of the double tools and keeps synchronous cutting.

6.2   Execute Z-axis Jiggle

It is used to adjust height of the tool, so as to find the best cutting height.

Before executing Z-axis jiggle, make sure the system is in **Running** or **Pause** status.

To execute Z-axis jiggle, do the following:

1. To open **Parameter** dialog box, do one of the following:
   o In **Operator** page, click ☰ → **Parameters**.
   o In **Technician** page, click **System** → **Parameters**.
2. Set the following parameters:
   o **Jiggle Distance(Z1)** / **Jiggle Distance(Z2)**: the moving distance of the machine tool corresponding to each jiggle.
   o **Jiggle Speed(Z1)** / **Jiggle Speed(Z2)**: the speed during jiggle.
3. To adjust the tool to the best cutting height, click **Z1+** / **Z1-** / **Z2+** / **Z2-** in the operation control area.

6.3   Execute Y1-axis and Y2-axis Jiggle

It is used to eliminate offset and reset datum of Y-axis when offset exists in the machine coordinates of Y1-axis and Y2-axis, so as to simplify operation and save time.

It is only available to machine tools with diverter double Y whose Y1-axis and Y2-axis motors are connected to two separate interfaces of the terminal board.

To execute Y1-axis and Y2-axis jiggle, do the following in **Technician** page:

1. To open **Y1Y2 Jiggle** dialog box, in the menu area, click **Machine** → **Y1Y2 Jiggle**, and turn switch **Disable Double Y Adjustment** to **ON** status:
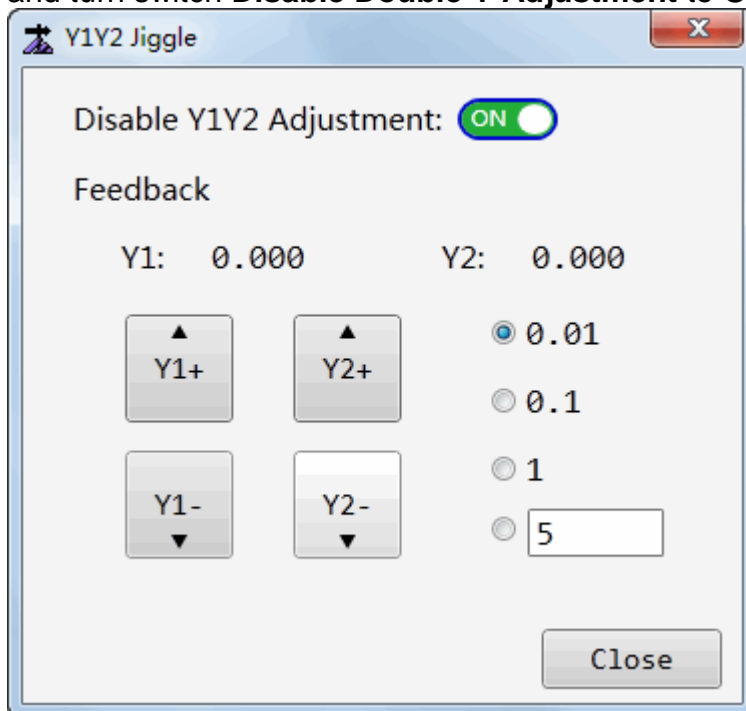
2. Select or customize a step size.
3. According to Y1-axis and Y2-axis offsets, click **Y1+** / **Y1-** / **Y2+** / **Y2-** to move Y1-axis or Y2-axis.

6.4 Use Remote Assistance

It is used to use remote assistance to control your system, which is good for commissioning and troubleshooting.

Before using remote assistance, make sure the system has been networked.

To use remote assistance, do the following in **Technician** page:

1. In the menu area, click **System → Remote Support**. The following dialog box pops up:

2. Send the identification code to someone who can help.

6.5 Set Safety Light Curtains

It is used to set safety protection around the machine tool, so as to avoid that outside objects enter into the machine tool during running.

To set safety light curtains, do the following:

1. To open **Parameters** dialog box, do one of the following:
   o In **Operator** page, click ▤ **→ Parameters**.
   o In **Technician** page, click **System → Parameters**.
2. Select **Manufacturer** permission, find and set parameter **Lower Limit of Safety Grating** and **Upper Limit of Safety Grating**.

