User Manual of NcStudio V10 Waterjet Cutting CNC System

Version: 2020.6 3rd Version
Author: Product Testing Department

1 Software Overview

Through this section, you can quick know the hardware and software of **NcStudio V10 Waterjet Cutting CNC System**.

Hardware

- Host
- Lambda terminal board
- EX series extended terminal board: EX31A

Software

NcStudio V10 Waterjet Cutting CNC System includes multiple softwares with different axes (General Five-axis AB, General Five-axis AC). This manual takes NcStudio V10 General Five-axis AC (Bus) as an example to introduce how to use this system.

About the software interface of **NcStudio V10 Waterjet Cutting CNC System**, see Software Interface.

1.1 Host

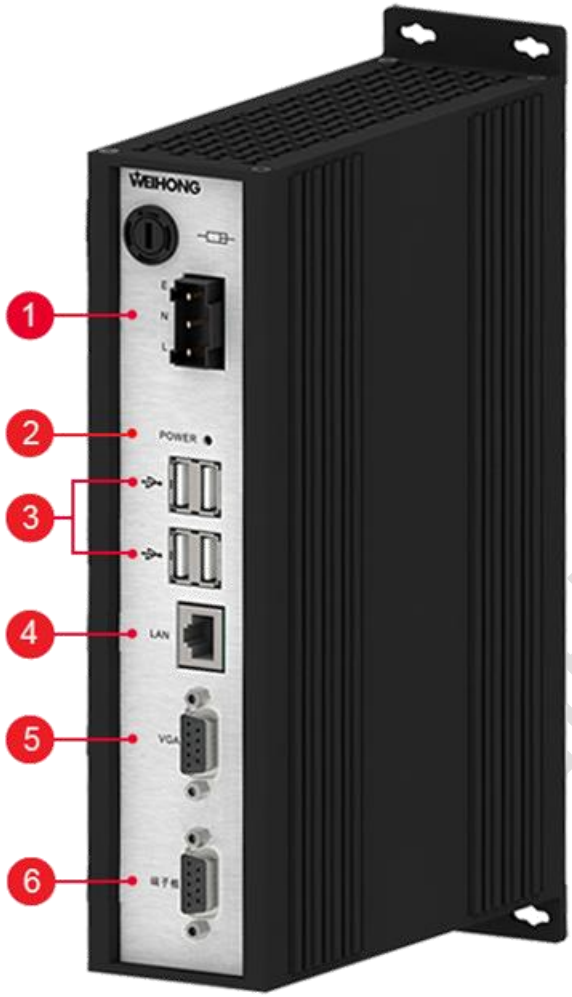
You can choose one of the following for host control:

- NC65C industrial personal computer
- PM95A control card

1.1.1 NC65C Industrial Personal Computer

NC65C industrial personal computer, also called **NC65C Controller** (hereinafter referred to as **NC65C**).

The structure and interface definition of **NC65C** are as follows:



1. Interface of LNE (live wire, neutral wire and earth wire)

   It is used to connect to a 220V power supply.

2. Indicator light

   It is an indicator light for the power supply.

3. USB 2.0 interface

   It is used to transmit data with a USB flash disk.

4. LAN interface

   Its transmission rate is 100 Mbps.

5. VGA interface for a display screen

   It is used to connect to a display screen.

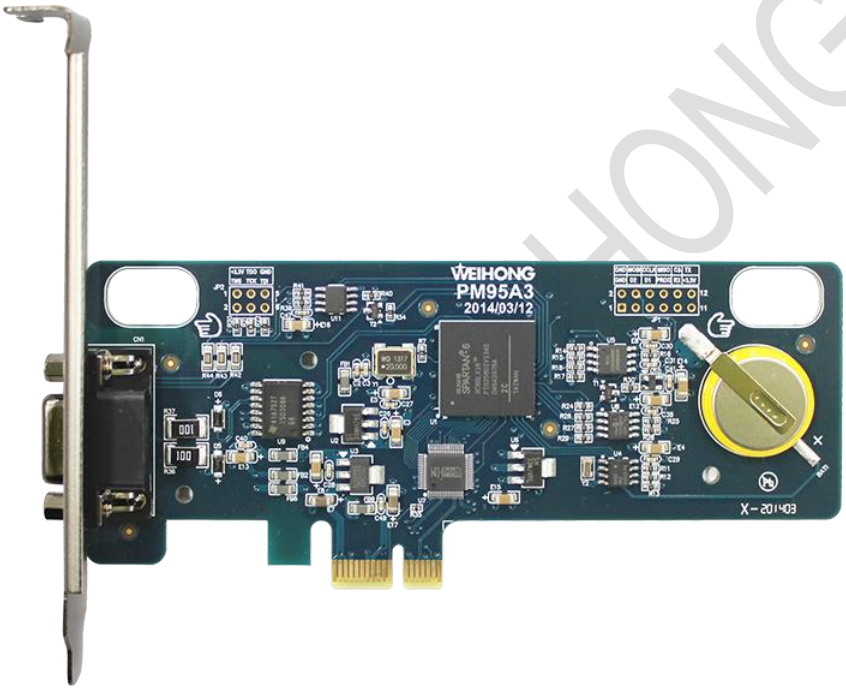
6. Interface for a Lambda terminal board

   It is used to connect to a Lambda terminal board.

1.1.2 PM95A Control Card

When PM95A control card is used with **NcStudio**, you need to insert it into PCI-E slot of your computer.

PM95A is shown as follows:



1.2 Lambda Terminal Boards

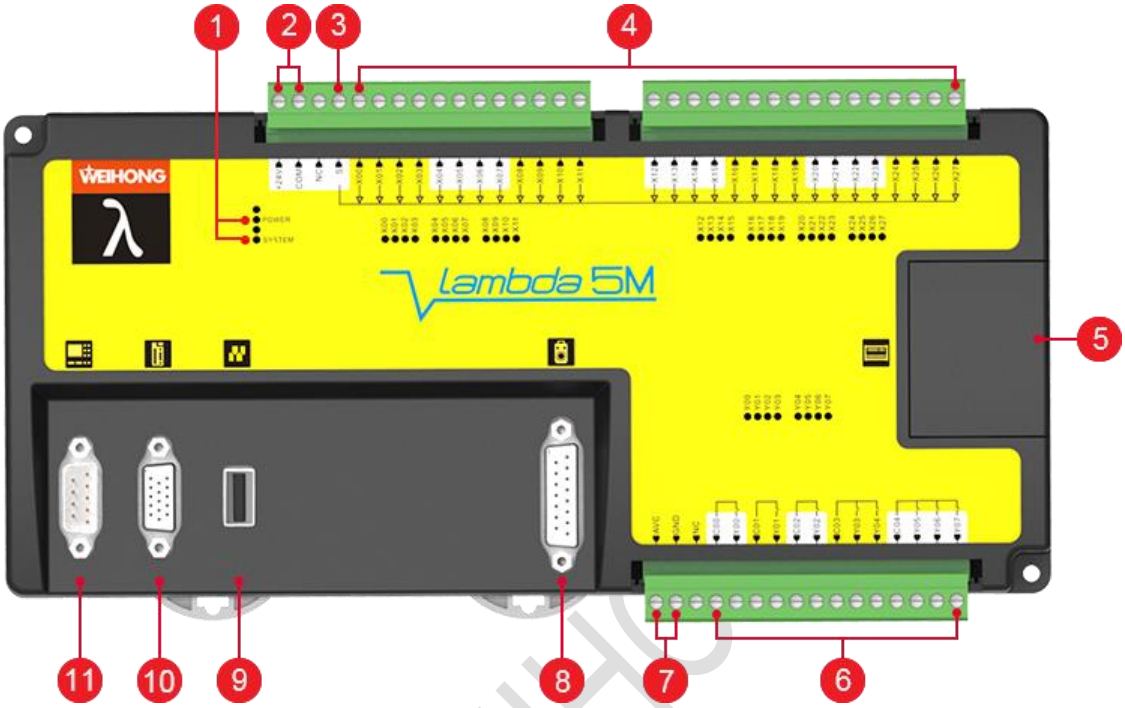
This system supports the following Lambda terminal boards:

- Lambda 5M
- Lambda 5E
- Lambda 21B

1.2.1 Lambda 5M

It is used for the bus control system.

Its structure and interface definition are as follows:



1. Indicator light
   - Power: An indicator light for the power supply
   - System: An indicator light for the system
2. 24V power interface

   It has anti-reverse connection protection.
3. Interface for switching high-low level

   It is used to switch between the high level and low one.
4. General input ports

   They include X00 ~ X27.
5. Interface for extended terminal board

   A high-speed 485 interface. It is used for cascade and adopts serial communication. The baud rate is 10Mbps.

6. General output ports

   They include Y00 ~ Y07 and 5 general ports.

7. Analog output port

   - Analog voltage: 0 ~ 10V
   - Precision: 0.2V

8. Handwheel interface

   It is used to connect to a handwheel (supporting the 6-axis handwheel at most).

9. M-II bus interface

   It is used to connect to a M-II bus drive. And the interpolation period of pulses is 1ms.

10. Moving-axis interface

    It is used to connect to a servo drive.

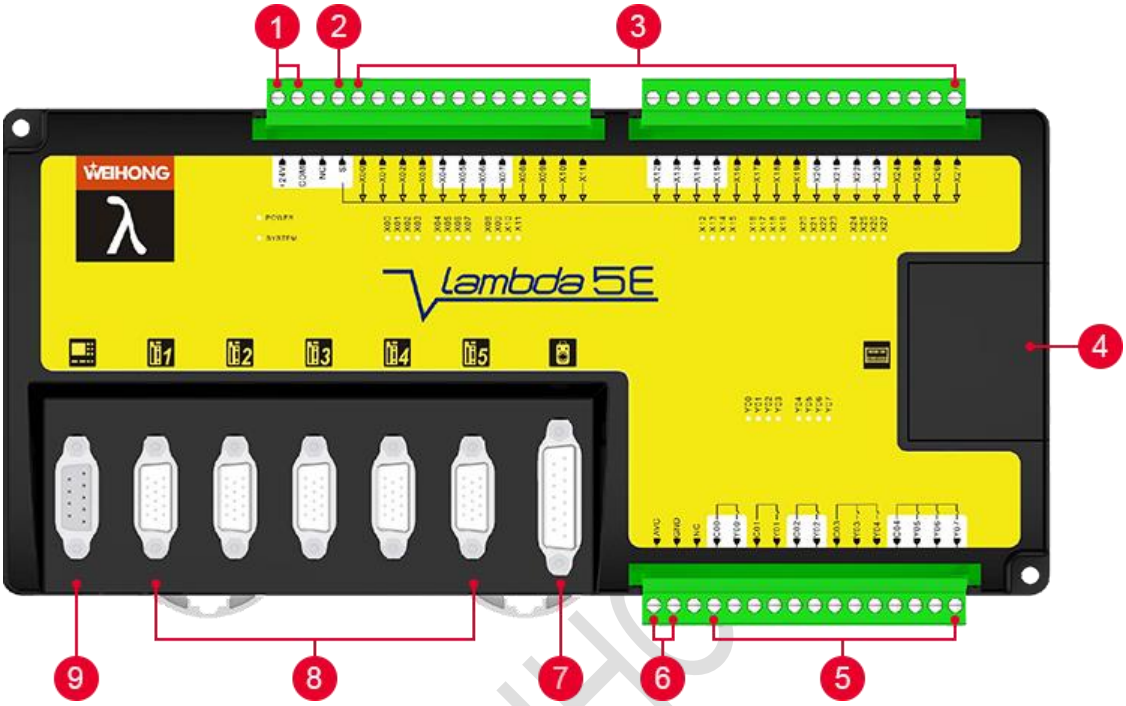
11. Host interface

    It is used to connect to NC65C, NC60A, etc.

1.2.2 Lambda 5E

It is used for the non-bus control system.

Its structure and interface definition are as follows:



1. Interface for a 24V power supply

   It has anti-reverse connection protection.

2. Interface for switching high-low level

   It is used to switch between the high level and low one.

3. General input ports

   They include X00 ~ X27.

4. Interface for extended terminal board

   A high-speed 485 interface. It is used for cascade and adopts serial communication. The baud rate is 10Mbps.

5. General Output ports

   They include Y00 ~ Y07 and 5 general ports.

6. Analog output port
   - Analog voltage: 0V ~ 10V
   - Precision: 0.2V
7. Handwheel interface

   It is used to connect to a handwheel (supporting the 6-axis handwheel at most).
8. Moving-axis interface

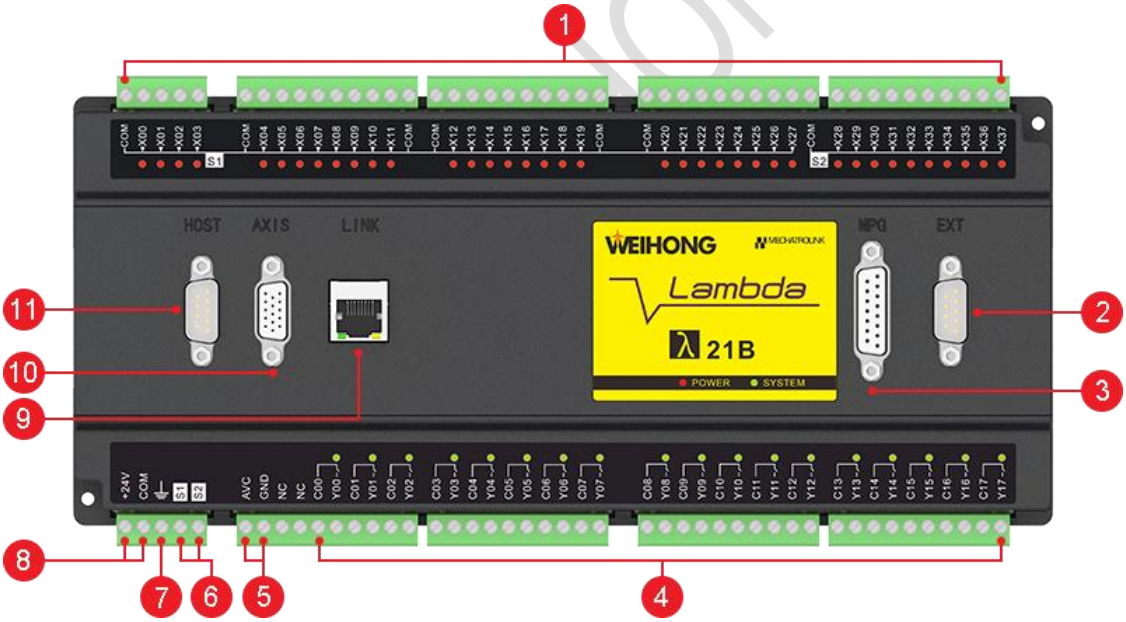
   It is used to connect to a servo drive.
9. Host interface

   It is used to connect to NC65C, PM95A, the integrated CNC system, etc.

1.2.3 Lambda 21B

It is used for the bus control system.

Its structure and interface definition are as follows:



1. General input ports

   They include X00 ~ X37.
2. Extended interface

   A DB9 interface. And it is called EXT.

3. Handwheel interface

   It is called MPG.

4. General output ports

   They include C00 ~ C17 / Y00 ~ Y17.

5. Output interface for analog voltages

   - AVC: An output interface for analog voltages
   - GND: The ground for analogy voltages

6. Configurable ports for I/O valid level

   NPN / PNP is configurable (S -> 24V / S -> com):

   - S1: Configure the valid level of X00 ~ X27.
   - S2: Configure the valid level of X28 ~ X37.

7. Ground interface

   It is used to connect to the ground.

8. Power supply interfaces

   - +24V: Positive input for 24V DC
   - COM: The ground for 24 DC

9. Mechatrolink-II communication interface

   A RJ45 interface. And it is called LINK. The axis quantity is at most 16.

10. Pulse-axis communication interface

    A DB15 interface. And it is called AXIS. It supports an incremental encoder.

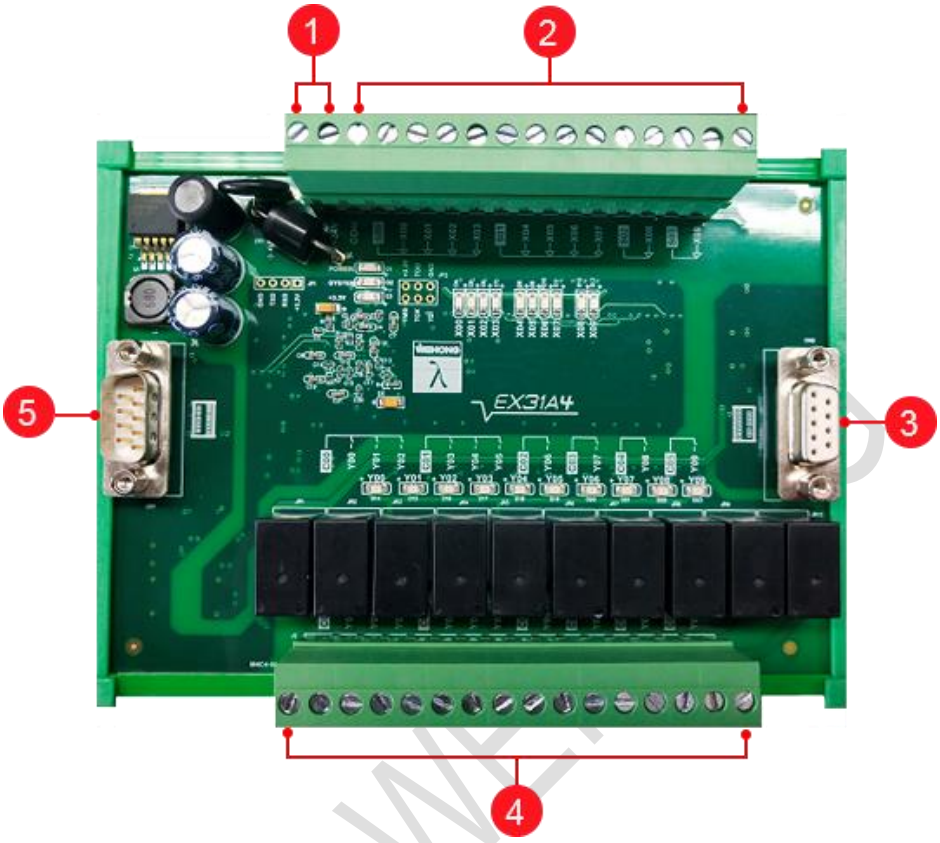
11. Host communication interface

    A DB6 interface. And it is called HOST. It follows the Phoenix bus protocol.

1.3 EX31A

EX31A, the extended board of the Lambda terminal board, adopts the mounting structure of module rack, for good protection and easy installation.

Its structure and interface definition are as follows:



1. Power supply input

   24V / 0.5A input (+24V, COM).

2. Input ports

   It supports a 10-path input and NPN / PNP can be configurable (S -> 24V / S -> COM)

   S ports include the following:

   - S0: X00 ~ X03
   - S1: X04 ~ X07
   - S2: X08
   - S3: X09

3. Communication interface

   It is used for the cascade port communication and follows the Phoenix bus protocol.

4. Output ports

   They support the 10-path relay contact output.

5. Communication interface

   Uplink port communication. It follows Phoenix bus protocol.

1.4 Software interface

NcStudio V10 Waterjet Cutting CNC System has the following interfaces:

- NcStudio interface for machining
- NcEditor interface for editing toolpath

1.4.1 NcStudio Interface

The main interface of NcStudio is as follows:

1. Title bar
2. Menu bar
3. CNC status bar
4. Machine tool control bar
5. Operational button bar
6. Function windows

1.4.1.1 Menu Bar

It includes the following menus:

- **File**: It is used to load / unload program files, generate an installation package, restart / shut down the system, etc.
- **Work Mode**: It is used to switch among **Auto** mode and **Manual** modes (including jog, handwheel and step modes).
- **Operate**: It is used to execute machining commands, such as executing single block, setting the workpiece origin, executing simulation, rotating / mirroring toolpath, etc.
- **Machine Tool**: It is used to execute operations related to machine tools, such as customizing valve ON / OFF order, checking time statistics for high pressure valve and oil pump, managing wearing parts, etc.
- **Window**: It is used to switch among track, machining information, position diagnosis, hardware port, log and setup interfaces, show or hide **NcEditor** interface, open **PLC** or **Manual Code** dialog box, etc.
- **Language**: It is used to switch a system language.
- **Help**: It is used to check the system information, register the software, set the setup interface, open **NcGateway** interface, use the velocity calculator, etc.
- **Params**: It is used to check and set common parameters about waterjet cutting.

1.4.1.2 CNC Status Bar

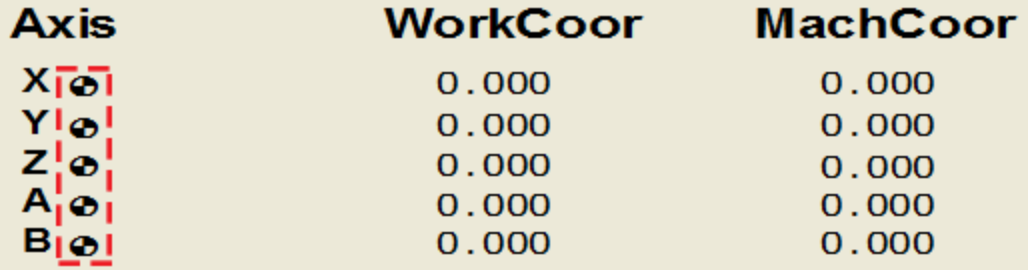
It includes the following:

- Current operation mode: Auto, manual, etc.
- Current system status: Running, idle, E-stop, etc.
- Current running status: Normal running, normal or abnormal stop, etc.
- Current workpiece coordinate system (WCS): G54 ~ G59
- Machining time
- Current system configuration: Bus type, standard A type and double-Y A type.
- System prompts, alarms, etc.