When tool 1 and tool 2 both move out of the set area of safety light curtains, the system automatically enters into **Pause** status and locks the screen to avoid accident.

6.6   Start Machining from the Positioning Point

It is used to cut the toolpath from the target position to the end of the toolpath again after machining finishes, if you are not satisfied with the machining result.

With the selected point on the function window as the circle center, the system draws circles whose radius is within 15mm. The point nearest to the circle on the graph is the positioning point.

To start machining from the positioning point, do the following in **Operator** page:

1. Select a point near the graph on the function window.
2. Right click, and do one of the following:
   - If you are not sure whether the positioning point is as expected, select **Locate to Here** to check the positioning point, and select **Start Machining from Here** if it is as expected.
   - If you are sure the positioning point is as expected, select **Start Machining from Here**.

The system starts machining from the positioning point.

6.7   Optimize Acceleration/Deceleration at Corners

It is used to optimize the cutting effect at corners for cutting thick metal.

To optimize acceleration/deceleration at corners, do the following:

1. To open **Parameter** dialog box, do one of the following:
   - In **Operator** page, click → **Parameters**.
   - In **Technician** page, click **System** → **Parameters**.
2. Select the manufacturer permission, set and find the following parameters:
   - **Acc and Dec at Inflection Point**: whether to accelerate and decelerate at inflection point.
   - **Speed Rate at Corner**: to control the cutting speed at corners, it equals to the cutting speed times the percent of speed at corners.
   - **Accelerating Distance at Inflection Point**: how far from the inflection point to start accelerating after the corner is cut.
   - **Decelerating distance at Inflection Point**: how far from the inflection point to start decelerating when the system is to cut the corner.
   - **Decelerating Distance for Last Segment**: the decelerating distance for the last segment of object.
   - **Min Angle to Decelerate at Corner**: only when angle at corner is greater than the minimum deceleration angle at the inflection point, the speed at corners is available.

7   Common Operations

This section mainly introduces common operations in **Double-tool CNC System** to help machining.

Common operations mainly include the following:

- Customize common parameters
- Execute I/O port related operations
- Check system logs
- Generate an installation package

If there is no special instruction, please execute common operations in **Technician** page (Manufacturer password is required).

7.1   Customize Common Parameters

It is used to make the parameter list to suit your own needs, so that you can quickly find and manage frequently used parameters.

To customize common parameters, do the following:

1. To switch to **Common Parameters** window, click **Common parameters** in the area of function window.
2. To open **Common Parameters** dialog box, click **Set** in the lower right corner:
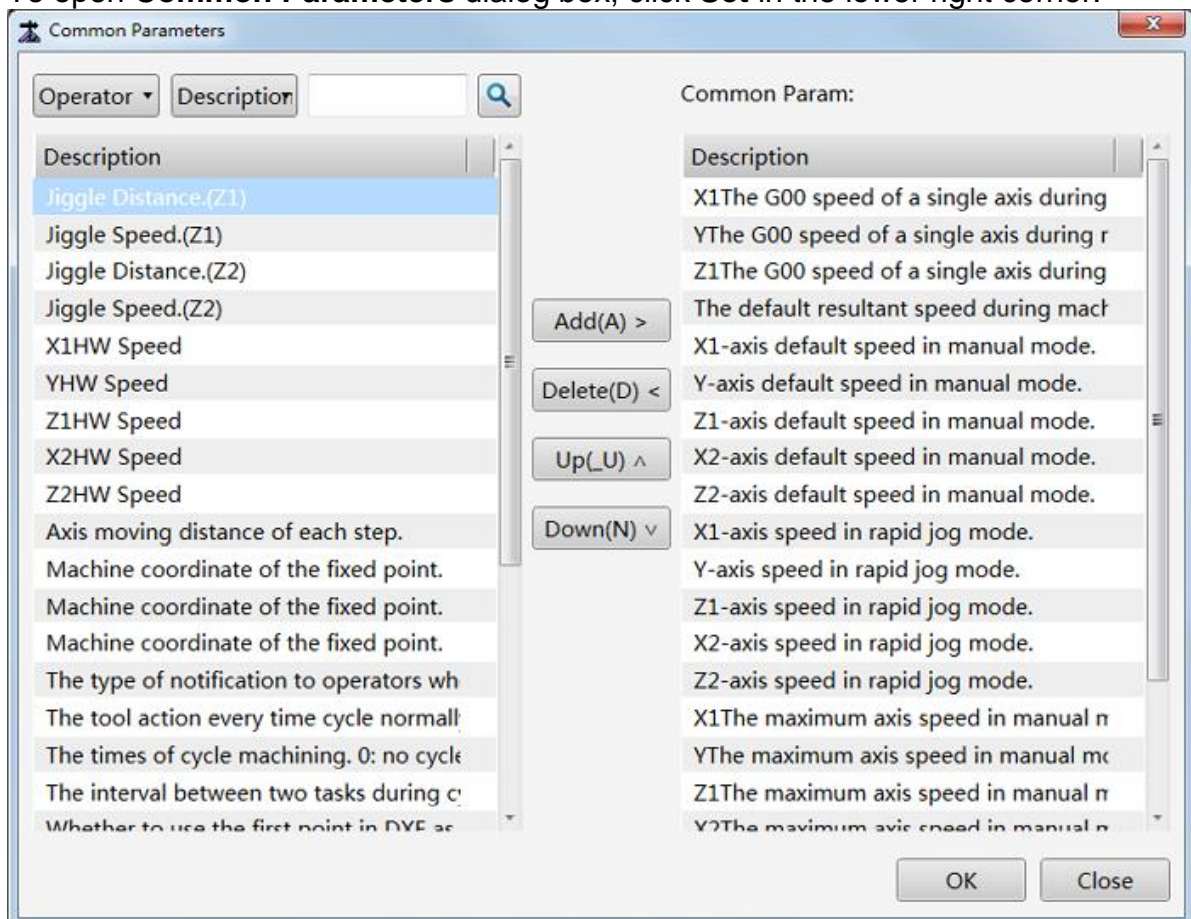
3. Find the parameters that you would like to add as a common parameter by one of the following:
   o Permission: including operator and manufacturer. Operator permission is the default.
   o Search condition: by description and addressing. Search by description is the default.
4. To add the parameter to the common parameter list, click **Add**.
5. Repeat step 3 and 4 until all target parameters are added.
6. **Optional:** To move a parameter out of the common parameter list, select the parameter in the common parameter list, and click **Delete**.

The customized parameters instantly show in **Common** window.

7.2 Execute I/O Port Related Operations

By controlling over input or output ports, it is used to monitor the status of the machine tool, conduct a simulation test, and so on.

To execute I/O port related operations, do the following:

1. To switch to **Port Info** window and check port information, click **Port** in the area of function window:

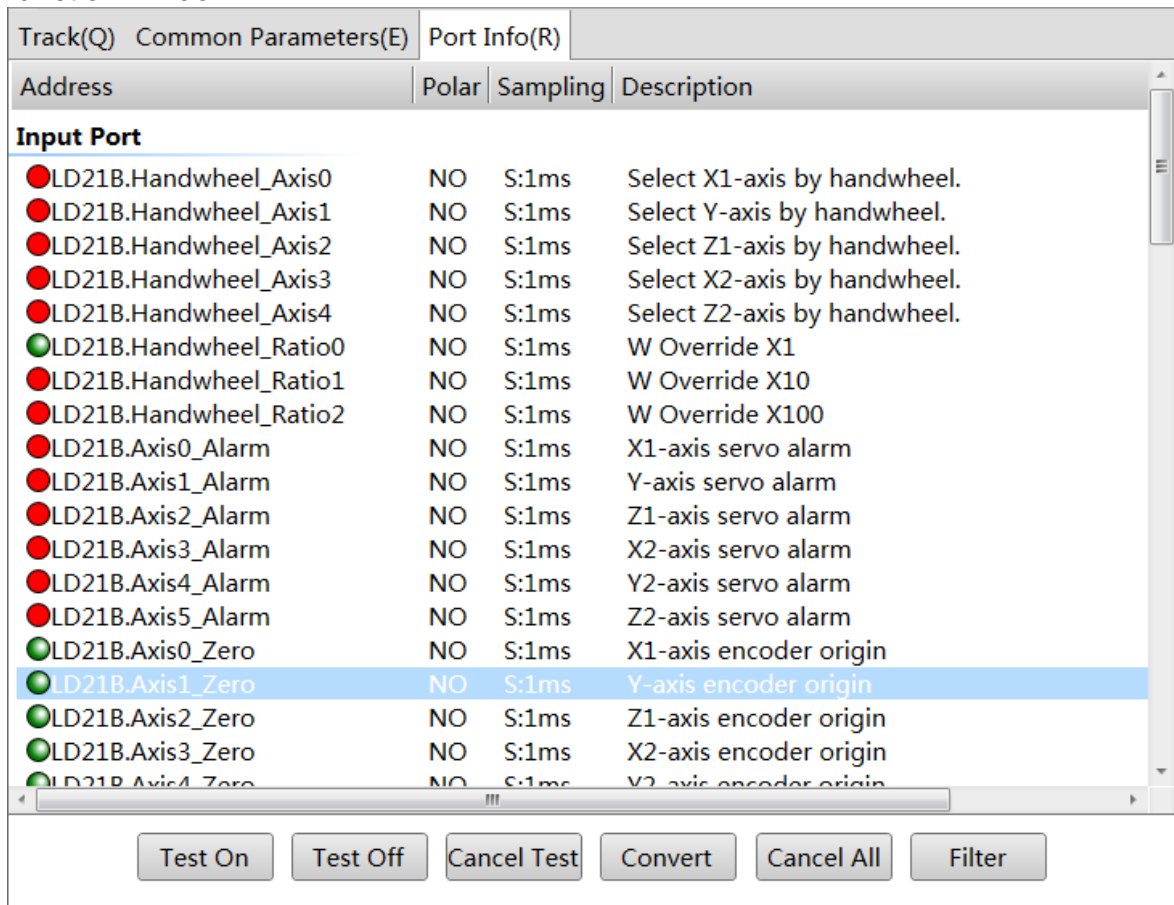| Track(Q)	Common Parameters(E)	Port Info(R)		
Address		Polar	Sampling	Description
Input Port				
🔴 LD21B.Handwheel_Axis0		NO	S:1ms	Select X1-axis by handwheel.
🔴 LD21B.Handwheel_Axis1		NO	S:1ms	Select Y-axis by handwheel.
🔴 LD21B.Handwheel_Axis2		NO	S:1ms	Select Z1-axis by handwheel.
🔴 LD21B.Handwheel_Axis3		NO	S:1ms	Select X2-axis by handwheel.
🔴 LD21B.Handwheel_Axis4		NO	S:1ms	Select Z2-axis by handwheel.
🟢 LD21B.Handwheel_Ratio0		NO	S:1ms	W Override X1
🔴 LD21B.Handwheel_Ratio1		NO	S:1ms	W Override X10
🔴 LD21B.Handwheel_Ratio2		NO	S:1ms	W Override X100
🔴 LD21B.Axis0_Alarm		NO	S:1ms	X1-axis servo alarm
🔴 LD21B.Axis1_Alarm		NO	S:1ms	Y-axis servo alarm
🔴 LD21B.Axis2_Alarm		NO	S:1ms	Z1-axis servo alarm
🔴 LD21B.Axis3_Alarm		NO	S:1ms	X2-axis servo alarm
🔴 LD21B.Axis4_Alarm		NO	S:1ms	Y2-axis servo alarm
🔴 LD21B.Axis5_Alarm		NO	S:1ms	Z2-axis servo alarm
🟢 LD21B.Axis0_Zero		NO	S:1ms	X1-axis encoder origin
🟢 LD21B.Axis1_Zero		NO	S:1ms	Y-axis encoder origin
🟢 LD21B.Axis2_Zero		NO	S:1ms	Z1-axis encoder origin
🟢 LD21B.Axis3_Zero		NO	S:1ms	X2-axis encoder origin
🟢 LD21B.Axis4_Zero		NO	S:1ms	Y2-axis encoder origin

[Test On] [Test Off] [Cancel Test] [Convert] [Cancel All] [Filter]

2. Do one of the following:
   o To conduct or close a simulation test, click **Test On** and **Test Off**.
   o To cancel a simulation test, click **Cancel Test**.
   o To modify the port polarity, click **Convert**.
   o To set the sampling interval and enable/disable filter, click **Filter**.