1.4.1.3 Machine Tool Control Bar

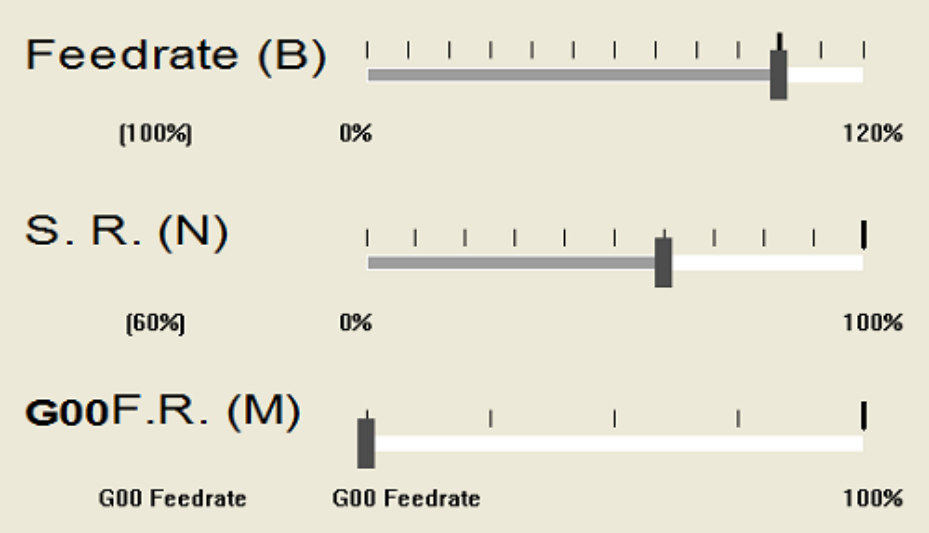
It includes the following:

- Axis coordinate area: It shows the current machine coordinate and workpiece coordinate of each axis:

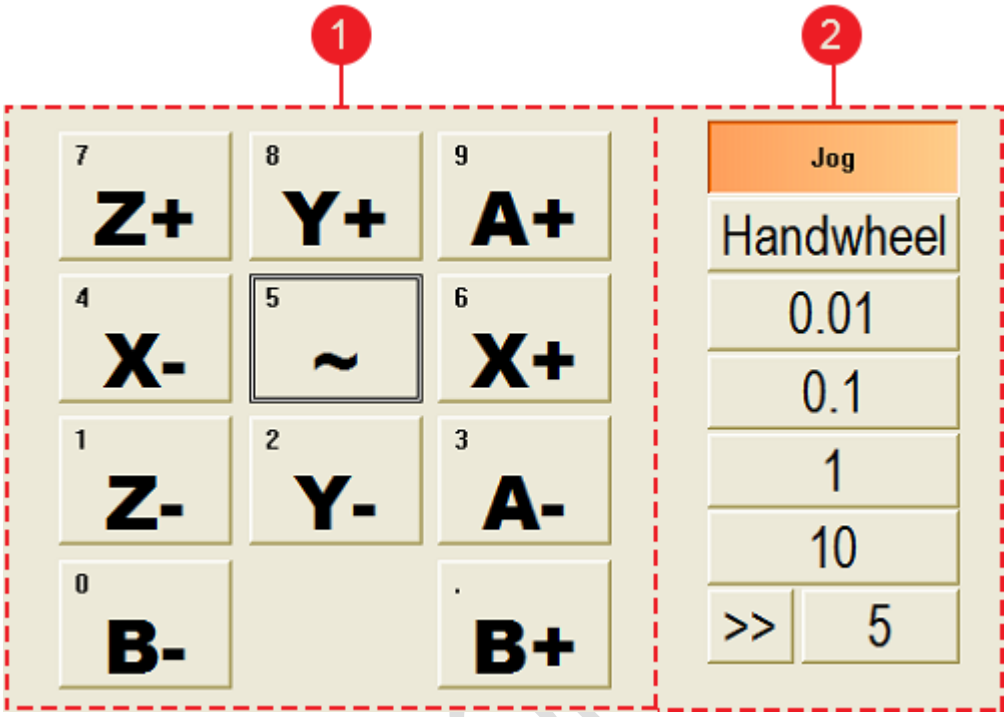


  After returning to the machine origin, the sign appears after each axis.

- Feed speed area: It shows the actual feedrate and set value of linear axes and rotary axes.
- Tool area: It shows the current tool and material thickness.
- File area: It shows the name of current program file.
- Operation mode area: The display differs in the operation mode:
  - **Auto** mode: It is used to adjust feedrate, abrasive valve and G00 rate during automatic machining:



– **Manual** mode:



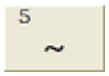
1. Axis direction buttons: It is used to move each axis towards positive / negative direction.

   Note: Please do not click an axis direction button too frequently because the system needs a certain time to execute the command.

2. Mode buttons: It is used to switch among the following modes:

   – Jog: Click **Jog** and do one of the following:

   - Click an axis direction button. The machine tool keeps running until you release the button.
   - Click several axis direction buttons. The clicked axes move at the same time at jog speed until you release buttons.

     This operation is mainly used to move X-axis and Y-axis at the same time.
   - Click 

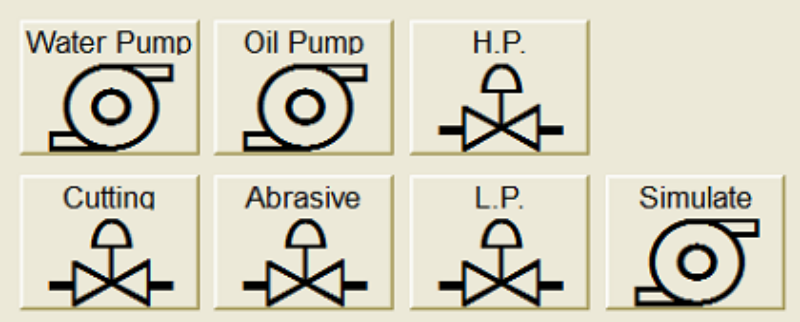
and an axis direction button. The machine tool moves at rapid jog speed.

   – Handwheel: Click **Handwheel**, select the axis direction and handwheel override and rotate the handwheel for a certain degrees. The machine tool is controlled by the handwheel and moves towards the selected direction.

- Step: Select a step size among 0.01, 0.1, 1,10 or click  to customize the step size (default value: 5mm), click an axis direction button and then release it. The machine tool moves the selected step size.

  Note: The step size should not be too large, otherwise, the machine tool may be damaged due to misoperations.

- Port control area: It is used to turn on / off the following common ports:



Note: The rotating workbench is used to load / unload relatively large or heavy sheets by its rotation.

1.4.1.4 Operational Button bar

Operational buttons differ in the operation mode. They are used to execute related operations.

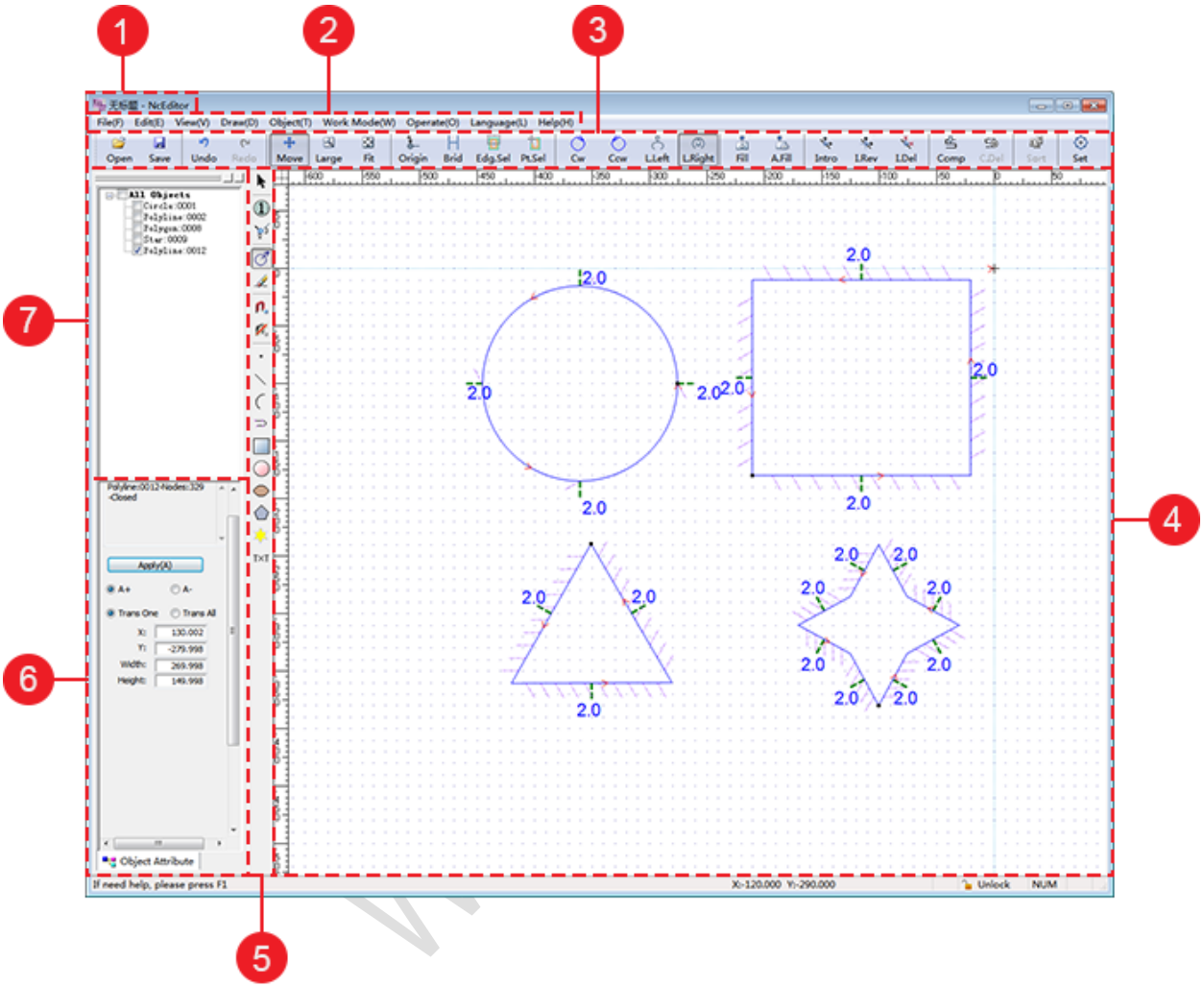
1.4.1.5 Function Windows

They include the following windows:

- **POS**: It is used to show the following:
  - Toolpath in real-time during machining or simulation.
  - Contents of the program file in **Auto** mode.
- **INFO**: It is used to show machining records, calculate the machining cost, etc.
- **POS DIAG**: It is used to show feedback coordinates, sent pulse and feedback pulse and set the datum for each axis.
- **IO PORT**: It is used to show the status of each IO port and communication status between the software and hardware.
- **LOG**: It is used to show different types of logs that can help troubleshoot.
- **SETTING**: It is used to set cutting parameters in **Auto** or **Manual** mode.

1.4.2 NcEditor Interface

The main interface of **NcEditor** is as follows:



1. Title bar
2. Menu bar
3. Tool bar
4. Drawing area
5. Drawing auxiliary bar
6. Object attribute area
7. Object list window

1.4.2.1 Title Bar

It is used to show the name of current program file.

1.4.2.2 Menu Bar

It includes the following menus:

- **File**: It is used to create / open / import / save a program file, generate an installation package, restart / shut down the software, etc.
- **Edit**: It is used to undo / redo the operation, select all objects, insert a file, etc.
- **View**: It is used to customize the displayed information in the main interface, adjust the view, return to **NcStudio** interface, etc.
- **Draw**: It is used to select the object shape, measure distance, set the workpiece origin, lock the view, etc.
- **Object**: It is used to set machining technics, such as chamfer, array, micro joint, bridge, etc.
- **Work Mode**: It is used to switch between **Auto** mode and **Manual** modes (including jog, handwheel and step modes).
- **Operate**: It is used to execute machining commands, such as executing single block, setting the workpiece origin, executing simulation and so on.
- **Language**: It is used to switch a system language.
- **Help**: It is used to check the system information, register the software and set the setup interface.

1.4.2.3 Tool Bar

It includes the following:

- 

: It is used to open a program file.
- 

: It is used to save a program file.
- 

: It is used to cancel the last operation.
- 

: It is used to recover canceled operations.
- 

: It is used to translate the view.
- 

: It is used to zoom in / out the view.

- 

: It is used to make the view fit to the drawing window.
- 

: It is used to set the workpiece origin.
- 

: It is used to bridge two adjacent objects.
- 

: It is used to set the edge for the tilt angle.
- 

: It is used to set the point for the tilt angle.
- 

: It is used to set the machining direction as clockwise for the target object.
- 

: It is used to set the machining direction as counterclockwise for the target object.
- 

: It is used to set the groove direction on the left of the machining direction for unclosed objects.
- 

: It is used to set the groove direction on the right of the machining direction for unclosed objects.
- 

: It is used to change the filling attribute for closed objects.
- 

: It is used to automatically set the filling attribute for closed objects.
- 

: It is used to set a lead line.
- 

: It is used to reverse the lead line.
- 

: It is used to delete the lead line.
- 

: It is used to set tool compensation.
- 

: It is used to delete the set tool compensation.

- 

: It is used to sort the machining order.
- 

: It is used to set the mode of vertical plunge, tilt angle and cutting speed.

1.4.2.4 Drawing Area

It is used to draw the machined objects.

1.4.2.5 Drawing Auxiliary Bar

It includes the following:

-  : It is used to select the object.
-  : It is used to set the machining order.
-  : It is used to show the machining order.
-  : It is used to show the groove direction.
-  : It is used to clear the machining track.
-  : It is used to set catch options.
-  : It is used to disable catching function.
-  : It is used to draw a dot.
-  : It is used to draw a line.
-  : It is used to draw an arc.
-  : It is used to draw a polyline.
-  : It is used to draw a rectangle.
-  : It is used to draw a circle.
-  : It is used to draw an ellipse.
-  : It is used to draw a polygon.
-  : It is used to draw a start.
-  : It is used to add text.

1.4.2.6 Object Attribute Area

It is used to set the following attributes for the selected object:

- For one object
  - X-axis and Y-axis coordinates
  - Width and height / diameter
- For all objects
  - Width
  - Height
  - Rotating angle

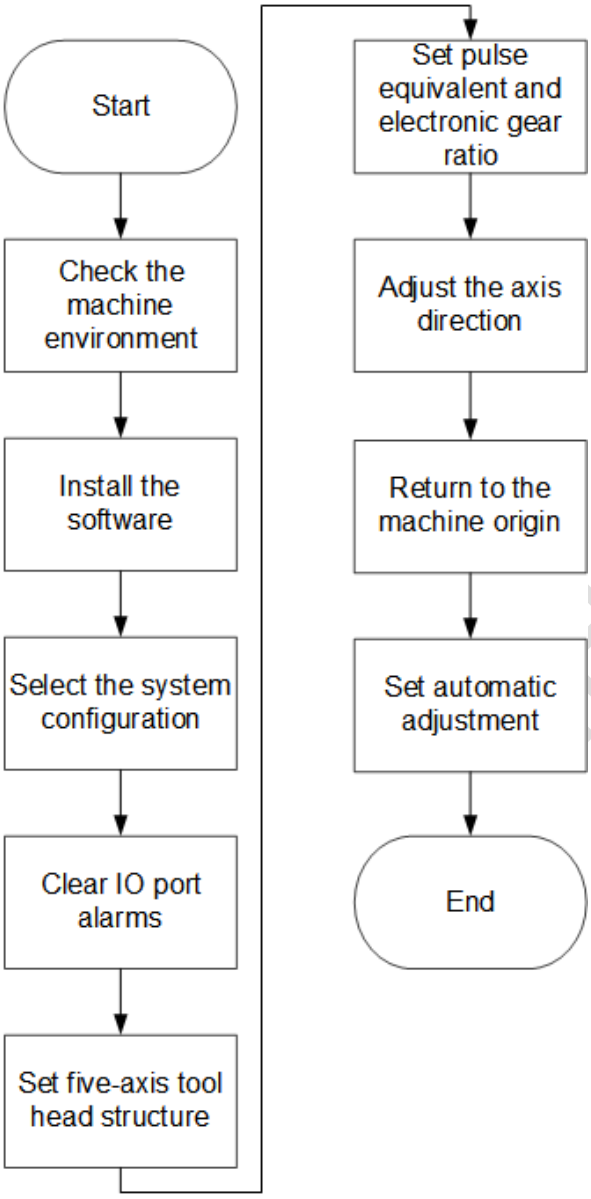
1.4.2.7 Object List Window

It used to show all objects in the drawing area according to the drawing order.

2 Basic Commissioning

Through this section, you can quick know the basic commissioning of **NcStudio V10 Waterjet Cutting Control System**.

The commissioning procedure is as follows:



During commissioning, note the following:

- If the manufacturer password is required during commissioning, please contact us.
- If parameters whose effecting time is **Reboot** is modified, please restart the software to validate the modification.

2.1 Check the Machine Environment

This operation is used to check the environment of the machine tool, so as to ensure the commissioning goes on well.

It includes the following:

- Check the mechanical structure
- Check the electronic wiring
- Check auxiliary circuits

2.1.1 Check the Mechanical Structure

This operation is used to check if the mechanical structure of machine tool is normal.

Check the mechanical structure from the following aspects:

- Installation of the moving servo axis and mechanism connection.
- Connection of the coupling, decelerator and other parts without a relative slide.
- Gear occlusion and something irrelevant on the screw rail.
- Anti-collision safety device.

2.1.2 Check Electrical Wiring

This operation is used to check if the electrical wiring of the machine tool is normal.

Check electrical wiring from the following aspects:

- Ground connection of the machine tool.
- Abnormal phenomena in the electrical box, such as short circuit and open circuit.
- Power supply for the host, Lambda terminal board, and EX serial extended terminal board.
- Communication among the host, Lambda terminal board, and EX serial extended terminal board.
- NO / NC connection of each inductive switch.
- Signal of E-stop button.
- Connection between the Lambda terminal board and servo drive.

2.1.3 Check Auxiliary Circuits

This operation is check if auxiliary circuits of the machine tool are normal.

Auxiliary circuits include the following:

- Hydraulic circuit
- Circuit of cooling water
- Circuit for abrasive sand supplying
- Circuit for gas supplying

2.1.3.1 Check the Hydraulic Circuit

Check the hydraulic circuit from the following aspects:

- Oil junctions.
- Temperature of hydraulic liquid.

  If it is too high, check the circuit of cooling water.
- Oil pressure.
  - Too high: Check if the oil channel is clogged.
  - Too low or unstable: Check if there is an oil leakage.

2.1.3.2 Check the Circuit of Cooling Water

Check the circuit of cooling water from the following aspects:

- Connection, such as reverse connection and other problems.
- Block.

2.1.3.3 Check the Circuit for Abrasive Sand Supplying

Check the circuit for abrasive sand supplying from the following aspects:

- Abrasive sand in the abrasive-sand tank.
- Flow of abrasive sand.

  If it is abnormal, check if the jewel nozzle and grit-sand circuit is clogged.

2.1.3.4 Check the Circuit for Gas Supplying

Check the circuit for gas supplying from the following aspects:

- Connection.
- Air pressure for abrasive sand supplying, water valve, height measuring, blowing and so on.

2.2 Install the Software

Before installing the software, ensure the following:

- The machine tool has been powered and can be normally used.
- The installation package of **NcStudio V10 Waterjet Cutting Control System** has been copied to the host.

To install the software, do the following:

1. Double click the installation package. The dialog box for selecting the language pops up.
2. Select the target language according to your need and click **Yes**. The dialog box about migrating parameters pops up.
3. Choose whether to migrate parameters:
   - To migrate related parameter settings of the previous software in the host to the new software, click **Yes**.
   - After installing the software, previous parameter settings remain.
   - **Note:** It is recommended to migrate parameters only when the versions of the previous software and the new one are the same or similar.
   - To reset parameters instead of using the previous parameter settings, click **No**.
   - After installing the software, values of all parameters are default.

2.3 Select the System Configuration

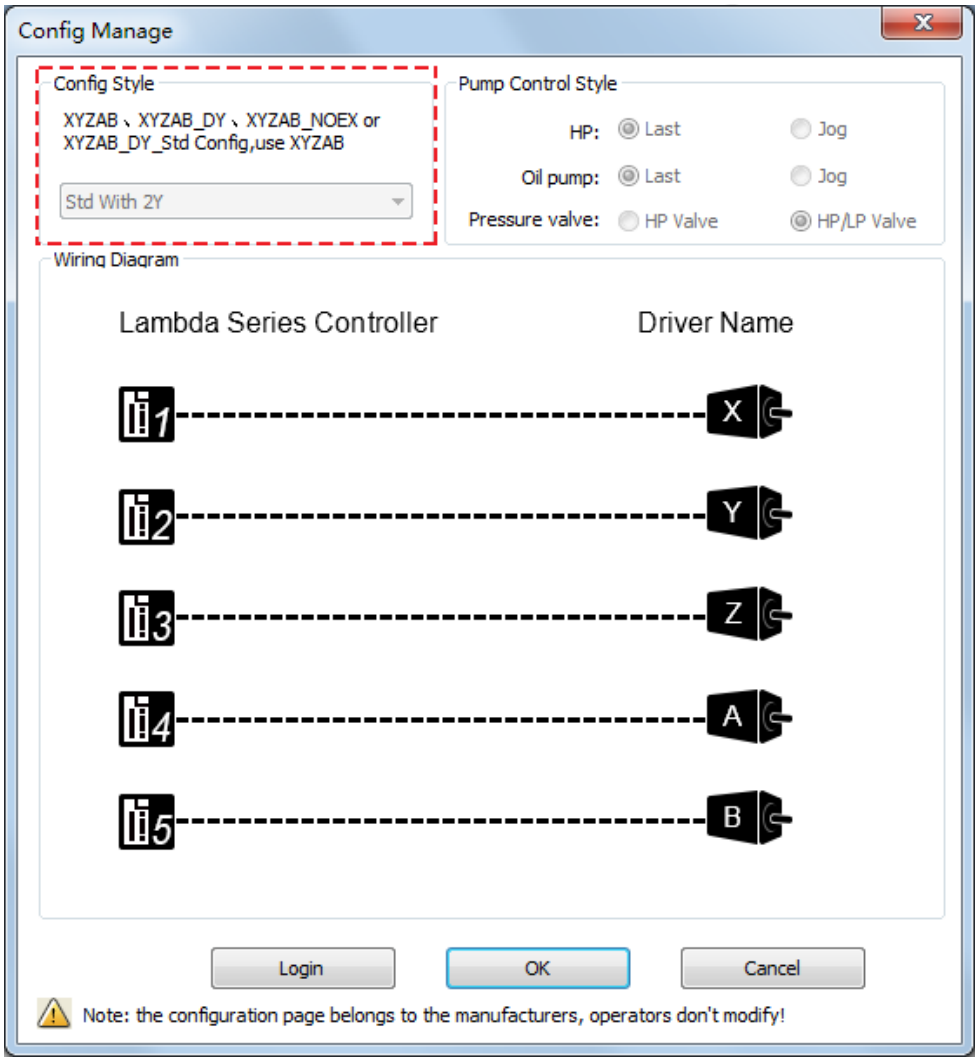
This operation is used to select the system configuration according to actual hardwares for the first installation.

The system currently supports the following configurations:

- Single Y-axis / double Y axes: It can be judged from the mechanical structure.
- Bus control / non-bus control: It can be judged from the control system.
- Standard / others: It can be judged from the actual wiring of terminal board and layout of IO ports.

To select the system configuration, do the following:

1. Click **Machine Tool** → **Config Manage**. **Config Manage** dialog box pops up:



2. To activate the selection box in **Config Style** area, click **Login** and enter manufacture password.

3. According to actual conditions, select the configuration in **Config Style** area and click **OK**.

The system automatically restarts to make the selected configuration effective.

2.4 Clear IO Port Alarms

This operation is used to clear IO port alarm before machining, so as to establish a good communication between the control system and servo drive.

Common alarms include the following:

- Alarm of terminal board disconnection
- E-stop alarm
- Limit alarm
- Drive alarm
- Alarm of port polarity error

After you fail to clear these alarms, please contact us.

2.4.1 Clear the Alarm of Terminal Board Disconnection

To clear the alarm of terminal board disconnection, check the following:

- Wiring related to the terminal board.

  If it is not correct and firm, do wiring again.

- Port polarities in **IO Port** window.

  If they are not correct, modify the polarity of the corresponding port and restart the software. See Check Port Information for details.

- Terminal board.

  If it is damaged, replace it.

2.4.2 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 **E-stop** port in **IO Port** window and restart the software.

2.4.3 Clear the Limit Alarm

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

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

2.4.4 Clear the Drive Alarm

To clear the drive alarm, check the following:

- Wiring of the drive.

  If it is not connected correctly and securely, do wiring again.

- Polarity of **Servo Alarm** port.

  If it is not correct, modify the port polarity of the corresponding port and restart the software.

- Parameter **Driver Station Address** (only for bus configuration).

  If it does not match with the actual station address of the servo drive, modify it according to the actual station address and restart the software.

For the bus configuration, alternatively, you can clear common drive alarms by clicking **Machine Tool → Clear Servo Alarm**.

2.4.5 Clear the Alarm of Port Polarity Error

If the port polarity does not match with the switch of machine tool, an alarm will occur.

To clear the alarm of port polarity error, modify the polarity of corresponding ports in **IO Port** window and restart the software.

2.5 Set the Structure of Five-axis Tool Head

Through RTCP algorithm, this operation helps to solve additional motions of the tool head during the rotary axis rotates, resulting from the fact that the control point of the CNC system may be not on the tool head due to the installation. As a result, the CNC system will automatically correct the control point and the tool head will move along the toolpath specified by commands.

At present, both five-axis AB and five-axis AC software support to customize the structure of tool head.

2.5.1 Structures of Tool Head

The description about structures of tool head is as follows:

- Main rotary axis: Its rotation can change the position of the other rotary axis.

  For the AC head tool, the main rotary axis is C-axis; for the AB head tool, it is B-axis.

- Auxiliary rotary axis: Its rotation cannot change the position of the other rotary axis.

  The auxiliary rotary axis is A-axis.

- Control point: The point where the center lines of two auxiliary rotary axes intersect.

- Cutting point: The optimal point where the spouted water and workpiece surface intersect.

  In general, it is 3mm ~ 5mm below the abrasive tube.

- Vector of main rotary axis: The direction vector of the main rotary axis in this coordinate system, taking the cutting point as the origin of space coordinate system.

  In general, one of X-axis, Y-axis and Z-axis coordinates is set to 1. E.g. (0,0,1)

- Vector of auxiliary rotary axis: The direction vector of the auxiliary rotary axis in this coordinate system, taking the cutting point as the origin of space coordinate system.

  In general, one of X-axis, Y-axis and Z-axis coordinates is set to 1. E.g. (1,0,0)

- Center of main rotary axis: The coordinate of an arbitrary point on the main rotary axis in this coordinate system, taking the cutting point as the origin of space coordinate system.

- Center of auxiliary rotary axis: The coordinate of an arbitrary point on the auxiliary rotary axis in this coordinate system, taking the cutting point as the origin of space coordinate system.

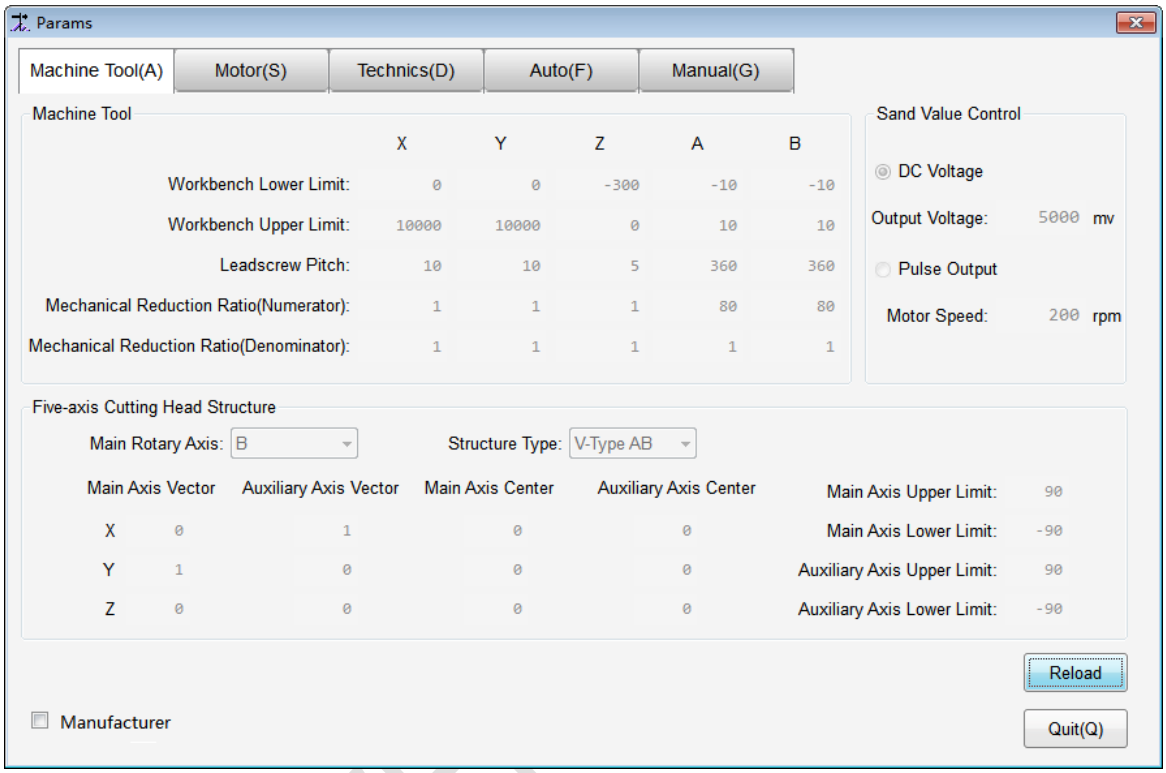
- Vector of tool rod: The direction vector of spouted water when each rotary axis rotates to 0°.

  Default: (0,0,-1)

2.5.2 Set Five-axis AB Tool Head

To set five-axis AB tool head, do the following:

1. Click **Params**. **Params** dialog box pops up:



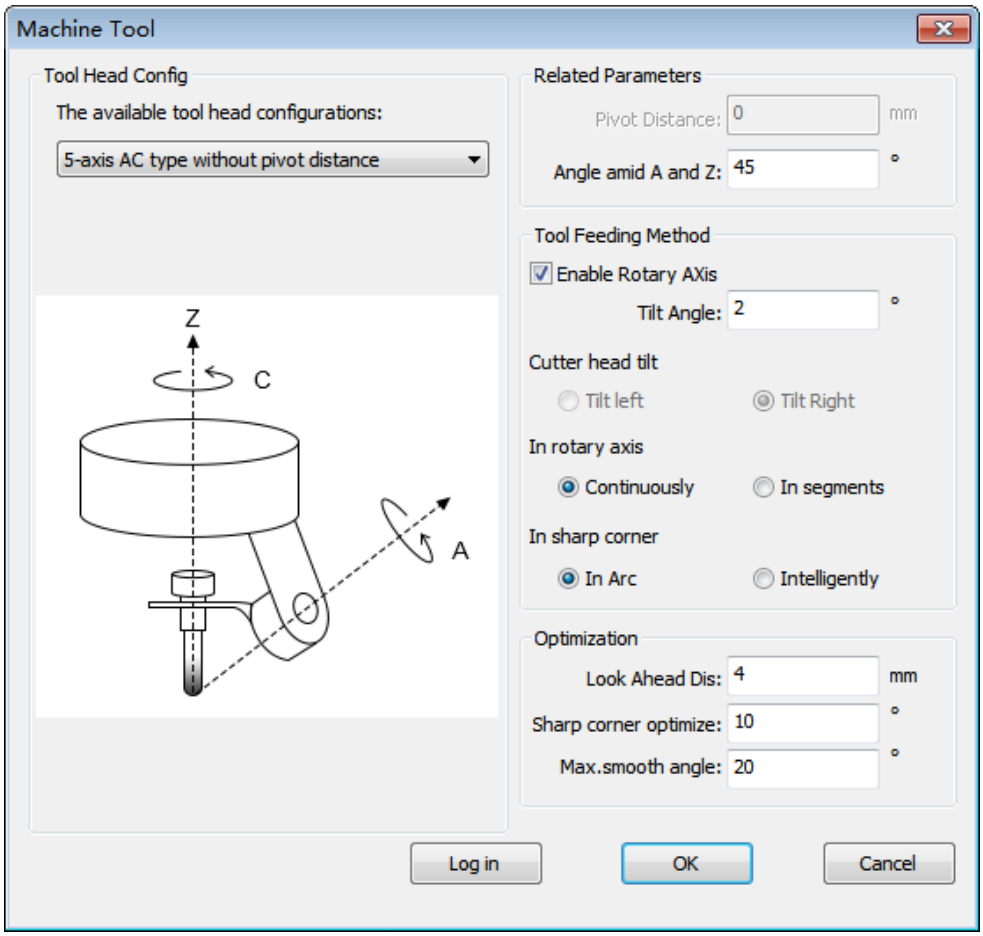
2. To activate input boxes, check **Manufacturer** and input the manufacturer password.
3. Select B-axis as the main rotary axis.
4. Select **Tool head structure** and do related operations:
   - For option **90°AB / 45°AB / V-type AB**, set **Waterjet tool length** in **Technics** interface.
   - For option **Customize**, customize all tool parameters.
   - About parameter details, see Structures of Tool Head.

To validate parameter settings, reload a program file.

2.5.3 Set Five-axis AC Tool Head

To set five-axis AC tool head, do the following:

1. Click **Machine Tool** → **Machine Tool**. **Machine Tool** dialog box pops up:

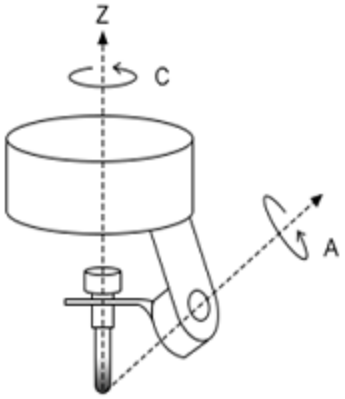


2. Click **Log in**, and input the manufacturer password.

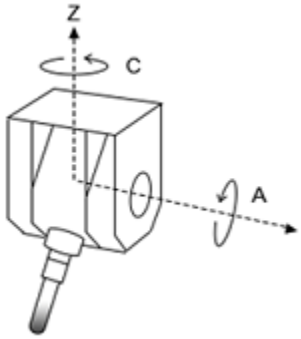
3. Select the structure of tool head and do related operations:

   – For option **5-axis AC type without pivot distance**, set **Angle amid A and Z** according to the actual situation.

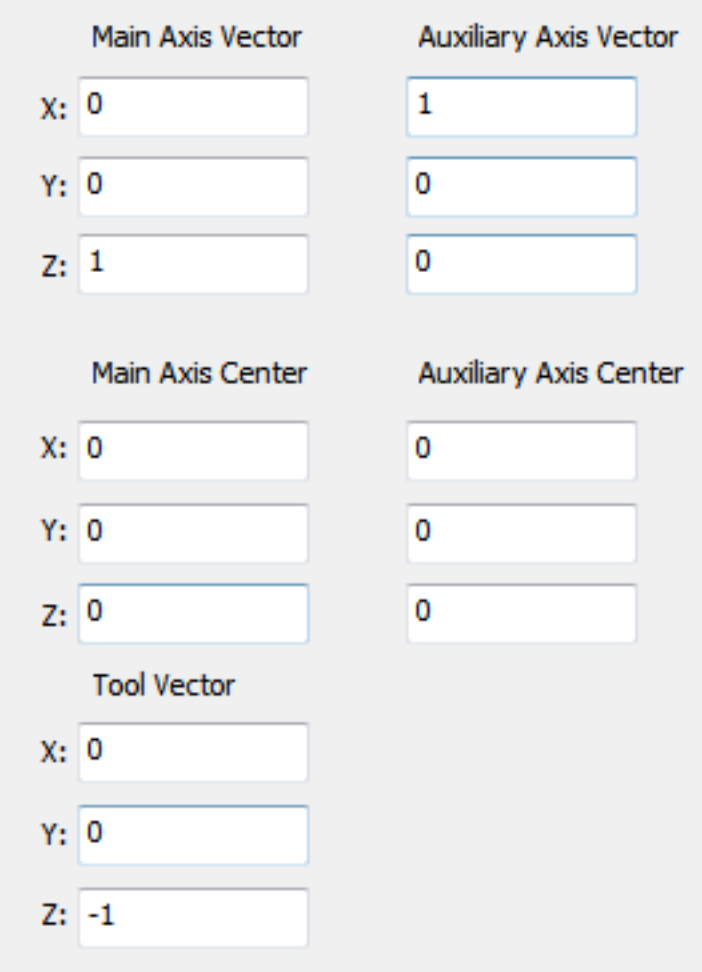
     In general, the angle between A-axis and C-axis is 45°.



   – For option **5-axis AC type with pivot distance**, set **Y offset of A-axis**, **X offset of C-axis**, **Y offset of C-axis** and **Pivot Distance**.



– For option **5-axis AC Custom Tool Head**, customize all tool parameters:



About parameter details, see Structures of Tool Head.

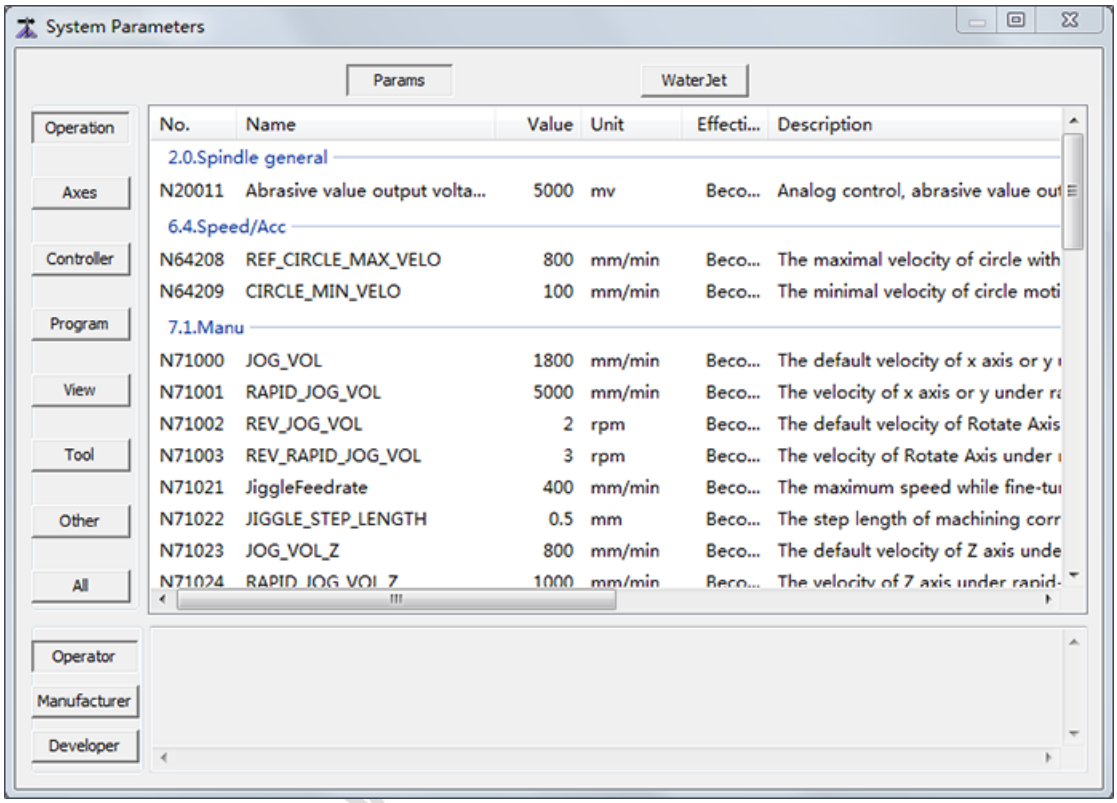
To validate parameter settings, reload the program file.

2.6 Set Commissioning Parameters

This operation is used to set related parameters for commissioning, so as to avoid damages to the machine tool caused by its motion.

To set commissioning parameters, do the following:

1. Click **Machine Tool** → **System Parameters**. **System Parameters** dialog box pops up:



2. Click **Params** → **Axes** → **Manufacturer**, input the manufacturer password and find the following parameters:

   – Bus configuration:

   - Screw Pitch: Its setting should match with the actual situation.
   - Mechanical Reducer Ratio: Its setting should match with the actual situation.
   - Encoder Digit: Its setting should match with the actual situation.
   - Electronic Gear Ratio: Its setting should match with the actual situation. Default: **1:1**.

- Non-bus configuration

  - Pulse Equivalent: It refers to the moving distance of screw or rotation degree of rotary axis per pulse sent by the system, the minimum available distance controlled by the system as well.
  - Electronic Gear Ratio: It can be calculated in term of the value of parameter **Pulse Equivalent**.

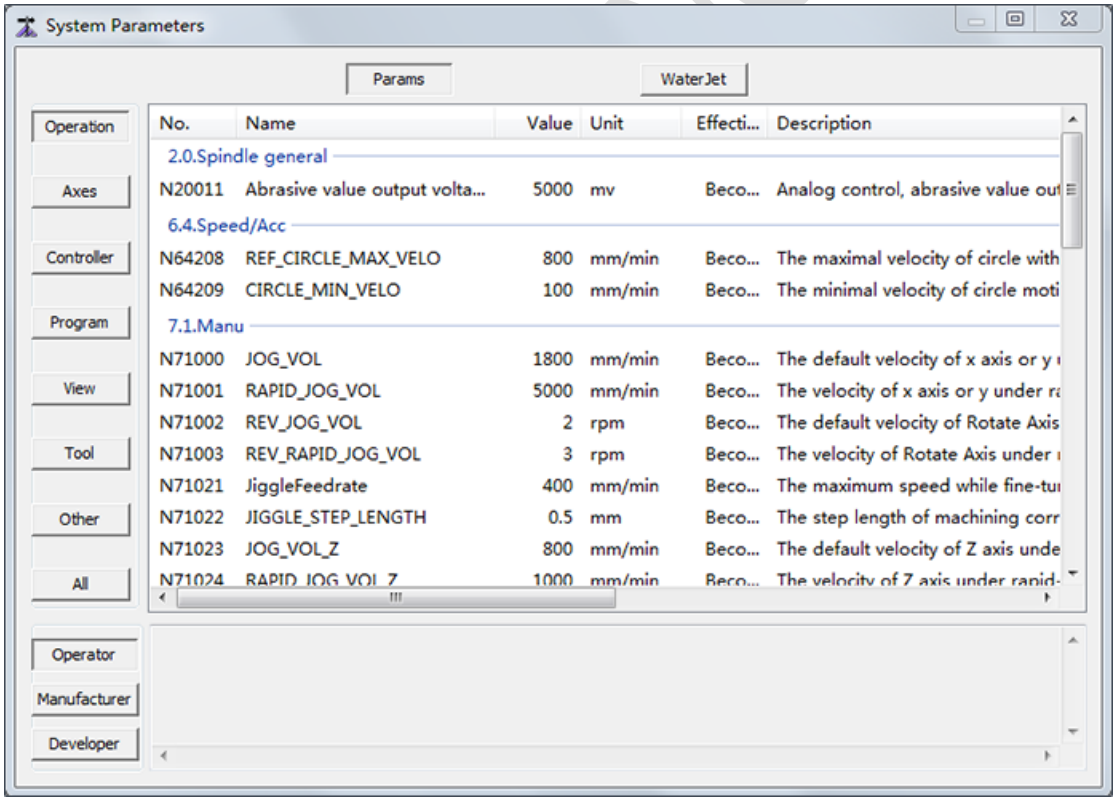
3. Double click the target parameter, input its value and click **OK**.

2.7 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**, and adjust it if not, 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. Click **Machine Tool** → **System Parameters**. **System Parameters** dialog box pops up:



2. Click **Params** → **Axes** → **Manufacturer**, input the manufacturer password and check the current value of parameter **Axis Direction (X)**:
   - 1: Positive direction
   - -1: Negative direction
3. Judge the positive direction of X-axis according to **Right Hand Rule**.
4. In **Manual** mode, move X-axis in positive direction.
5. Observe if the actual positive direction of X-axis is the same with that stipulated by **Right Hand Rule**:
   - If they are the same, the parameter **Axis Direction (X)** is set correctly.
   - If they are opposite, modify the parameter **Axis Direction (X)** to the opposite value.

2.8 Execute Returning to the Machine Origin / Set the Datum

This operation can help the machine coordinate system of **NcStudio V10 Waterjet Cutting Control System** match with that of the machine tool for a preciser machining.

Before returning to the machine origin, ensure all alarms have been cleared.

According to the encoder type, it can be divided into the following:

- For an absolute encoder, set the datum.
- For an incremental encoder, execute returning to the machine origin.

2.8.1 Set the Datum

This operation is used to directly set the zero for the absolute encoder.

Except the following conditions, this operation is unnecessary because the system will automatically read datum of each axis:

- It is the first time to install the software.
- The software is updated.
- The absolute encoder runs out of battery.
- The datum file has been damaged.

Before setting the datum, ensure the following:

- Hardwares have been connected well.
- The axis direction is correct.

For double Y axes configuration, before setting Y-axis datum, to adjust parallelism of double Y axes, make the fine adjustment for Y1-axis and Y2-axis in **POS DIAG** window and make a trial cutting.