7.3 Check System Logs

It is used to check different types of system logs, and it is good for troubleshooting.

To check system logs, do the following:

1. To open **Log** dialog box, in the menu area, click **System → Log**:
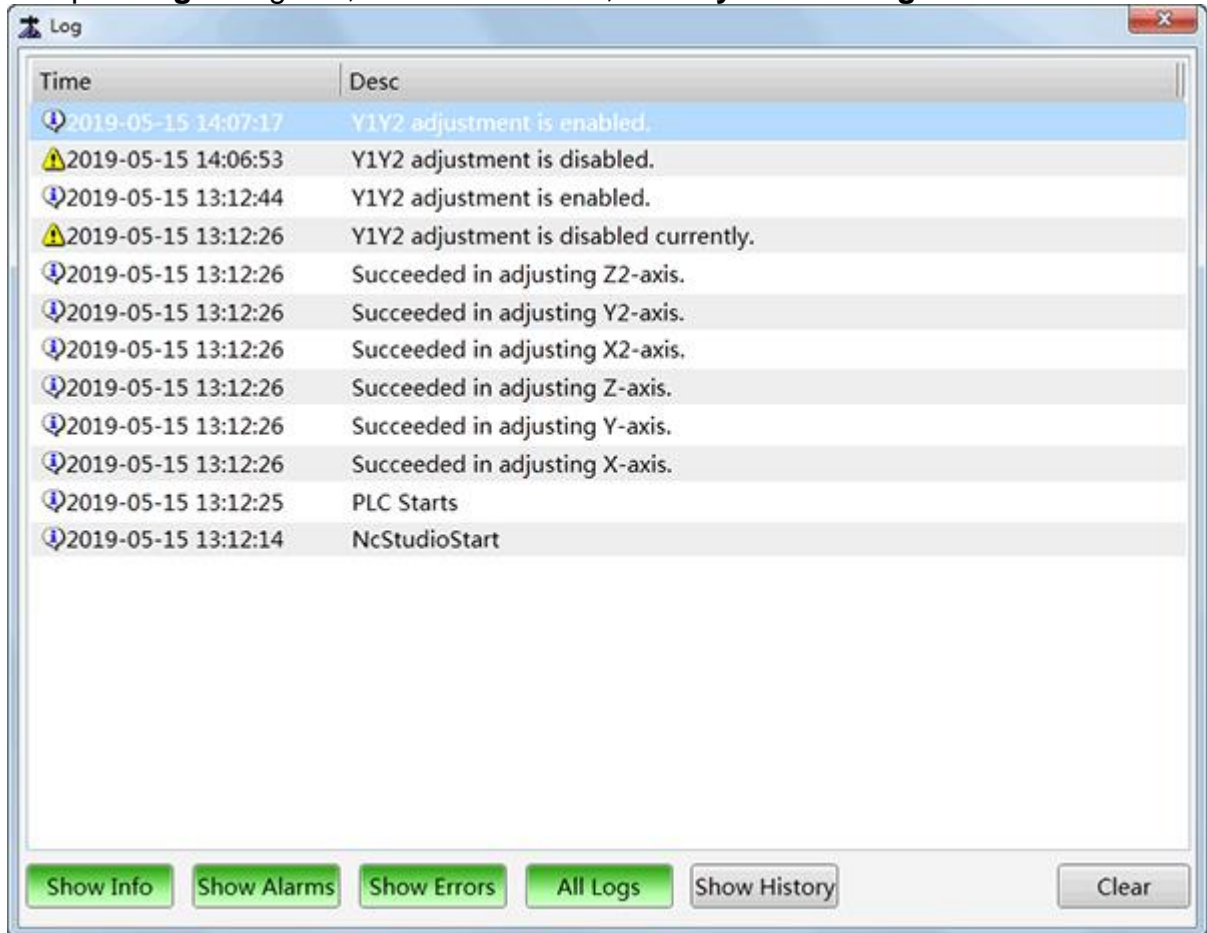
2. Check different types of system logs for troubleshooting:
   o To show logs about running status, click **Show Info**. Logs with icon 🛈 show in the dialog box.
   o To show warning logs, click **Show Alarms**. Logs with icon ⚠ show in the dialog box.
   o To show error logs, click **Show Errors**. Logs with icon ❌ show in the dialog box.
   o To show logs since power on, click **All Logs**.
   o To show all logs in the past, select at least a kind of the above log type, and click **Show History**.