Taking X-axis as an example, to set the datum, do the following:

1. To enter **POS DIAG** window, click **POS DIAG**.
2. To activate datum setting buttons, click **RefSet** or press **F3**, and input the manufacturer password.
3. To open **Table Travel Set** dialog box, click **X Datum** → **Yes**.
4. To set the upper limit of the workbench travel, move X-axis in the positive direction to a reasonable position, and click **Upper Limit**.
5. To set the lower limit of the workbench travel, move X-axis in the negative direction to a reasonable position, and click **Lower Limit**.
6. Click **OK**.

The system starts to read datum data.

Repeat the steps above to set datums for other axes, and restart the software to validate settings.

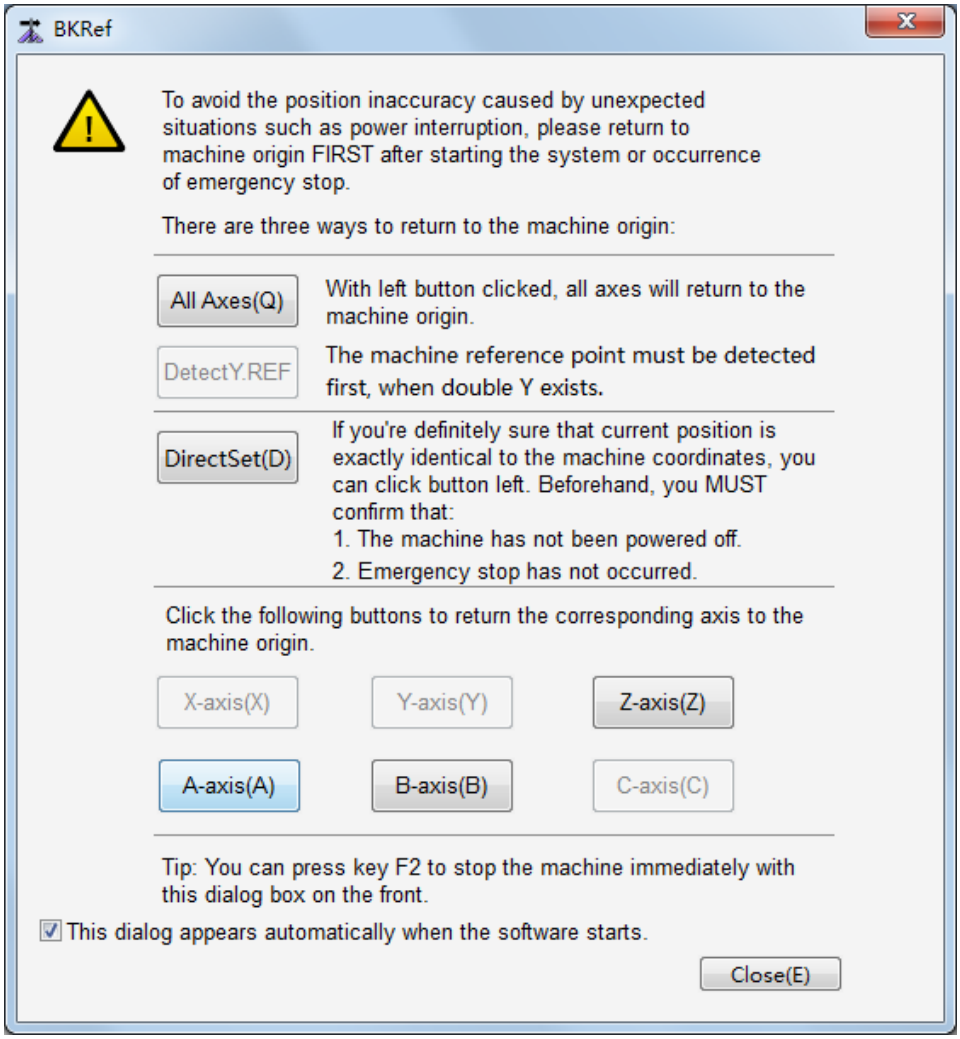
To directly use the set datums after updating the software, export the set datums as an `INI` file by clicking **Export Datum**. After updating the software, insert the USB flash disk with the `INI` file and click **Import Datum**.

2.8.2 Execute Returning to the Machine Origin

With an incremental encoder, every time you restart the software, you need to return to the machine origin before machining. But if no abnormal phenomenon occurs during this machining, you do not need to execute this operation before the next machining.

To execute returning to the machine origin, do the following:

1. To open **BKRef** dialog box, do one of the following:
   - Click **Operate → Back to Reference Point**.
   - In **Manual** mode, click **REFER** in **POS** window.



You can check **This dialog box appears automatically when the software starts** at lower left corner. **BKRef** dialog box will automatically pop up when the software starts next time.

2. Do one of the following:

   - To return Z-axis to the machine origin first and return the other axes, click **All Axes**.
   - If you ensure the following, click **Direct Set** to directly set the current position as the machine origin:
     - The current position is the same with the machine coordinate.
     - The machine has never been turned off.
     - E-stop has never occurred.
   - To return a single axis to the machine origin, click **X-axis** / **Y-axis** / **Z-axis** / **A-axis** / **B-axis** / **C-axis**.

     For safety, it is recommended to return Z-axis first.

After executing returning to the machine origin, the sign appears after the corresponding axis in coordinate display area.

2.9 Set Automatic Adjustment

Taking the WISE drive as an example, according to the current machine tool status, this operation is used to adjust the gain of servo drive and related parameters, so as to improve responsiveness.

It can be executed only for one axis among X-axis, Y-axis and Z-axis at a time.

Before setting automatic adjustment, ensure connection and communication between the servo drive and control system is correct and normal.

To set automatic adjustment, do the following:

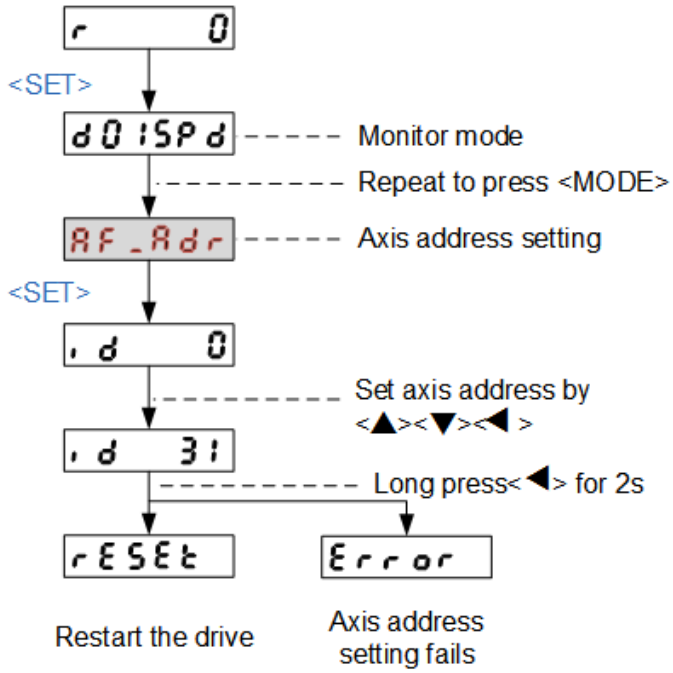
1. Set the station address for the drive.
2. Set drive parameters.
3. Execute the automatic adjustment: It is applicable to the bus control system.

2.9.1 Set the Station Address for the Drive

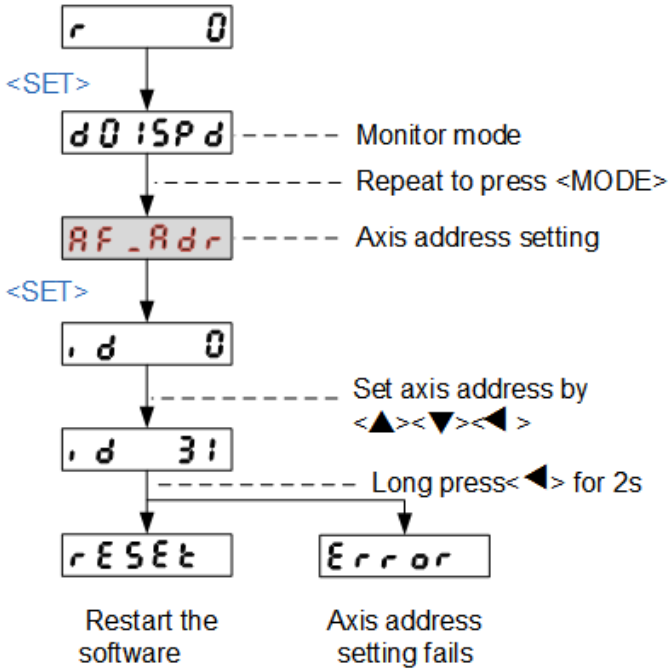
This operation is used to set the station address by setting the drive parameter on the operation panel.

To set the station address for the drive, do the following:

1. To select the position control mode, set the drive parameter **Pr001 Control mode setup** to **1** according to the following procedure:



2. According to the following procedure, set numbers of the station address:



It is recommended to set numbers of the station address in order (E.g, X-axis: 1; Y-axis: 2; Z-axis: 3).

Note: Station address **0** means disabling the communication function.

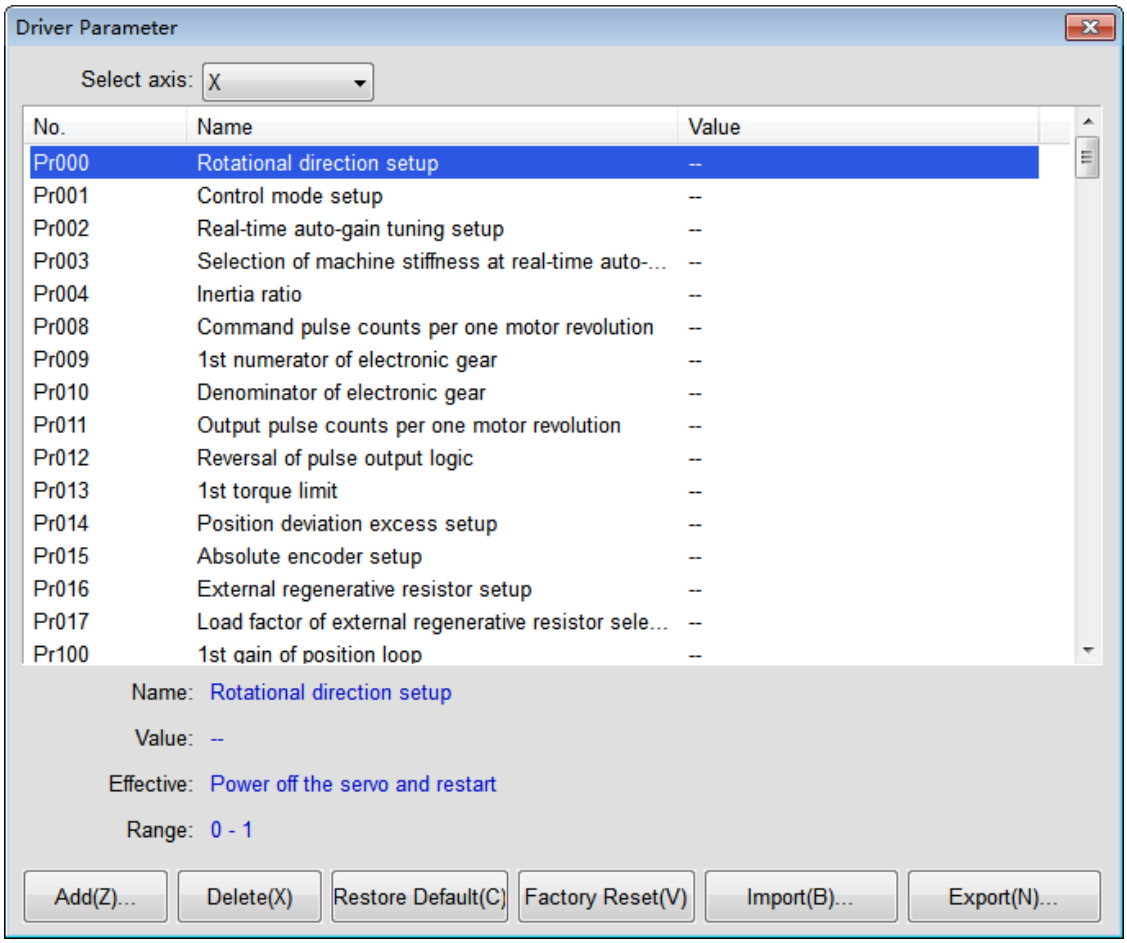
2.9.2 Set Drive Parameters

This operation is used to set drive parameters through **NcStudio**.

After **NcStudio V10 Waterjet Cutting Control System** is well connected to the drive, the system automatically reads drive parameters. And you can reset the drive parameters according to the structure of machine tool and your needs.

To set drive parameters, do the following:

1. Click **Machine Tool** → **Set Servo Param**. **Driver Parameter** dialog box pops up:



2. Select the target axis.

3. Double click target parameters, input the parameter value and click **OK**.

After setting drive parameters, do one of the following in **Driver Parameter** dialog box:

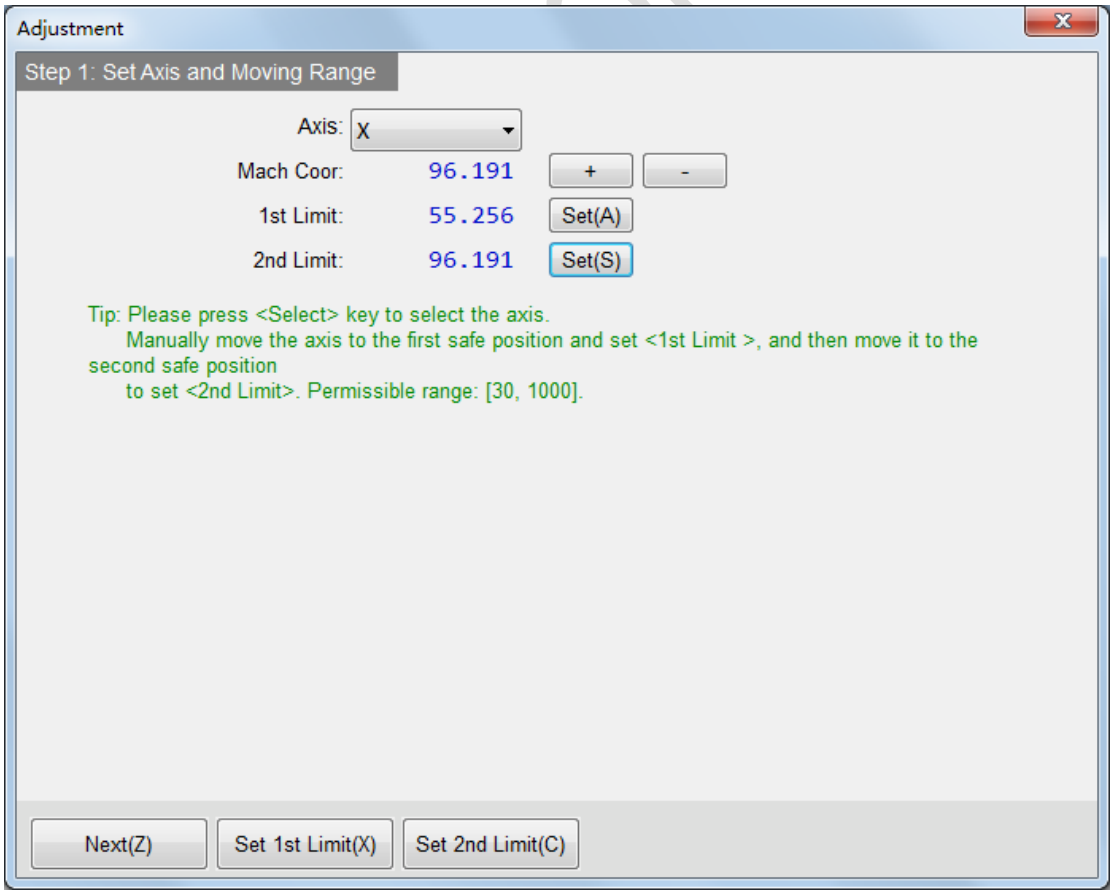
- Increase parameters.
- Delete parameters.
- Restore the initial list.
- Restore factory settings.
- Input settings of servo parameters.
- Export settings of servo parameters.

2.9.3 Execute the Automatic Adjustment

This operation is applicable to the bus control system. It is used to estimate inertia ratio according to the rigidity, limit range and track of each axis and load conditions, set the rigidity for each axis and then execute the related adjustment.

To execute the automatic adjustment, do the following:

1. Click **Machine Tool→ Auto Adjust**. **Adjustment** dialog box pops up:



2. To set axis moving range, do the following:

    1. Select the target axis.

    2. To set the first limit, move the machine tool to one safety position, and click **Set** after **1st Limit** or **Set 1st Limit** at the bottom of the dialog box.

    3. To set the second limit, move the machine tool to another safety position, and click **Set** after **2nd Limit** or **Set 2nd Limit** at the bottom of the dialog box.

    Note: The distance between the first limit and the second one should be within [15, 1000].

3. To select function options, click **Next**, and select an initial mode and rigidity mode.

4. Click **Next**, and click **Estimation Start**. The system automatically starts estimation.

5. To set rigidity parameters, click **Next**, and set these parameters according to machining needs.

6. Click **Start**. The system starts automatic adjustment.

    If you are not satisfied with the result, you can adjust related parameters and execute automatic adjustment again.

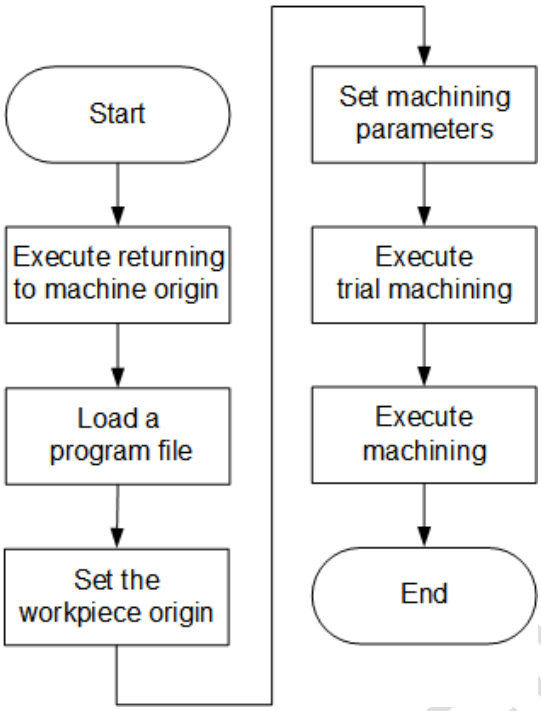
During automatic adjustment, according to your need, do one of the following:

- To stop automatic adjustment, click **Stop**.
- To save the parameter settings for later use, click **Save**.

3 Quick Start

Through this section, you can quick know the basic machining procedure of **NcStudio V10 Waterjet Cutting Control System** and start machining.

The basic procedure is as follows:



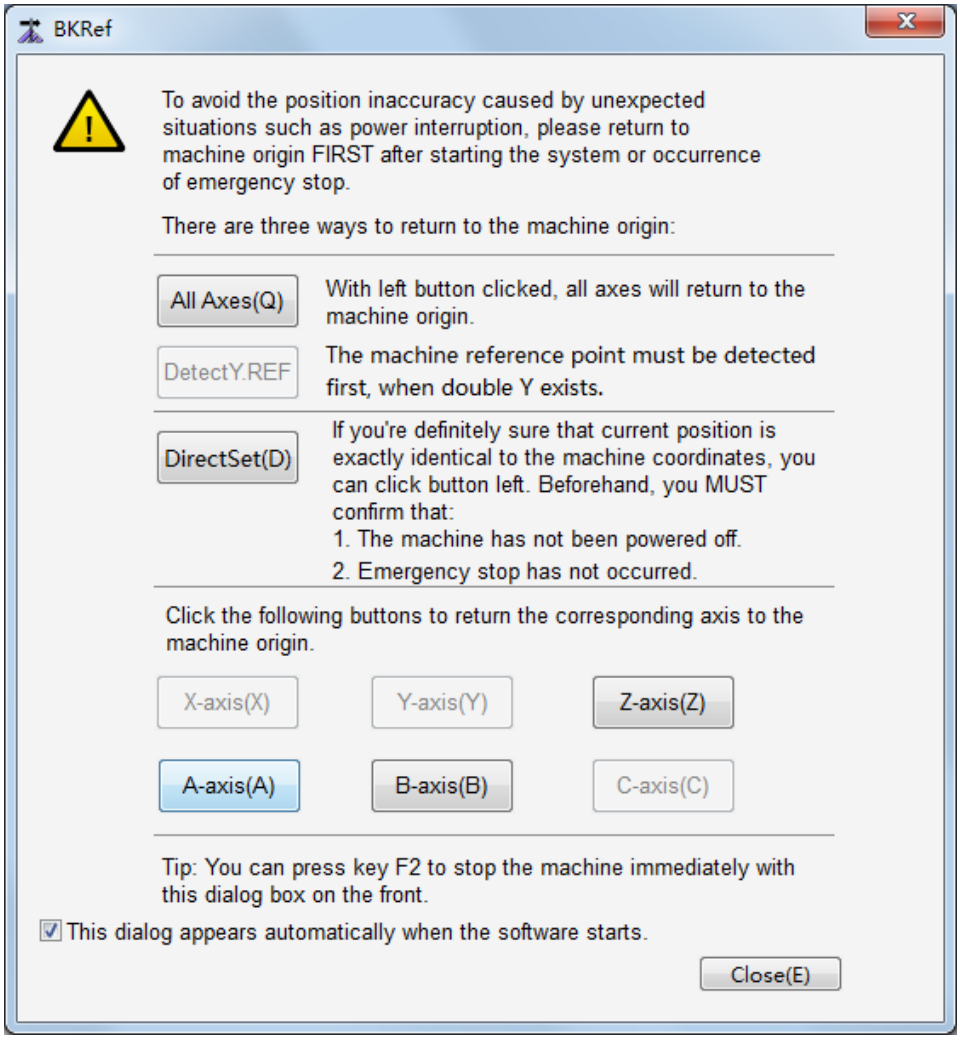
3.1 Execute Returning to the Machine Origin

This operation is used to make the machine tool return to the machine origin and adjust the coordinate system.

It is only applicable to the machine tool with an incremental encoder.

To execute returning to the machine origin, do the following:

1. To open **BKRef** dialog box, do one of the following:
   - Click **Operate → Back to Reference Point**.
   - In **Manual** mode, and click **REFER**.



You can check **This dialog box appears automatically when the software starts** at lower left corner. **BKRef** dialog box will automatically pop up when the software starts next time.

2. Do one of the following:

   - To return Z-axis to the machine origin first and return the other axes, click **All Axes**.
   - **Note:** For a machine tool with double Y axes, before clicking **All Axes**, you need to click **Detect Y REF** to detect the difference of encoder origins of double Y-axes.
   - If you ensure the following, click **Direct Set** to directly set the current position as the machine origin:
     - The current position is the same with the machine coordinate.
     - The machine has never been turned off.
     - E-stop has never occurred.
   - To return a single axis to the machine origin, click **X-axis** / **Y-axis** / **Z-axis** / **A-axis** / **B-axis** / **C-axis**.
   - For safety, it is recommended to return Z-axis first.

After executing returning to the machine origin, the sign appears after the corresponding axis in coordinate display area.

3.2 Load a Program File

This operation is used to load a program file before machining.

The supported format of the program file includes `G`, `DXF`, `ENG` and `PLT`.

To load a program file, do one of the following:

- Load a program file in NcStudio: You cannot edit the program file in **NcStudio**.
- Load a program file in NcEditor: You can edit the program file in **NcEditor**.
- Draw and load an object in NcEditor: You can load the object drawn in **NcEditor**.

The loaded program file shows in **POS** window.

3.2.1 Load a Program File in NcStudio

To load a program file in **NcStudio**, do one of the following:

- Click **File** → **Open and Load**, select the target program file and click **Open**.
- In **Auto** mode, in **POS** or **INFO** window, click **Load** and select the target program file.

3.2.2 Load a Program File in NcEditor

To load a program file in **NcEditor**, do one of the following:

- Click **File** → **Open** or directly click 

, select the target program file and click **Open**.
- Click **File** → **Import XXX File**, open the target program file with the corresponding format and click **Save**.

  The program file is saved as a NCE file. And it replaces the previous program file.

If there is already a program file and you need to insert another one, click **Edit** → **Insert XXX File**, open the target program file with the corresponding format and click **Save**. The program file is saved as a NCE file.

3.2.3 Draw and Load an Object in NcEditor

To draw and load an object in **NcEditor**, do the following:

1. To select a drawing tool, do one of the following:
   - Click **Draw**, and select the target tool.
   - Select the target tool in the drawing tool bar.
2. Draw an object in the drawing area.
3. To save the object as a NCE file, click **Save**. The object is automatically loaded.

3.3 Set the Workpiece Origin

The workpiece origin is the zero of X-axis / Y-axis / Z-axis / A-axis / B-axis / C-axis in the program file. Before machining, you need to set it.

To set the workpiece origin, do one of the following:

- Do clearing: by directly clearing workpiece coordinates.
- Set the workpiece offset: by setting workpiece offset.
- Set the current position as workpiece origin: by directly setting the current position.
- Set the workpiece origin for the object: by setting workpiece origin in **NcEditor**.

3.3.1 Do Clearing

This operation is used to clear the current workpiece coordinate of each axis and set machine coordinates of each axis as the corresponding workpiece offset.

To do clearing, do the following:

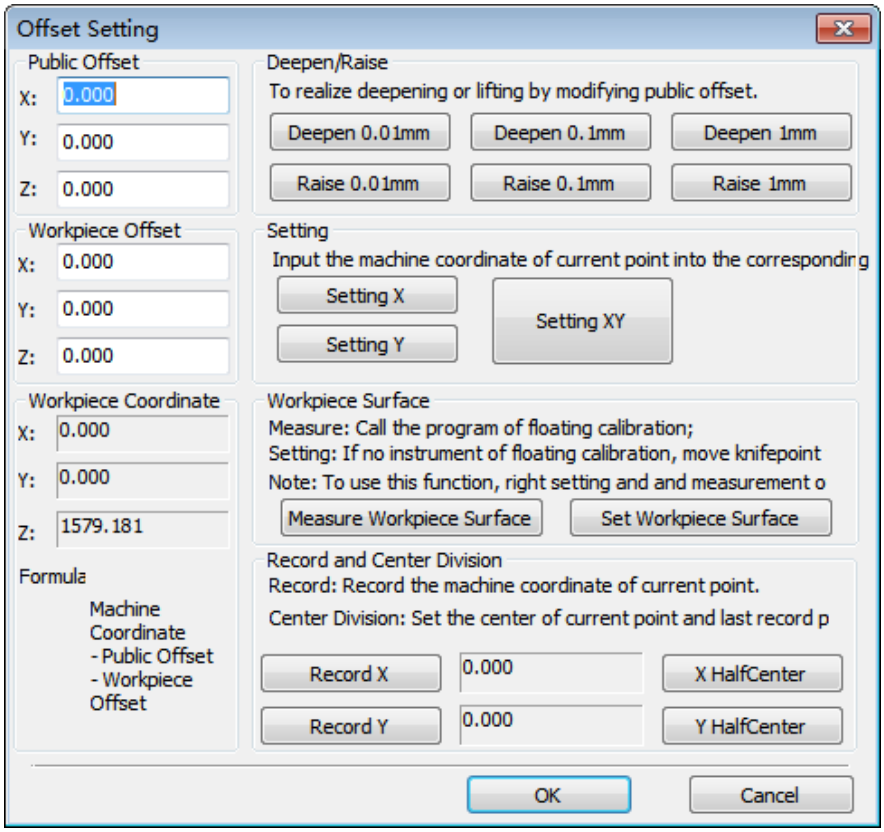
1. In **Manual** mode, move the machine tool to the target position that is to be set as the workpiece origin.
2. To do clearing, in **POS** window, do one of the following:
   - For one or several axes, click **Clear X**, **Clear Y**, **Clear Z** or **Clear XY**.
   - For all axes, click **Operate** → **Set Workpiece Origin**.
3. Click **Yes** in the confirmation dialog box. The system clears the current workpiece coordinates.

3.3.2 Set the Workpiece Offset

This operation is used to set the offset of workpiece origin that is relative to the machine origin in the corresponding WCS (G54 ~ G59).

To set the workpiece offset, do the following:

1. In **Manual** mode, click **Operate→ Offset→ Advanced Setting**. **Offset Setting** dialog box pops up:

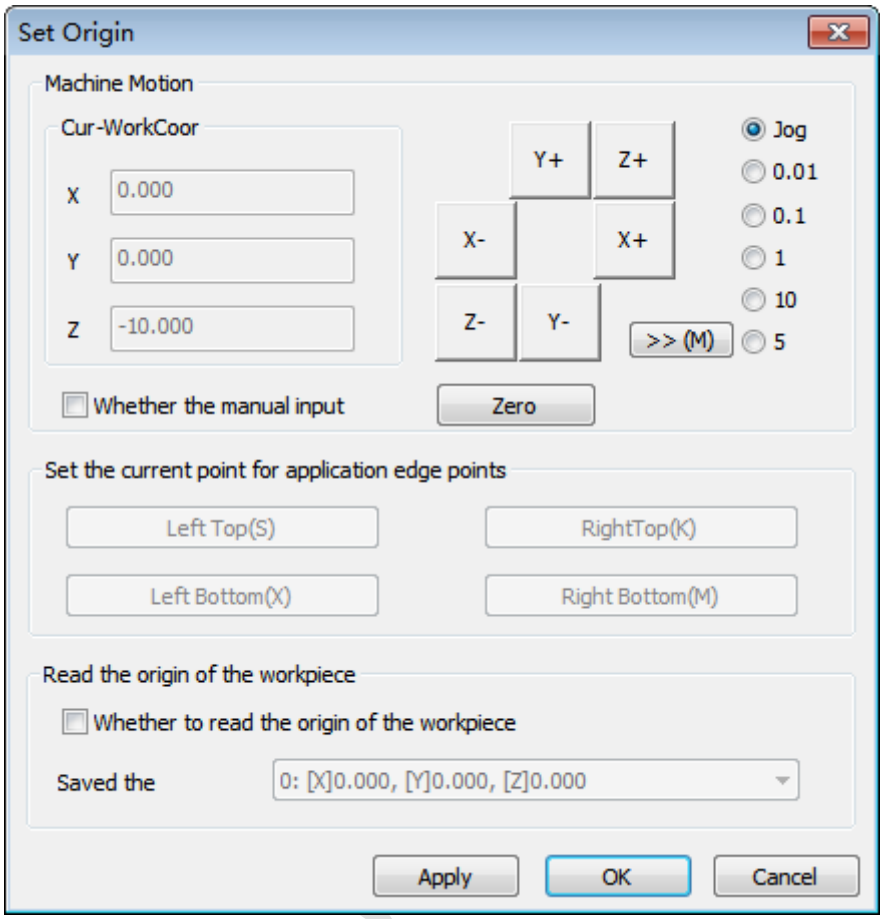


2. Click X / Y / Z input box in **Workpiece Offset** area, input the corresponding value and click **OK**.

3.3.3 Set the Current Position as Workpiece Origin

To set the current position as workpiece origin, do the following:

1. In **Manual** mode, click **Operate** → **Set Current Origin**. **Set Origin** dialog box pops up:



2. Do one of the following:

   - To clear workpiece coordinates of current position, move the machine tool to the workpiece origin by axis direction buttons and click **Zero**.
   - To directly input coordinates of the workpiece origin, check **Manual Input**, input coordinates of the workpiece origin and click **Apply** or **OK**.
   - Alternatively, click **Operate** → **Work Coor**, click the target input box, input coordinates of the workpiece origin and click **OK**.
   - To set the angular point as the workpiece origin, in **Set the current point for application edge points** area, click the corresponding button.

     Note: If the program file is empty, this way is unavailable and related buttons show gray.

- To load the saved workpiece origin, in **Read the origin of the workpiece** area, check **Whether to read the origin of the workpiece**, select the target workpiece origin and click **Apply** or **OK**.
- Alternatively, click **Operate** → **Load Workpiece Origin** and select the target coordinates.

3.3.4 Set the Workpiece Origin for the Object

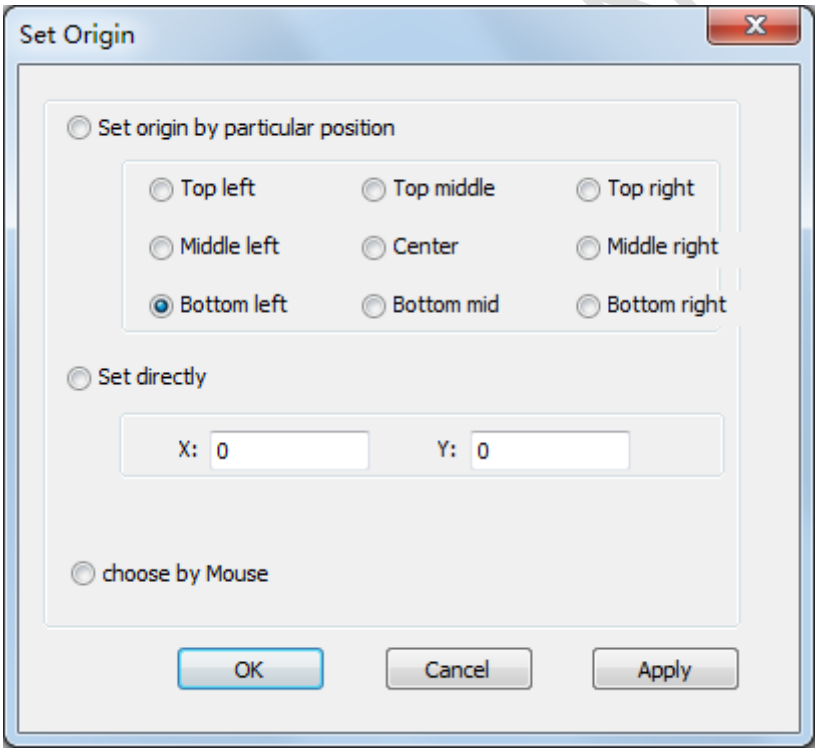
This operation is used to set the workpiece origin for the object in **NcEditor**.

To set the workpiece origin for the object, do one of the following:

- To set the workpiece origin in **Set Origin** dialog box, do the following:

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

.



2. Do one of the following:
   - To set the feature point on the object as workpiece origin, select **Set origin by particular position**, and select a feature point.

- To directly input coordinates of the workpiece origin, select **Set directly** and input its coordinates of X-axis / Y-axis.
- To set the workpiece origin by your mouse, select **Choose by mouse**, left click the target position in the drawing area and right click for confirmation.

- To directly set the workpiece origin in the drawing area, drag the sign to the target position that is to be set as the workpiece origin and right click for confirmation.

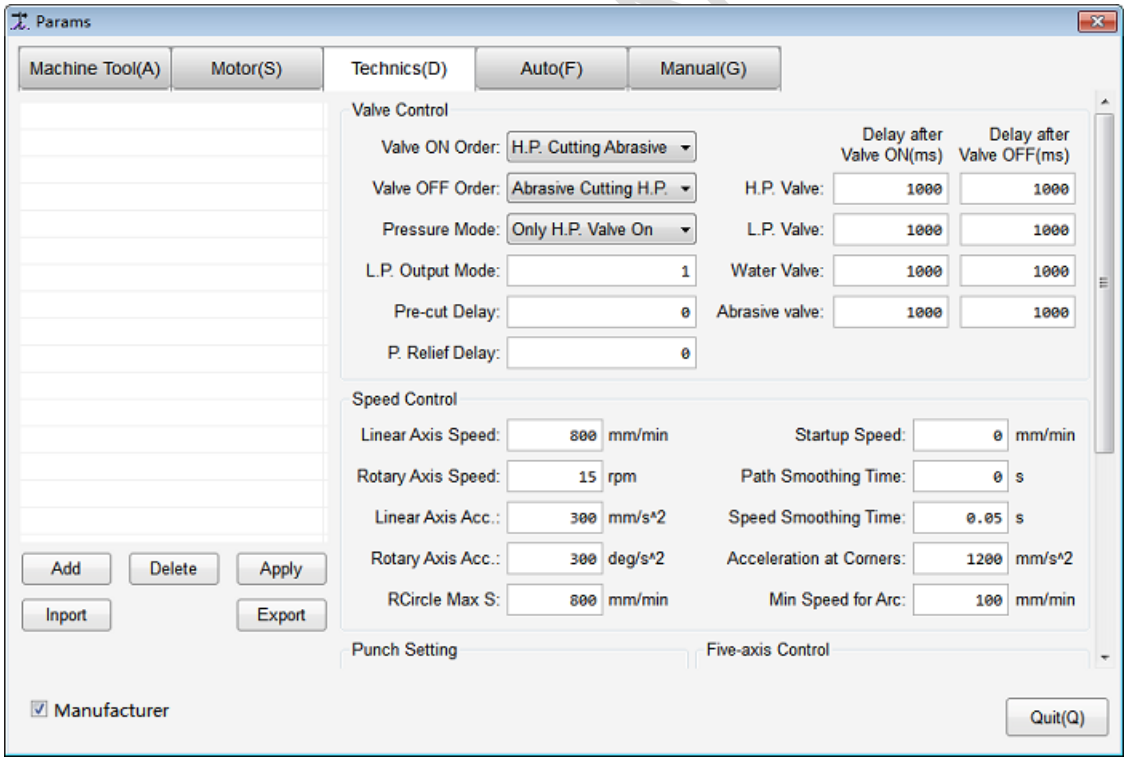
  is at the intersection of horizontal scale and vertical scale in the upper left corner of the drawing area.

3.4 Set Machining Parameters

This operation is used to set machining parameters before machining.

To set machining parameters, do the following:

1. Click **Machine Tool** → **Params**. **Params** dialog box pops up:



2. To set machining parameters, click **Technics** / **Auto** / **Manual** and set the related parameters:

   - Parameters in **Technics** interface
     - Valve control
     - Speed control
     - Punch setting
     - Five-axis control
   - Parameters in **Auto** interface
     - G00 speed
     - Z-axis jiggle
     - Safety setting for the tool head
     - Tool actions after machining
   - Parameters in **Manual** interface
     - Returning to REF point
     - Manual speed

   See Parameters for details.

3.5 Execute Simulation

This operation helps to know the machining track in advance, so as to avoid damages to the machine tool caused by programming mistakes.

Before executing simulation, ensure the program file has been loaded.

To execute simulation, do one of the following:

- Execute simulation by the system: The machine tool is not involved in machining.
- Execute a trial by the machine tool: The machine tool is involved in machining.

3.5.1.1 Execute Simulation by the System

Through this operation, **NcStudio V10 Waterjet Cutting Control System** does not drive the machine tool and only displays the toolpath in **POS** window, and you can visually check the machining program.

To execute the simulation by the system, do one of the following:

- Click **Operate** → **Simulation Mode**.
- In **Auto** mode, in operation button bar, click **Simmu** → **Start**.

The system automatically executes simulation. And 

shows in the upper left corner of **POS** window.

3.5.1.2 Execute a Trial by the Machine Tool

Through this operation, **NcStudio V10 Waterjet Cutting Control System** will drive the machine tool, and you can check coordinates of the machine tool and the toolpath in **POS** window.

In the area of valve control, click **Simulate** → **Start**. The trial automatically starts.

3.6 Execute Machining

To execute machining, do one of the following:

- Execute automatic machining
- Execute single block
- Execute machining from a near point / execute selective machining

Before executing machining, ensure the program file has been loaded.

3.6.1 Execute Automatic Machining

This operation is used to automatically execute machining from the beginning to the end.

To execute automatic machining, in **Auto** mode, do one of the following:

- In the menu bar, click **Operate** → **Start**.
- In operation button bar, click **Start**.

During automatic machining, you can do one of the following:

- To pause machining, do one of the following:
  - In the menu bar, click **Operate** → **Pause**.
  - In operation button bar, click **Pause**.
- To stop machining, do one of the following:
  - In the menu bar, click **Operate** → **Stop**.
  - In operation button bar, click **Stop**.

3.6.2 Execute Single Block

This operation is used to executing program by blocks. And after executing one block, the system enters pause status. Thus, it can be used for error diagnosis and troubleshooting.

To execute single block, do the following:

1. To enable single block, click **Operate** → **Single Block**.
2. In **Auto** mode, in operation button bar, click **Start**. The system enters pause status after executing one block of the program.
3. To continue to execute an another block, click **Start** again.
4. Repeat step 3 until the whole program has been executed.

3.6.3 Execute Machining from a Near Point / Selective Machining

This operation differs in the format of program file:

- For NCE format, this operation is used to execute machining from a near point.
- For the other formats, this operation is used to execute selective machining.

3.6.3.1 Execute Machining from a Near Point

During machining NCE files, with this method, you can move the tool head to some point precisely and visually and restart machining from this point.

To execute machining from a near point, do the following:

1. Click **Machine Tool** → **System Parameters**. **System Parameters** dialog box pops up.
2. Click **Operate**, find and set the parameter **Near Point Range** to a reasonable value.
3. In **Manual** mode, move the tool head to the position near the target point.

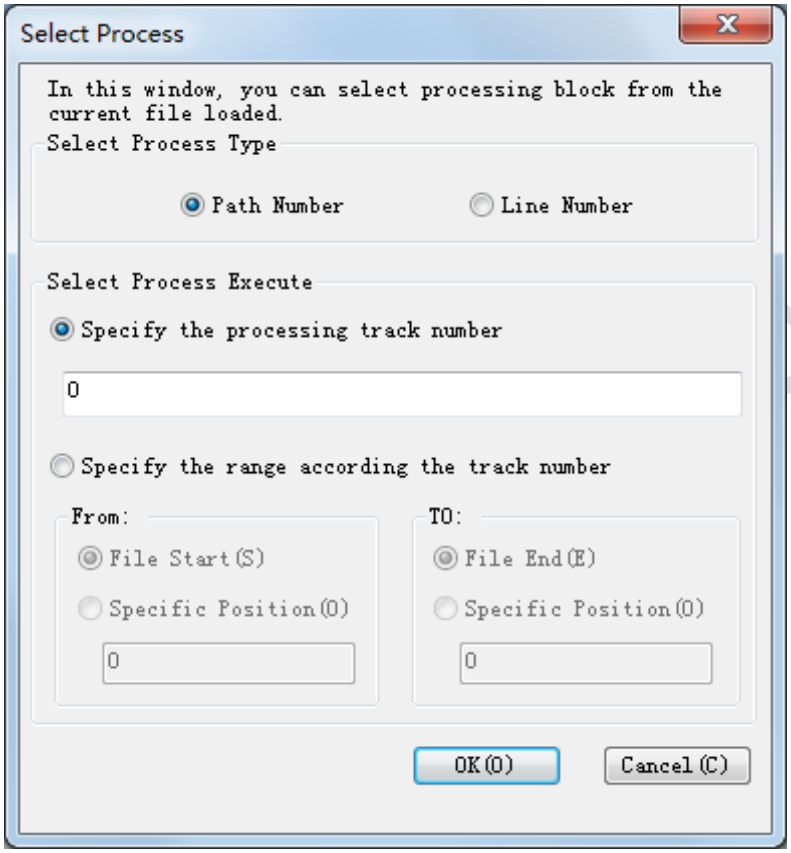
   The distance between the tool and the target point should not be greater than the value of parameter **Near Point Range**.
4. In **Auto** mode, to start machining from the position nearest to the tool position, do one of the following:
   - In the menu bar, click **Operate** → **NearPoint Processing Block**.
   - In operation button bar, click **Near Point**

3.6.3.2 Execute Selective Machining

During machining the program file whose format is not NCE, with this method, you can select any block to machine.

To execute selective machining, do the following:

1. In **Auto** mode, click **Operate** → **NearPoint Processing Block**. **Select Process** dialog box pops up:



2. Select a machining type:
   - Path number: It is automatically generated in **POS** window.
   - Line number: It shows in **POS** window.
3. Specify the machining range:
   - Select **Specify the processing track number** and input a specified path number.

     It is available when the machining type is **Path number**.

– Select **Specify the range according to track number** and do the following:

  - To specify the start position, select **File Start**, or select **Specific Position** and input a specified path number or line number.
  - To specify the stop position, select **File End**, or select **Specific Position** and input a specified path number or line number.

4. Click **OK**. The system starts machining within the specified range.

4 Special Operations

Through this section, you can quick know special operations of **NcStudio V10 Water Cutting CNC Control System**.