   The first four types of system logs are shown by default.
3. **Optional:** To clear all logs, click **Clear**.
   Note: Please regularly do clearing. Otherwise, too many log files will slow down the system.

7.4   Generate an Installation Package

It is used to create a complete installation package on the basis of current system data, which is useful to back up system files and save a stable version of the system.

To generate an installation package, do the following:

1. In the menu area, click **File → Generate Installation Package**. **Packup Tool** dialog box pops up.
2. Input the machine model.
3. Modify the name of the installation package.
4. Click **Browse** to select the storage path for the generated installation package.
5. **Optional:** Click **Advanced**, and set the following advanced settings:
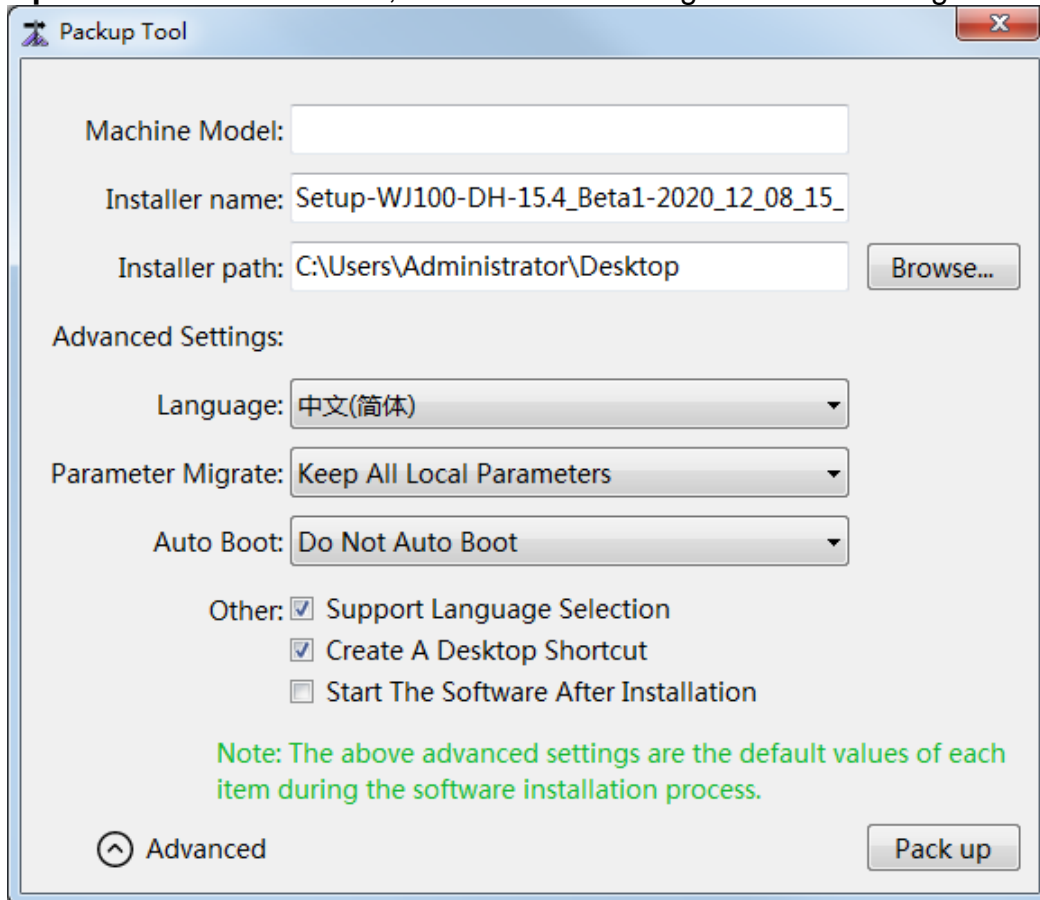
- o Language: at present, the system supports Chinese and English.
- o Parameter migration
  - **Keep All Local Parameters**
  - **Keep Only Machine-Specific Parameters**
  - **Fresh Installation**: Keep no parameters.
- o Auto boot: whether to automatically start the software once the machine boots.
- o Support language selection: whether to support switching the current language.
- o Create a desktop shortcut: whether to create a shortcut on the desktop.
- o Start the software after installation: whether to automatically start the software after installation.