Special operations include the following:

- Execute Manual Contouring
- Height Measurement
- Use Manual Locating
- Execute Scanning QR
- Set Punching Parameters for the Waterjet Tool
- Check Time Statistics for High Pressure Valve and Oil Pump
- Execute Z-axis Jiggle
- Execute Y1Y2 Jiggle
- Back up Technic Parameters
- Nest Parts
- Use the Drill
- Use the Velocity Model Calculator
- Shoot with a Camera
- Adjust Five-axis Tool Head Errors

4.1 Execute Manual Contouring

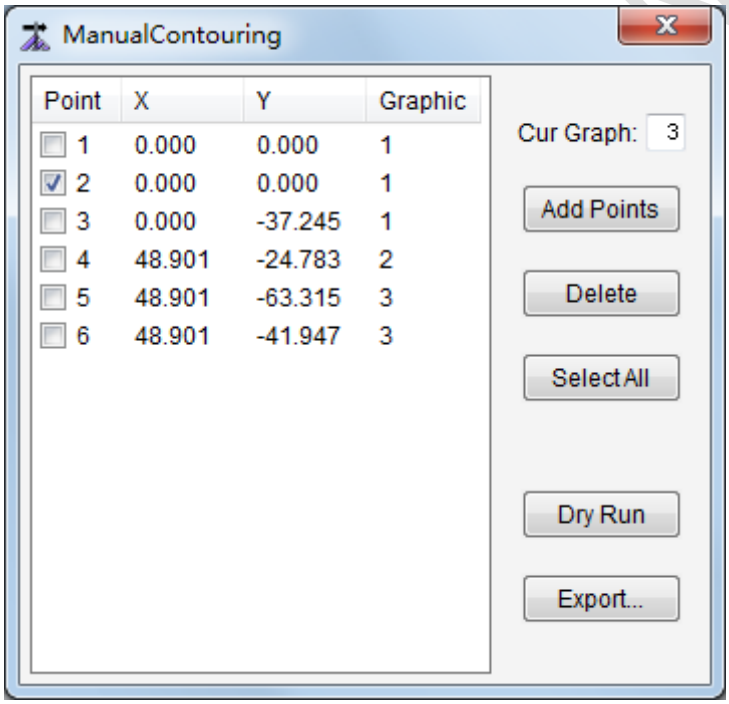
In stone mosaic industry, this operation is used to deal with sheets with flaws or cracks by moving the tool head around the area that you would like to remove, and generating a DXF file according to the regular-shaped contour. When drawing or nesting toolpaths, you can open the DXF file to remove the area.

Before executing manual contouring, ensure the following:

- Each axis has been returned to the machine origin.
- The version of the following softwares meet requirements:
  - **NcStudio**: 10.515.2 or above
  - **CAD**: 2004 or above.

To execute manual contouring, do the following:

1. In **Manual** mode, move the tool head to the target position.
2. Click **Operate** → **Manual Contouring**. **Manual Contouring** dialog box pops up:



3. To add the current position of tool head to the list in **Manual Contouring** dialog box, click **Add Points**.
4. To get the whole contour of the target area that you would like to remove, repeat step 2 and 3.

5. **Optional:** If there are several areas that you would like to remove, to add points for the target areas, modify the parameter **Cur Graph** to the corresponding number of area and repeat step 2 ~ 4.

   All added points belong to the area whose number matches with the value of parameter **Cur Graph**. For example, when the value is 2, then all added points belong to the area 2 after clicking **Add Point**.

6. Check if Z-axis is higher than or at the safety height:

   - Yes: Proceed to the next step.
   - No: Lift Z-axis to the safety height.

7. Click **Dry Run** and check if the contour is reasonable:

   - Yes: Proceed to the next step.
   - No: To modify the contour, do one of the following:
     - To delete unnecessary or wrong points, check these points and click **Delete**.
     - To delete all points, click **Select All** → **Delete**.
     - To modify the coordinate of a point, double click its coordinate input box and input the target coordinate.

8. To export the contour as a DXF file, click **Export**.

   The workpiece origin is the zero point of exported DXF file.

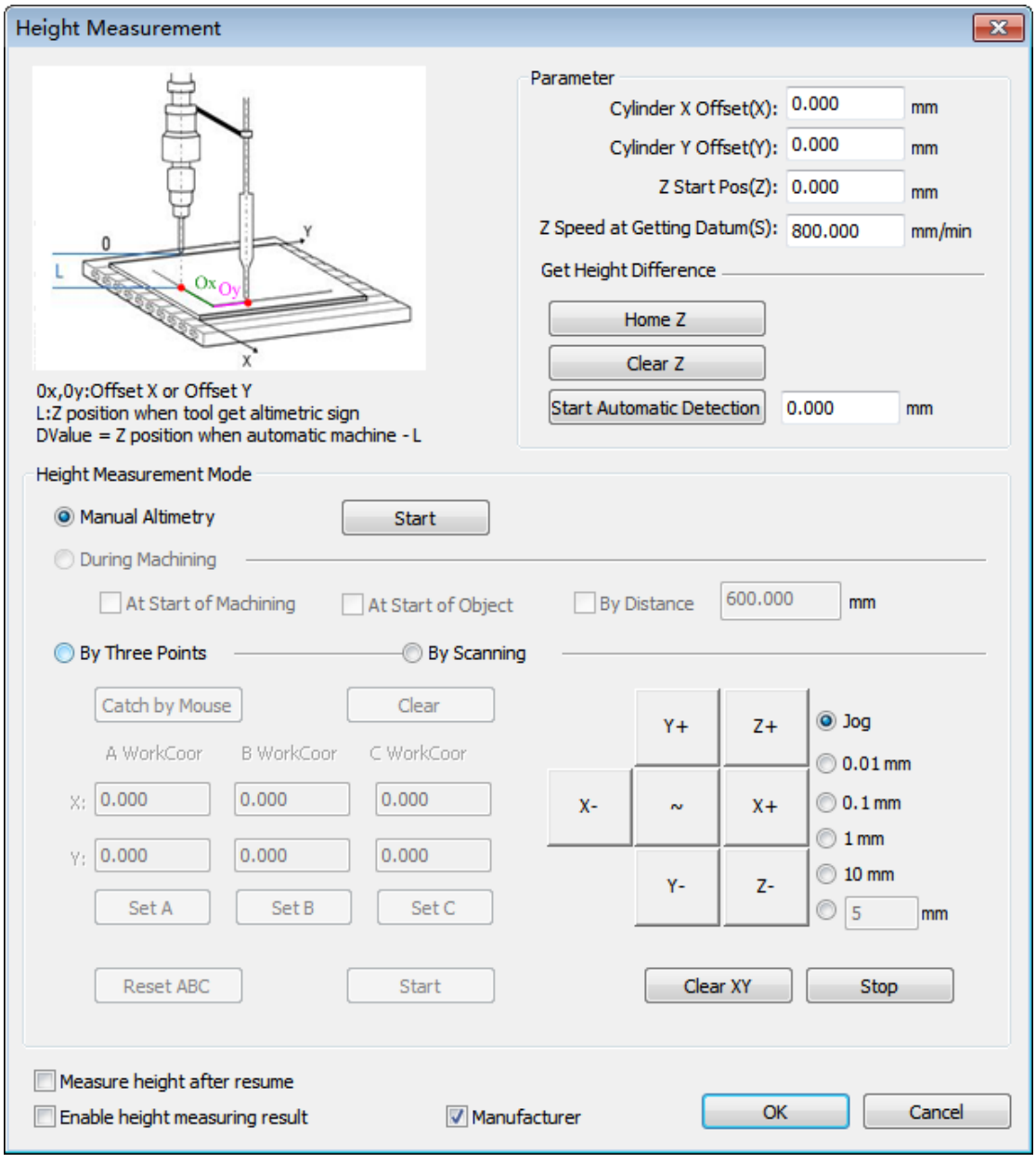
To draw or nest the contour, open the DXF file with **CAD** software and do related edition.

4.2 Measure Height

This operation is used to guarantee the same distance from the tool head to the workpiece surface, so as to avoid the impact on cutting quality caused by change of cutting height.

To measure height, do the following:

1. In **Manual** mode, click **Altimetry**. **Height Measurement** dialog box pops up:



2. To activate input boxes in **Parameter** area, check **Manufacturer** and input the manufacturer password.

3. Set the following standard parameters in **Parameter** area:
   - Cylinder X Offset: The offset of height measuring unit relative to the tool head in X-axis direction.
   - Cylinder Y Offset: The offset of height measuring unit relative to the tool head in Y-axis direction.
   - Z Start Pos: The machine coordinate of Z-axis before starting height measurement.
   - Z Speed at Getting Datum: The speed of Z-axis at the time of getting datum position.
4. **Optional:** For height measurement with the laser sensor, set the baud rate of serial ports.
5. Choose a way of height measurement:
   - With the cylinder:
     - Manual height measurement
     - In-process height measurement
     - Three-point height measurement
   - With the laser sensor:
     - Manual height measurement
     - Three-point height measurement
     - Scanning height measurement

4.2.1 Set Standard Parameters

This operation is used to set related standard parameters of height measurement.

To set standard parameters, do the following:

1. In **Parameter** area of **Height Measurement** dialog box, according to the actual situation, set parameters **Z Start Pos** and **Z Speed at Getting Datum**.
2. To set parameters **Cylinder X Offset** and **Cylinder Y Offset**, do the following:
   1. Move the tool head to the proper position, click **Clear XY** and punch a hole on the workpiece.
   2. Open the cylinder of height measuring sensor and push out the height measuring sensor.
   3. To move the laser spot to the hole center, turn on laser and move X-axis and Y-axis.

4. Respectively input the current workpiece coordinates in **Cylinder X Offset** and **Cylinder Y Offset** input boxes.

3. To get the height difference, do the following:

   1. Manually move the abrasive tube to the cutting height and click **Clear Z**.

   2. Click **Start Automatic Detection**.

   After detection, the system automatically fills in height difference.

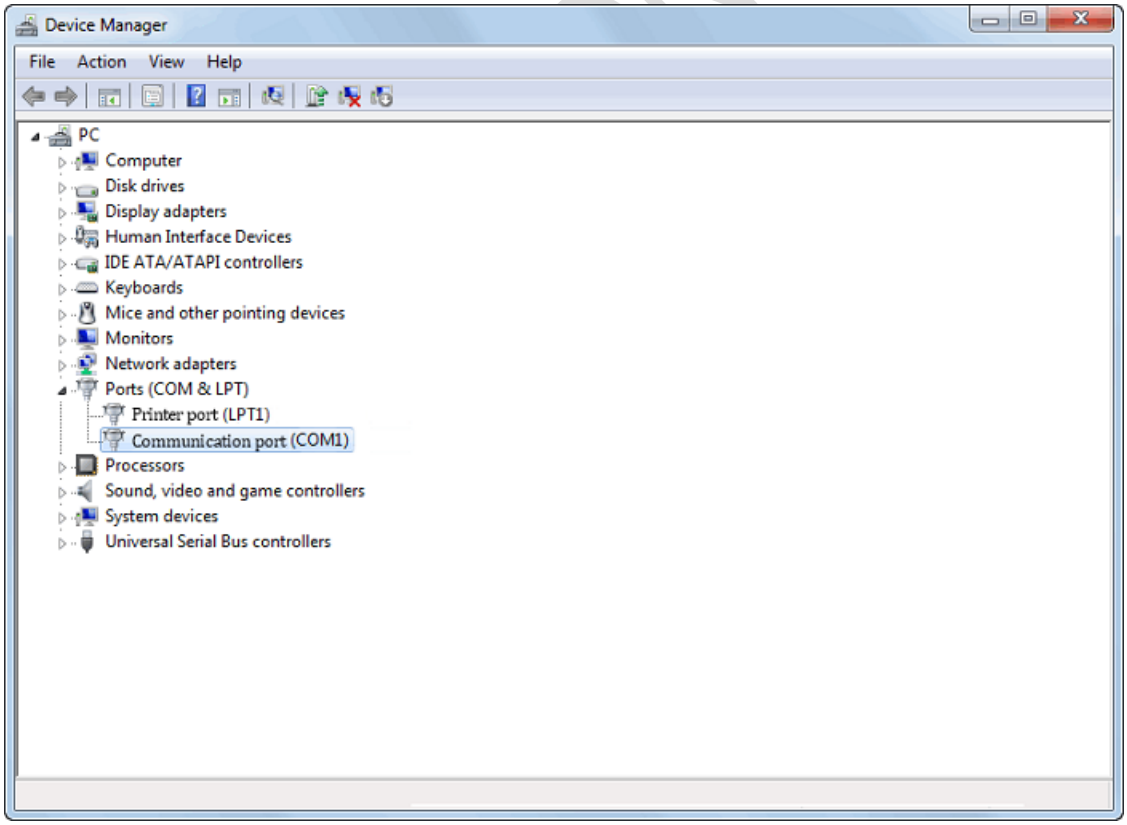
4.2.2 Set Baud Rate of Serial Ports

This operation is used to set the baud rate of serial ports on the host, so as to ensure the communication between the laser sensor and host.

It is applicable to height measurement with the laser sensor.

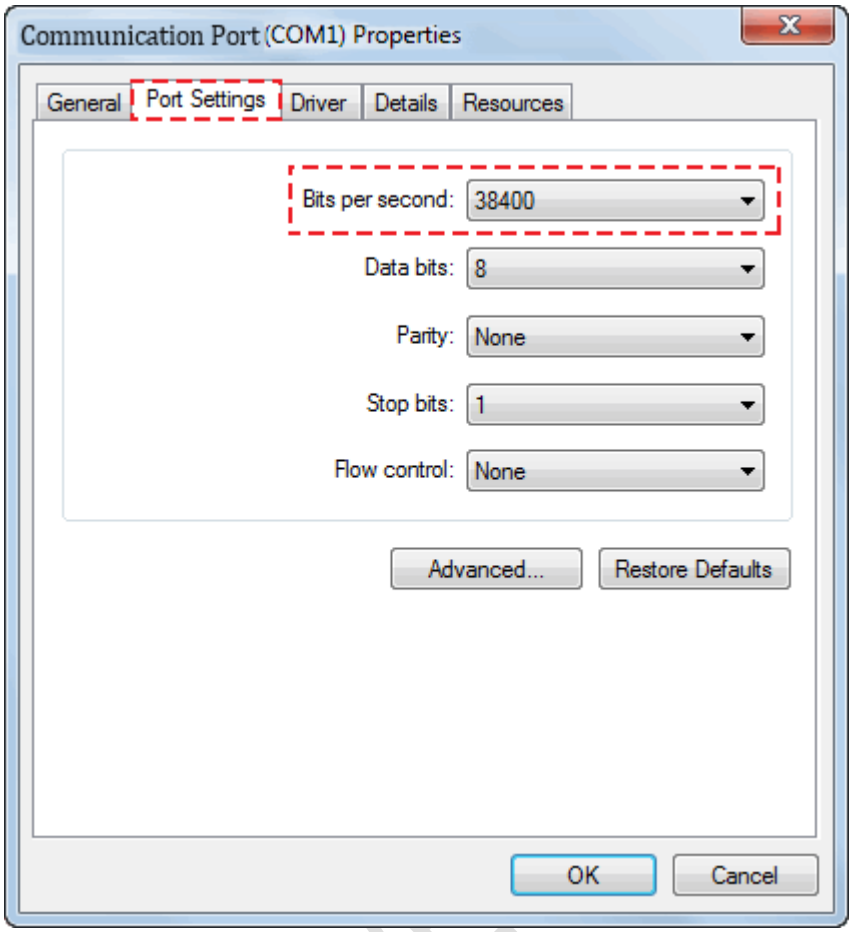
To set baud rate of serial ports, do the following:

1. To open **Device Manager** dialog box in the computer desktop, click **Start** → **Control Panel** → **Device Manager**. **Device Manager** dialog box pops up:



2. Click **Ports(COM/LPT)**, and double click the target communication port. **Properties** dialog box pops up.

3. Click **Port Settings** and set **Bits per second** to **38400**:



4.2.3 Measure Height with the Cylinder

This operation is used to measure height by traditional contact measuring method.

Before measuring height with the cylinder, ensure the following:

- The target workpiece has been placed on the workbench.
- The parameter **N79201 Software Configuration** has been set to **1**.
- Z-axis has returned to the machine origin.
- The workpiece origins of X-axis and Y-axis have been defined.

To measure height with the cylinder, in **Height Measurement Mode** area of **Height Measurement** dialog box, select one of the following measuring methods:

- Manual height Measurement
- In-process height measurement
- Three-point height measurement

Note: The result of height measurement is still effective after reloading the program file or restarting the software. To disable the result, please uncheck **Enable height measuring result** in **Height Measurement** dialog box .

To measure height during breakpoint resume, check **Height measuring or not when resume**.

4.2.3.1 Execute Manual Height Measurement

This operation is used to measure height for a single-point, and execute tool calibration by one key which saves troubles of tool calibration by clearing.

To execute manual height measurement, do the following:

1. Move the tool head to the target measuring point.
2. In **Height Measurement Mode** area of **Height Measurement** dialog box, select **Manual Altimetry**.
3. Click **Start**. Height measurement starts.

During height measurement, you can do the following if there is something wrong:

- If exceptions in the height measuring unit occur, measuring position is incorrect, or it is going to collide with the machine tool, click **Stop** to stop height measurement.
- If the toolpath origin is set incorrectly, correct it, click **XY Clear** to clear workpiece coordinate of X-axis and Y-axis in the correct position, and measure height again.

4.2.3.2 Execute In-process Height Measurement

This operation is used to measure height during cutting. And it is applicable to the workpiece with a relatively hard and smooth surface, whose machining effect is not influenced by scratches.

To execute in-process height measurement, do the following:

1. In **Height Measurement Mode** area of **Height Measurement** dialog box, select **During Machining**.
2. To select the start point for height measurement, do one of the following:
   - Check **At Start of Machining**. Height measurement starts at the start point of program file.
   - Check **At Start of object**. Height measurement starts at the start point of each object.
   - Check **By Distance** and set the distance between the workpiece surface and tool head. If the actual distance is equal to the set distance during cutting, height measurement starts.
3. Click **OK**. Height measurement starts.

4.2.3.3 Execute Three-point Height Measurement

This operation is used to identify a plane through three points. And it is applicable to the workpiece with an even surface and a tilt angle relative to the workbench.

To execute three-point height measurement, do the following:

1. In **Height Measurement Mode** area of **Height Measurement** dialog box, select **By Three Points**.
2. To get three points (A, B, C) within the workbench, do one of the following:
   - Manually input workpiece coordinates of A, B and C.
   - Respectively move X-axis and Y-axis to A / B / C and click **Set A** / **Set B** / **Set C**. A, B and C are set as a measuring point.

     If you are not satisfied with the results, click **Reset ABC** and reset them.
   - To directly pick A, B and C on the object, click **ABC by mouse** and select three points in **POS** window.

     If you are not satisfied with the results, click **Clear** and reset them.
3. Click **Start**. Height measurement starts.

During height measurement, you can do the following if there is something wrong:

- If exceptions in the height measuring unit occur, measuring position is incorrect, or it is going to collide with the machine tool, click **Stop** to stop height measurement.
- If the toolpath origin is set incorrectly, correct it, click **XY Clear** to clear workpiece coordinate of X-axis and Y-axis in the correct position, and measure height again.

4.2.4 Measure Height with the Laser Sensor

Compared with height measurement with the cylinder, this way can greatly improve the measuring precision and avoid possible damage to the workpiece by keeping the same distance between the tool head and the workpiece surface. And it is a perfect choice for rock workpieces with three-dimension or irregular surfaces.

Before using height measurement with the laser sensor, ensure the following:

- The laser sensor has been installed correctly.
- The wiring between the laser sensor and host is correct.
- The baud rate of serial ports has been set.

To use height measurement with the laser sensor, do the following:

1. In **Height Measurement Mode** area of **Height Measurement** dialog box, select **By Scanning** and set **Sensor Port Address** to the same port address with that in **Device Manager** dialog box of the host.
2. Restart the software.

   When the log *Serial port has been successfully enabled* appears, the laser sensor is successfully connected with the host.
3. In **Manual** mode, click **Altimetry** in operation button bar, select **By Scanning** and click **Open the Light**, and observe if the light can be normally turned on or off:
   - Yes: Proceed to the next step.
   - No: Check the wiring and related settings of the communication port and laser sensor.
4. To initialize the laser sensor, click **Initialize Sensor**.

   When the prompt *Succeeded in initializing the laser sensor* appears, the laser sensor is successfully initialized.

5. To execute height measurement, select one of the following measuring methods:
   - Execute manual height measurement
   - Execute three-point height measurement
   - Execute scanning height measurement

4.2.4.1 Execute Manual Height Measurement

See Execute Manual Height Measurement for details.

4.2.4.2 Execute Three-point Height Measurement

See Execute Three-point Height Measurement for details.

4.2.4.3 Execute Scanning Height Measurement

With this completely new method, the scanning track can be automatically generated according to the preset workpiece information.

During height measurement, the system automatically gets the datum at the start point of scanning track, then executes continuous scanning along the preset track at rapid speed, and gives feedback about precise height though the laser sensor.

To execute scanning height measurement, do the following:

1. In **Height Measurement Mode** area of **Height Measurement** dialog box, select **By Scanning**.
2. Set the following parameters:
   - Scanning direction: The axis moving direction during scanning height measurement.
   - Horizontal space: It takes effect when scanning type is array scanning.
   - If the scanning direction is positive direction of X-axis, it is recommended to use the default horizontal space (50mm).
   - Vertical space: It takes effect when scanning type is array scanning.
   - If the scanning direction is positive direction of Y-axis, it is recommended to use the default vertical space (50mm).
   - Scanning speed: The axis moving speed during scanning height measurement.
   - To ensure the measuring precision, it is recommended to set it less than 15000mm/min.
   - Travel lower limit: The minimum machine coordinate of Z-axis during scanning height measurement.

- Scan type:
  - Array scanning: Scan the workpiece along reciprocating lines that are automatically generated according to the object.
  - Track scanning: Scan the workpiece along the toolpath.
- Height tolerance: The maximum deviation of height when XY-plane moves 10mm.
- Max height difference: The tolerate of difference between the maximum height and minimum one during scanning height measurement.
- Ignore abnormal: Whether to ignore the limit of the parameter **Max Ignorable Distance** during scanning height measurement.

About parameter details, see Laser Height Measurement.

3. Click **Start**. The system automatically generates the scanning track according to parameter settings and executes scanning height measurement.

After height measurement, check **Enable height measuring result** and click **OK**.

During height measurement, when one of the following situations occurs, data acquisition fails, an alarm *Failed to collect data for Nth measuring point* occurs and the system automatically stops scanning:

- An exception occurs during scanning, e.g. exception in the power supply of the laser sensor, or the communication of the laser sensor, scanning is out of range, etc.
- The height difference of the measuring area exceeds the set value of parameter **N93214 Height Measurement Offset**.
- Sheet flaws.

To continue to scan the remaining track, click **B.P. Resume** after dealing with these problems.

4.3 Use Manual Locating

This operation is applicable to the following situations:

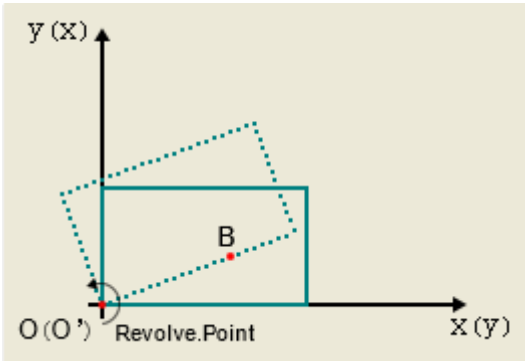
- During loading, precisely locating workpiece whose size is too large and the weight is too heavy and establishing a new coordinate system.
- Based on the actual cutting situation, maximize utilization rate of workpiece during nesting.

Before using manual locating, ensure the following:

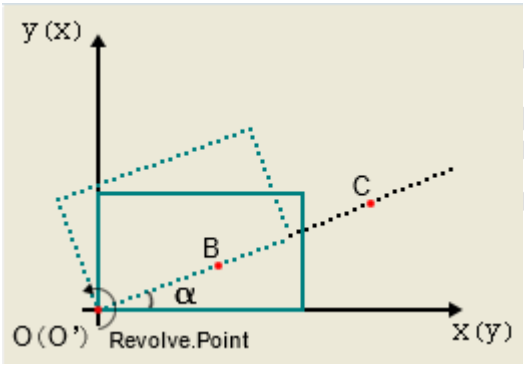
- The NCE file has been loaded into **NcStudio** software.
- The workpiece has been placed on the workbench.

The methods of manual locating include the following:

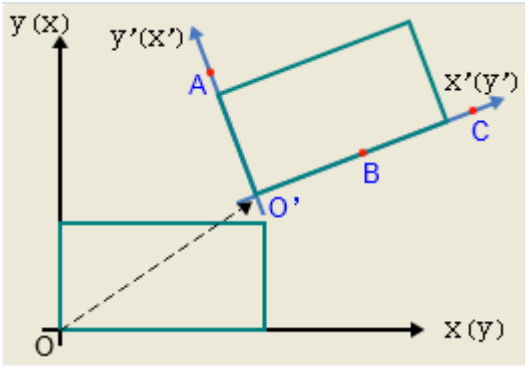
- One-point



- Two-point

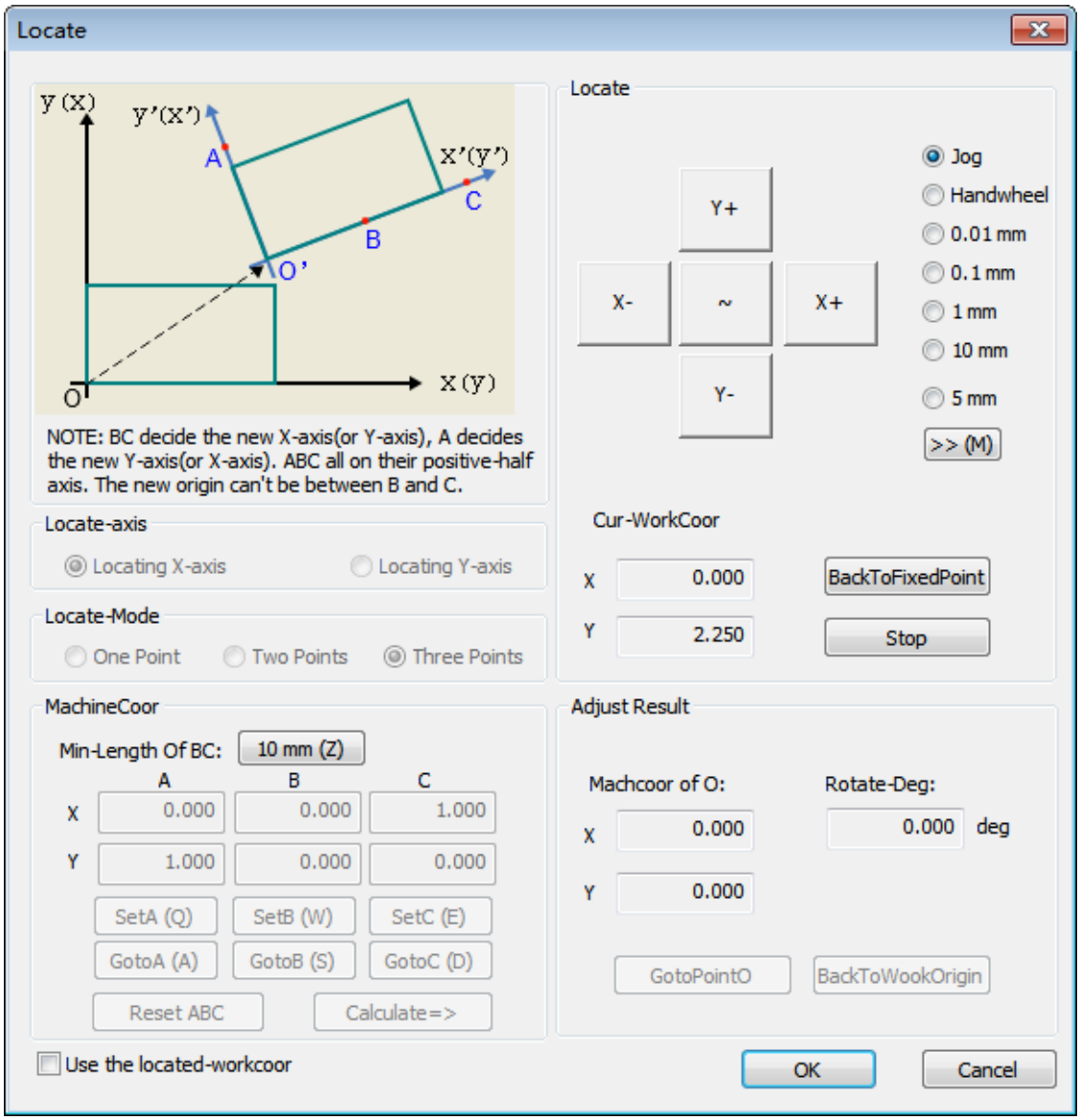


- Three-point



Taking the two-point method as an example, to use manual locating, do the following:

1. In **Manual** mode, click **Operate** → **Locate Coor**. **Locate** dialog box pops up:



2. Check **Use the located work-coor** and select the axis whose travel is larger to locate in **Locate-axis** area.

3. Select **Two-point** in **Locate-Mode** area and set **Min-Length of BC** in **Machine Coor** area.

4. To get the machine coordinate of a point (B) on the workpiece edge, move the tool head in positive direction of the axis to the workpiece edge and click **Set B** in **Machine Coor** area.

5. To get the machine coordinate of an another point (C) on the workpiece edge, move the tool head in positive direction of the axis to the workpiece edge and click **Set C**.

   Note: Please ensure B and C are in positive direction. Otherwise, the new origin cannot be between B and C.

If you do not need to use manual locating again before machining normally ends, uncheck **Use the located-workcoor** in **Locate** dialog box and click **OK**.

After locating:

- If C coordinate is correct, the software automatically calculates the rotating angle of the workpiece relative to the machine coordinate system. Every time the system starts cutting, the workpiece coordinate system is automatically rotated to the corresponding angle.
- If the actual distance between B and C is less than the setting value of **Min-Length of BC**, the prompt *B and C is too close.* appears.
- If the rotating angle exceeds 45°, it means that the workpiece is over tilted. Please place the workpiece and execute locating again.

After cutting, locating results will be automatically disabled but not removed:

- If locating is not done for another cutting process, the cutting starts without rotating angle.
- If the results are desired for another cutting process, check the box in front of **Use the located-workcoor** to apply last locating results to this cutting.

4.4 Scan a QR Code

This operation is used to recognize and load a program file in **NcStudio V10 Waterjet Cutting CNC System** with a barcode scanner.

4.4.1 Name the Program File

The supported format of program files include DXF, NC, G and NCE.

According to the following conventions, name the program file:

- The file name should contain at least 1 non-space character, and at most 28 characters.
- The file name can only contain the following:
  - English letters
  - Numbers
  - Special characters: “~”, “!”, “@”, “#”, “$”, “%”, “^”, “&”, “( )”, "_“,”+“,”{ }“,”[ ]“,”;“,”“,”,“,”.“,”`“,”=" and space

4.4.2 Recognize the Program File

This operation is used to recognize the program file with a barcode scanner.

Before recognizing the program file, ensure the following:

- The version of **NcStudio** software is 10.515.2 or above.
- The name of program file is correct.
- The characters on the barcode are the same with the name of program file.
- The system is in English input mode.
- Parameter **Barcode Enable** is set to **Yes**.
- The barcode scanner is connected correctly.

To recognize the program file, do the following:

1. Start the system and ensure it is in idle status.
2. To set a storage path for the program file, click **File** → **Set File Path**.
3. Put the target program file under the set storage path.

   If there exist files with the same name, but in different formats, the format loaded first is decided by the parameter **Priority Loading File Format**.
4. To ensure the focus is on the software interface, click on it by your mouse.
5. Scan the corresponding barcode with the barcode scanner.

If scanning succeeds, the system automatically loads the program file and shows the machining track.

If scanning fails, troubleshoot the following:

- If the prompt *The file path is illegal. Please check.* appears, modify the storage path for the program file.
- If the prompt *Current file does not exist. Please check.* appears, modify the storage path for the program file or put the program file under the set storage path.
- Check if parameter **Barcode Enable** is set to **No**.
- Check if the system is in English input mode.

4.5 Set Punching Parameters for the Waterjet Tool

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 reduce material consumption in a great extent.

To set punching parameters for the waterjet tool, do the following:

1. Click **Machine Tool** → **System Parameters**. **System Parameters** dialog box pops up.
2. Click **Water Jet** and set punching parameters.

   About punching parameters, see Punching Parameters.
3. Reload the program file. The modification to punching parameters takes effect.

   See Load the Program File for details.

During machining, the whole process is as follows:

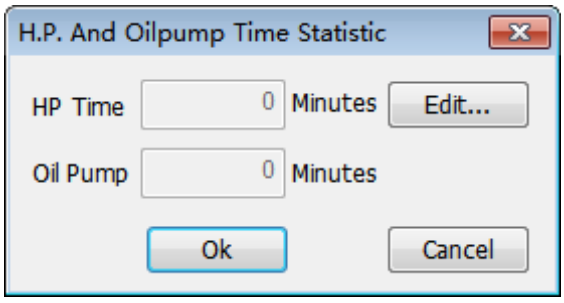
1. Execute punching.
2. Cut the lead line.
3. Machine the program file.

4.6 Check Time Statistics for High Pressure Valve and Oil Pump

This operation is used to check the used time of valuable and wearing parts high pressure valve and oil pump, so as to regularly maintain them and replace them in time.

To check time statistics for high pressure valve and oil pump, do the following:

1. Click **Machine Tool** → **H.P and Oil Pump Time Statistic**. **H.P and Oil Pump Time Statistic** dialog box pops up:



2. Check the used time for the high pressure valve and oil pump, and judge if it is necessary to maintain or replace them.

   If necessary, maintain or replace them and do the following:

   1. To activate input boxes **HP Time** and **Oil Pump**, click **Edit** and input the manufacturer password.

   2. To clear the used time for the high pressure valve and oil pump, set parameters **HP Time** and **Oil Pump** to **0**.

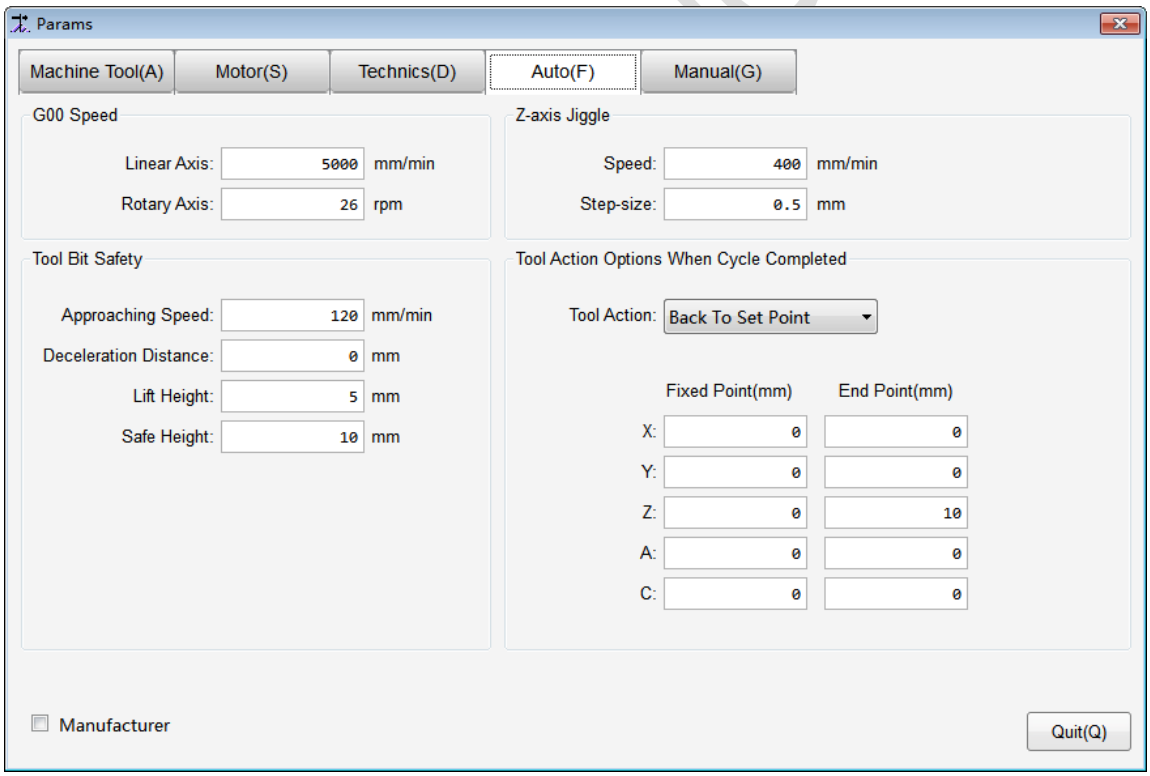
4.7 Execute Z-axis Jiggle

This operation is used to adjust height of the tool head, so as to find the best cutting height.

Before executing Z-axis jiggle, ensure the system is in running status or pause status.

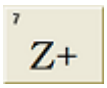
To execute Z-axis Jiggle, do the following:

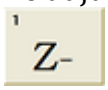
1. Click **Params**. **Params** dialog box pops up:



2. Click **Auto** to enter **Auto** interface and set the following parameters in **Z-axis Jiggle** area:

   - Speed: The speed of moving axis during jiggle.
   - Step size: The step size of moving axis during jiggle.

3. To adjust the tool head to the best cutting height, in **Auto** mode, click 

or 

(or press **7** or **1** on the keyboard).

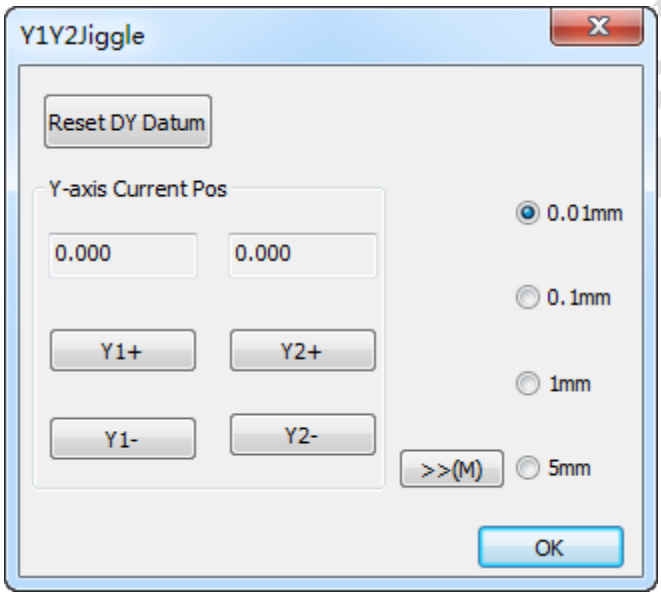
4.8 Execute Y1Y2 Jiggle

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

It is only applicable 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 Y1Y2 jiggle, do the following:

1. To activate the button **Y1Y2 Jiggle**, in **POS DIAG** window, click **Fix Y Only**.
2. Click **Y1Y2 Jiggle**. **Y1Y2 Jiggle** dialog box pops up:



3. Select a step size or click 

to customize a step size.
4. According offsets of Y1-axis and Y2-axis, click axis buttons and move Y1-axis or Y2-axis.

For an absolute encoder, after executing Y1Y2 jiggle, click **Reset DY Datum** to directly reset the datum of Y-axis.

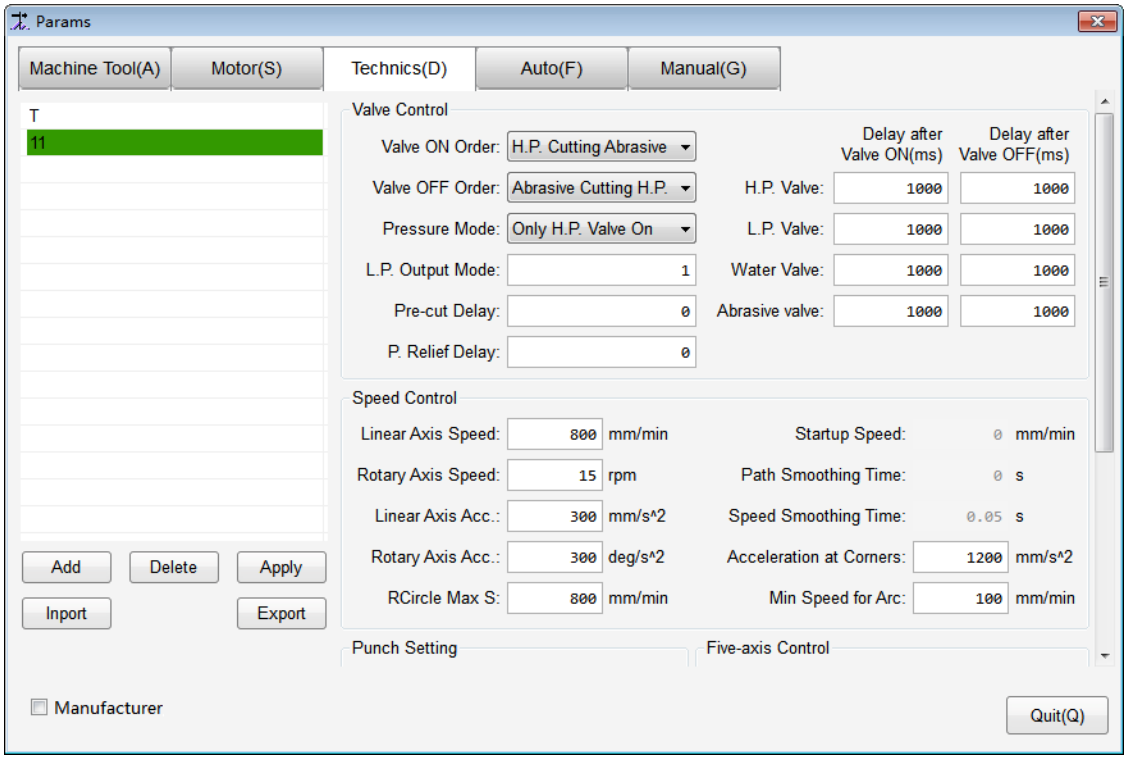
4.9 Back up Technic Parameters

This operation is used to back up technic parameters, so as to conveniently edit and use them.

Before backing up technic parameters, ensure the program file has been loaded.

To back up technic parameters, do the following:

1. Click **Params**. **Params** dialog box pops up.
2. To enter **Technics** interface, click **Technics**:



3. Click **Add** in the lower left of list.
4. Set a group name for all technic parameters.

The system backs up all technic parameters.

After backing up technic parameters, according to the actual situation, do one of the following in the left list:

- To delete a group of technic parameters, select the target group name and click **Delete**.
- To apply your back-up parameters, select the target group name, click **Apply**.

  After the operation, reload the program file to make back-up parameters effective.

- To import the local group of technic parameters, click **Import**, select the target local file and click **Open**.

  After the operation, reload the program file to make back-up parameters effective.

- To export the local group of technic parameters, select the target group name, click **Export**, input the file name and click **Save**.

- To rename the group name, double click the target group name, input a new name and click **OK**.

4.10 Nest Parts

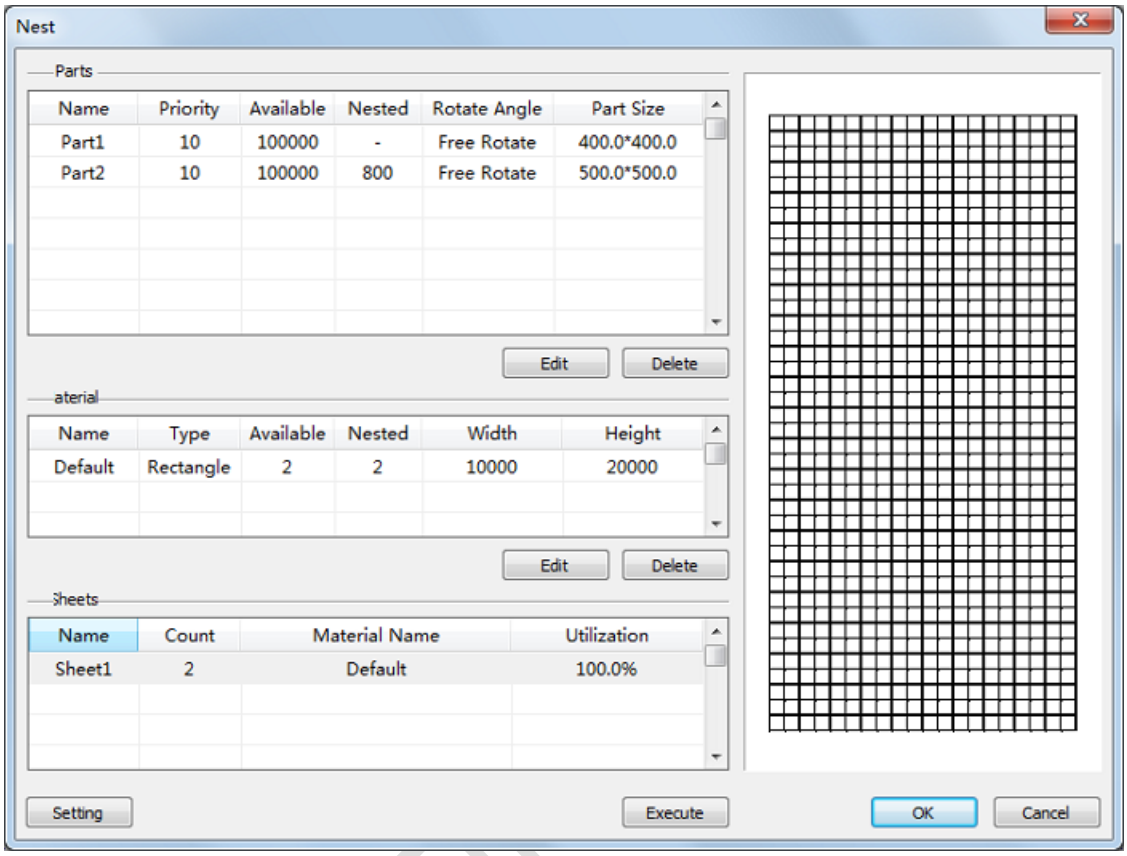
This operation is used to nest one or more kinds of parts in **NcEditor** based on part priority and the highest material utilization rate. Through previewing nesting effect and analyzing nesting results, including the sheet utilization rate and distribution of scrap material, material utilization and machining efficiency can be improved.