6. Click **Pack up**. The system starts to generate the installation package.

After finishing, find the installation package under the set path.

8　FAQs

This section mainly answers common questions that you may encounter during using **Double-tool CNC System**.

Common questions can be divided into the following:

- Questions during returning to the machine origin
- Questions about warnings
- Questions about alarms

8.1　Questions during Returning to the Machine Origin

8.1.1　Cannot Detect Origin Signal

Cause

Error in the origin switch.

Solution

1. Touch the origin switch, check and observe related ports in the function window:
   - If they have signals, contact us.
   - If they have no signal, proceed to the next step.

   See Execute I/O Port Related Operations for checking I/O ports.
2. Check indicator lights of terminal **X00**, **X03**, **X06**, **X09** and **X12** on the terminal board.
   - If they are on, check the connecting line between the motion control card and the terminal board:
     - If it loosens, tighten it.
     - If it does not loosen, contact us.
   - If they are off, check the electric circuit between the origin switch and the terminal board:
     - If it loosens, tighten it.
     - If it does not loosen, replace the origin switch.

8.1.2　Incorrect Axis Direction

Cause

1. Incorrect polarity of port **Machine Origin**.
2. Incorrect value of parameter **Axis Direction**.

Solution

1. Convert its polarity.
   See Execute I/O Port Related Operations for details.
2. Change its value to the opposite value.
   See Adjust the Axis Direction for details.

8.1.3　Speed during Coarse Positioning is Too Slow

Cause

1. Too small value of parameter **Coarse Positioning Speed**.
2. The polarity of port **Machine Origin** does not match with type of the origin switch.

Solution:

1. Modify its value to a larger value.
2. Convert its polarity.
   See Execute I/O Port Related Operations for details.

8.2 Questions about Warnings

8.2.1 Simulation Result Shows Running Range Exceeded Machine Stroke. Switch to Log Window to Learn More

Cause

The workbench stroke in the toolpath file exceeds the set upper limit and lower limit of the workbench stroke.

Solution

1. Check the workpiece origin:
   o If it is set reasonably, proceed to the next step.
   o If it is not set reasonably, set it again.
     See Set the Workpiece Origin for details.
2. Check the workbench stroke in the toolpath file:
   o If it is reasonable, modify parameter **Upper Limit of Worktable Travel** and **Lower Limit of Worktable Travel** to enlarge the workbench stroke.
   o If it is not reasonable, modify the toolpath file.

8.2.2 Cannot Execute the Operation. The System has not Returned to the Machine Origin

Cause

The system has not returned to the machine origin.

Solution

Return to the machine origin first.

See Confirm Machine Coordinates of Double Tools for details.

8.2.3 The Parser is Busy. Cannot Execute the Operation

Cause

Cannot execute the operation in running status.

Solution

Execute the operation in idle status by stopping machining or waiting for the end of the machining.

8.3 Questions about Alarms

8.3.1 Limit Alarm

Cause

1. Incorrect polarity of port **Negative Limit** / **Positive Limit**.
2. Axis collided with the limit switch.
3. Limit switch failure.

Solution

1. Convert its polarity.
   See Execute I/O Port Related Operations for details.
2. Manually move the axis away from the limit switch.
3. Replace the limit switch.

8.3.2 Servo Alarm
Cause

An alarm in the servo drive.

Solution

Check the servo connecting wire.

8.3.3 E-stop Alarm
Cause

1. The E-stop button is pressed.
2. Incorrect polarity of port **E-stop**.
3. An alarm in the inverter.

Solution

1. Rotate the E-stop button clockwise.
2. Convert its polarity.
   See Execute I/O Port Related Operations for details.
3. Troubleshoot the alarm according to the alarm type of the inverter.

8.3.4 Alarm in Disconnection of Lambda Terminal Board
Cause

1. The cable of Lambda terminal board has not been firmly connected.
2. Failure of Lambda terminal board.

Solution

1. Re-plug the cable, and restart the software.
2. Check indicator lights on Lambda terminal board:
   o If they are on, contact us.
   o If they are off, replace the terminal board.

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.

SPECIALIZED / CONCENTRATED / FOCUSED