Before nesting parts, ensure you have bought the dongle from your supplier, and plugged it into the USB interface of your host or industrial personal computer (IPC).

To nest parts, do the following:

1. Select toolpaths in the drawing area.

2. Click **File** → **Nest**. **Nest** dialog box pops up:



3. To edit part information, select the target part in **Part** area and do one of the following:

   - Click **Edit** and modify the following:

     - Priority: On the premise of maximizing sheet utilization, the system prioritizes nesting toolpaths with higher priority.
     - Part Quantity: Total parts that you would like to nest.
     - Rotate Angle: Rotation way and rotation angle of parts. The default selection is free rotation.

   - Double click **Priority**, **Part Quantity** or **Rotate Angle** column for the target part and do modification.

4. Select the target sheet in **Material** area, click **Edit**, and modify the following:
   - Sheet Name
   - Sheet Quantity: Total sheets. And it should meet the following: `Sheet Quantity * Sheet Size > Part Quantity * Part Size`
   - Width: Actual sheet width.
   - Height: Actual sheet height.
5. Click **Setting**, set the following nesting parameters and click **OK**:
   - Material Margin and Part Space: The distance between parts and the edge of sheet, and distance between parts.
   - CoEdge: Parts with zero space share a line. It helps to reduce operations and save time.
   - Start Corner: The position to start nesting.
   - Nest direction: The nesting direction, including vertical direction and horizontal direction.

   You can preview nesting effect in the right area.
6. Click **Execute**. Nesting results (sheet utilization) show in **Sheet** area.

After nesting, do one of the following:

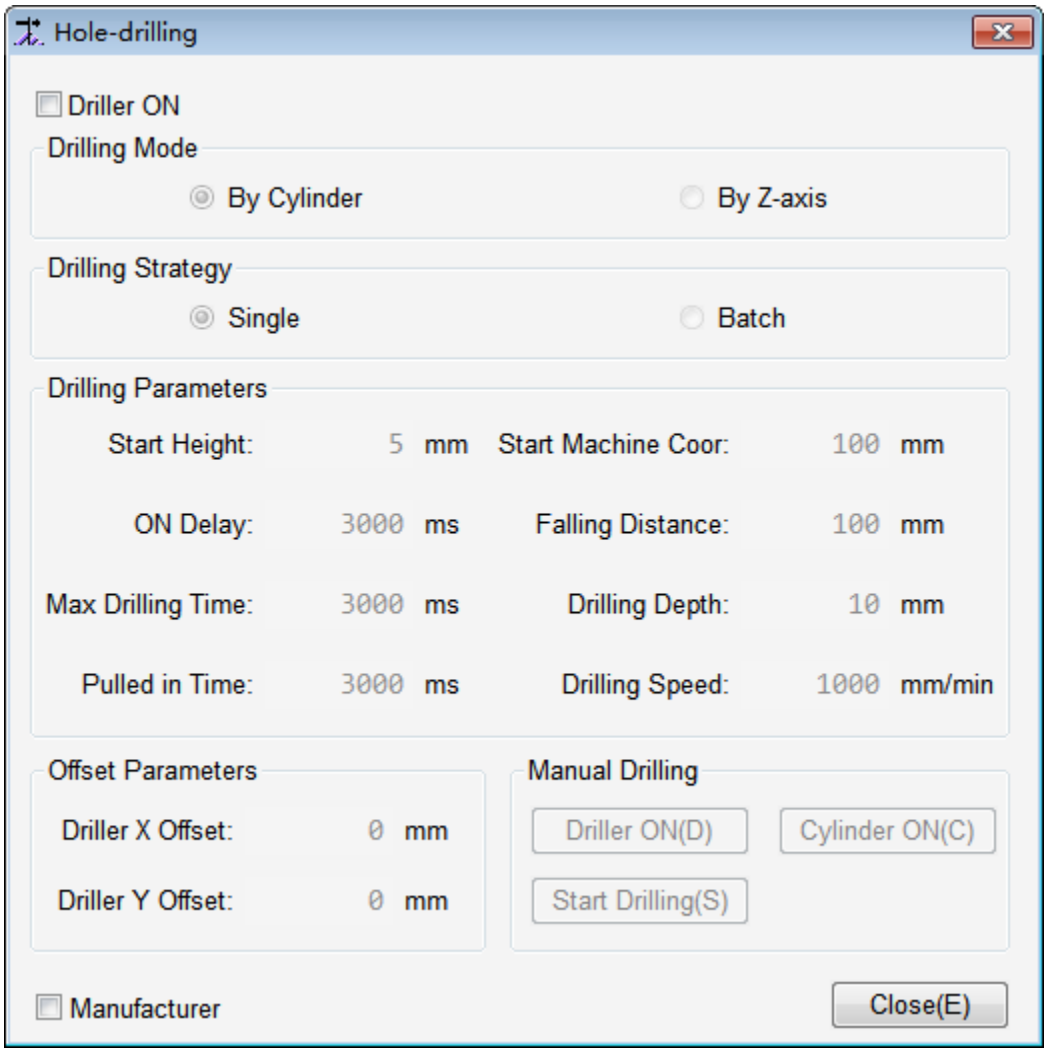
- If you are not satisfied with the results, reset part information, material information and nesting parameters and do nesting again.
- If you are satisfied with the nesting results, click **OK**. The nested parts show in the drawing area.

4.11 Use the Drill

This operation is used to bore a hole at the start point of object with the drill unit, so as to save time to cut objects and improve machining efficiency.

To use the drill, do the following:

1. In **Manual** mode, click **Drill**. **Hole-drilling** dialog box pops up:



2. Check **Driller ON** at the upper left corner.

3. Select a drilling mode in **Drilling Mode** area:

   - By Cylinder: The drill spindle is fixed on the cylinder while the cylinder on the Z-axis. And boring holes are realized by push-out actions of cylinder.

   - By Z-axis: The drill spindle is fixed on the cylinder while the cylinder on the Z-axis. After the cylinder pushes out the drill in position, boring holes are realized by up-down actions of Z-axis.

4. Select a drilling strategy in **Drilling Strategy** area:

   - Single: During cutting multiple objects, bore a hole in one object and then cut this object before executing the same operation for one another.
   - Batch: During cutting multiple objects, bore a hole in each object and then cut them in turn.

5. To bore a hole, do one of the following:

   - Automatic drilling:

     1. Set drilling parameters in **Drilling Parameters** area.

        About parameter details, see Drilling.

     2. Measure the X-axis / Y-axis offset between centers of the drill and the abrasive tube, check **Manufacturer** at the lower left corner, input the manufacturer password, and input the X-axis / Y-axis offset in **Offset Parameters** area.

     3. In **Auto** mode, click **Start**. The system starts boring a hole.

   - Manual drilling

     In **Manual Drilling** area. click **Driller ON** → **Cylinder ON** → **Start Drilling**. The system starts boring a hole.

Note: The system automatically switches to the drill unit during boring a hole and switches to the waterjet tool during cutting.

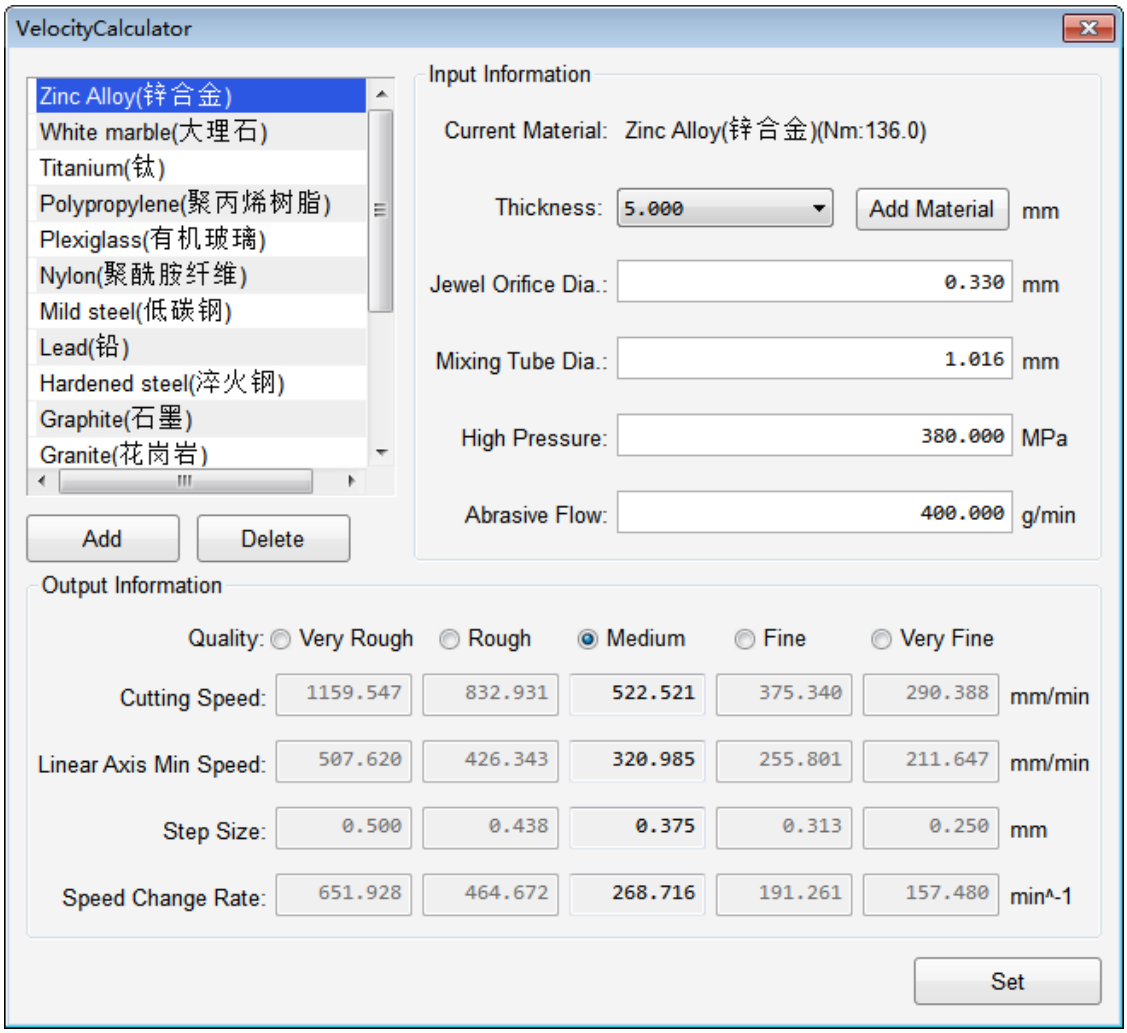
4.12 Use the Velocity Model Calculator

As an auxiliary tool for improving the machining effect, the velocity model calculator is supported by the relatively accurate theoretic math model. It can provide an reference for machining velocity, so as to solve such problems as the coarsely cut surface, over cutting at the corner and bad quality of small arcs.

This operation is applicable to the high-precise machining for metal, glass, thick sheets and the other special material.

To use the velocity model calculator, do the following:

1. Click **Help** → **Velocity Calculator**. **Velocity Calculator** dialog box pops up:



2. To add material, do one of the following:
   - Drag the slider bar and directly select the target material type.
   - Click **Add** and input the material name and machinability index.
   - **Note:** About the machinability index of the added material, contact us.
3. According the actual conditions, input material information in **Input Information** area:
   - Thickness: You can choose thickness 5mm, 10mm and 15mm, or customize thickness by clicking **Add Material**.
   - Jewel Orifice Dia.: The diameter of the jewel nozzle.
   - Mixing Tube Dia: The diameter of the abrasive tube.
   - High Pressure: The high pressure during machining.
   - Abrasive Flow: The flow of abrasive sand.
4. Select a quality level for cutting in **Output Information** area and click **Set**.
5. Click **Machine Tool** → **Optimize Cutting Speed**, check **Optimize Cutting Speed** in the pop-up dialog box and click **Reload**.

The speed calculated by the velocity model calculator takes effect during cutting.

4.13 Shoot with a Camera

This operation is used to precisely locate the workpiece and extract its contour, which is good for layout and quick cutting.

Before shooting with a camera, do the following for commissioning:

1. Correctly install a camera and light and do wiring.
2. Adjust the camera, including its height, shooting direction and brightness and focus.
3. Install the visual algorithm software (WeiHongCanon.exe) and **NcStudio V10 Waterjet Cutting Control System** whose version is 10.515.2 or above.
4. Plug the dongle with the visual algorithm into the USB interface of your host.

To shoot with a camera, do the following:

1. Do camera settings.
2. Extract the contour.
3. Use the gallery and lay out objects.

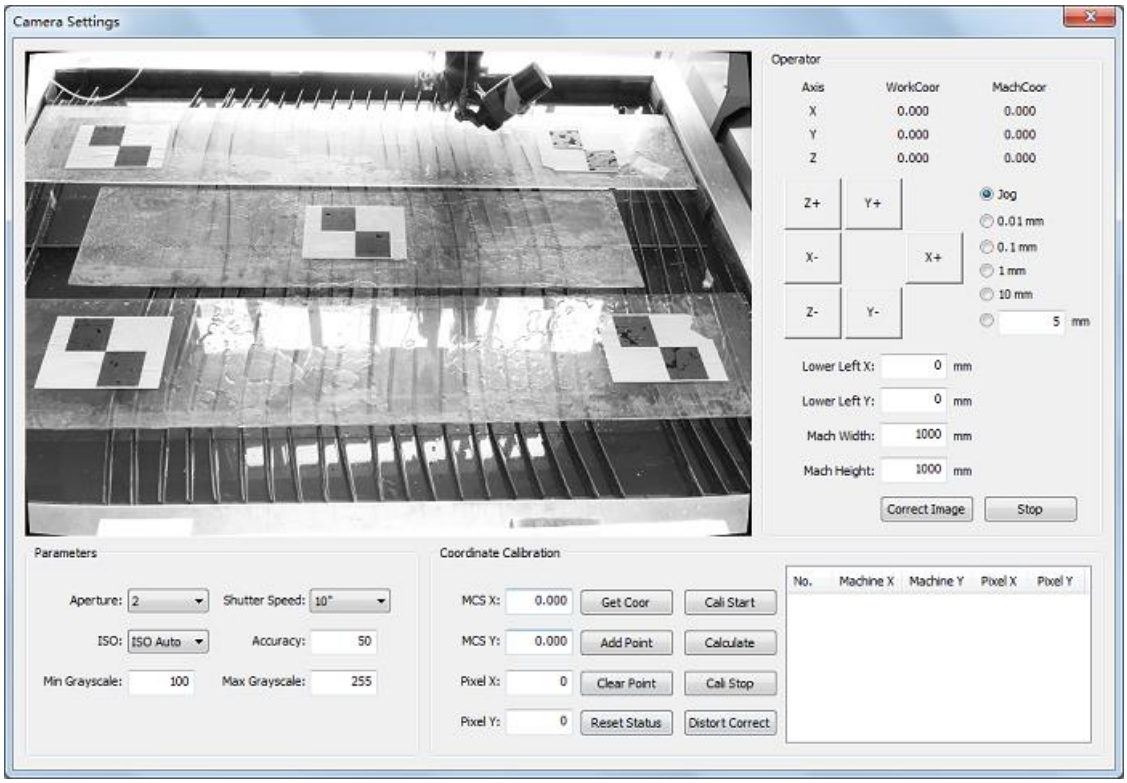
4.13.1 Do Camera Settings

This operation is used to set the positioning parameters for the camera, and ensure shooting goes on well.

To do camera settings, do the following:

1. In **NcStudio**, click **Machine Tool** → **System Parameters**. **System Parameters** dialog box pops up.
2. Click **Param** → **Other** → **Manufactory** and input the manufacturer password.
3. Find and set positioning parameters.

   About parameter details, see Positioning.
4. In **NcEditor**, click **Draw** → **Camera Settings**. **Camera Settings** dialog box pops up:



5. Set camera parameters.
6. Calibrate coordinates.
7. Correct the image.

4.13.1.1 Set Camera Parameters

In **Camera Settings** dialog box, set the following in **Parameters** area:

- Aperture: The larger the value, the smaller the aperture. At this time, less light gets through in unit time, the shoot becomes darker and the field in your shoot deeper.

  Range: 2~22

  Recommended value: 22

- Shutter speed: The larger the value, the slower the shutter speed. At this time, more light gets through in unit area and the shoot becomes brighter.

  Range: 1/8000~10

- ISO: It measures the sensitivity of the image sensor. The larger the value, the more sensitive the image sensor. At this time, the shoot becomes brighter but noise becomes more.

  Recommended value: 100

- Accuracy: The larger the value, the fewer the contour detail.

  Range: 50~255

- Min gray scale: It will influence the contour extracting effect. If the background is black and the workpiece is white, set **Max Grayscale** to **255** and adjust the minimum gray scale.

- Max gray scale: It will influence the contour extracting effect. If the background is white and the workpiece is black, set **Min Grayscale** to **0** and adjust the maximum gray scale.

4.13.1.2 Calibrate Coordinates

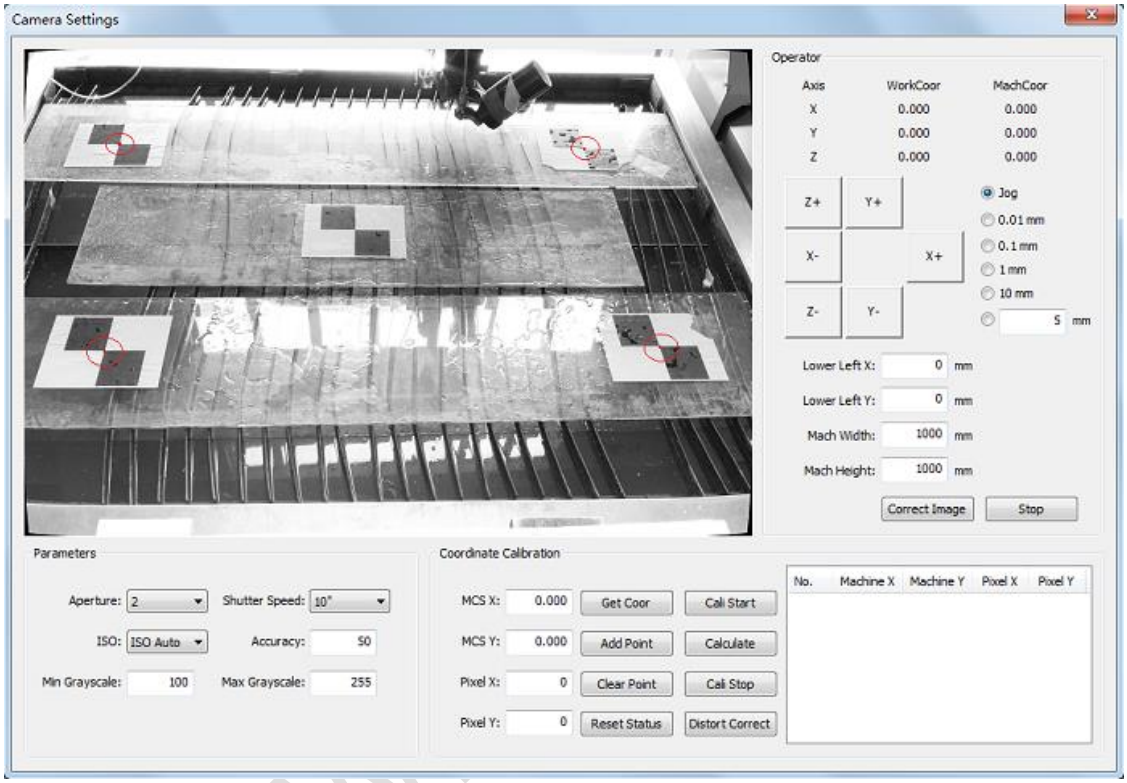
This operation is used to establish the corresponding relation between the calibrated coordinates and actual coordinates on the machine tool.

Please carefully execute each step. Otherwise, calibration precision will be affected.

4.13.1.2.1 Execute Calibration

To execute calibration, in **Calibration** area of **Camera Settings** dialog box, do the following:

1. To shoot the machine tool and get four calibration points (red points), place four calibration blocks on the machine tool, click **Cali Start**:



2. To get machine coordinates of the target calibration point, aim this calibration point with the laser sensor and click **Get Coor**. The system automatically fills in machine coordinates into **MCS X** and **MCS Y** input boxes.

   ```
   Coordinates of the Calibration Point = Current Coordinates - Offset betwe
   en the Tool Head and Laser Sensor
   ```

3. To get pixel coordinates of the corresponding calibration point, move the mouse pointer on this calibration point, scroll the mouse wheel to zoom in the point, right click until finding the precise position of the calibration point. The system automatically fills in pixel coordinates into **Pixel X** and **Pixel Y** input boxes.

4. To add the current calibration point, click **Add Point**. The system automatically fills in its calibrating coordinates into the table in the lower right.

5. To add at least four calibration points, repeat above steps.

   If added calibration points are incorrect, click **Clear Point** and add them again.

6. After getting four groups of calibrated coordinates, click **Calculate**.

After calculation ends, click **Cali Stop**. The corresponding relation between travels of the pixel coordinate and machine coordinate is established.

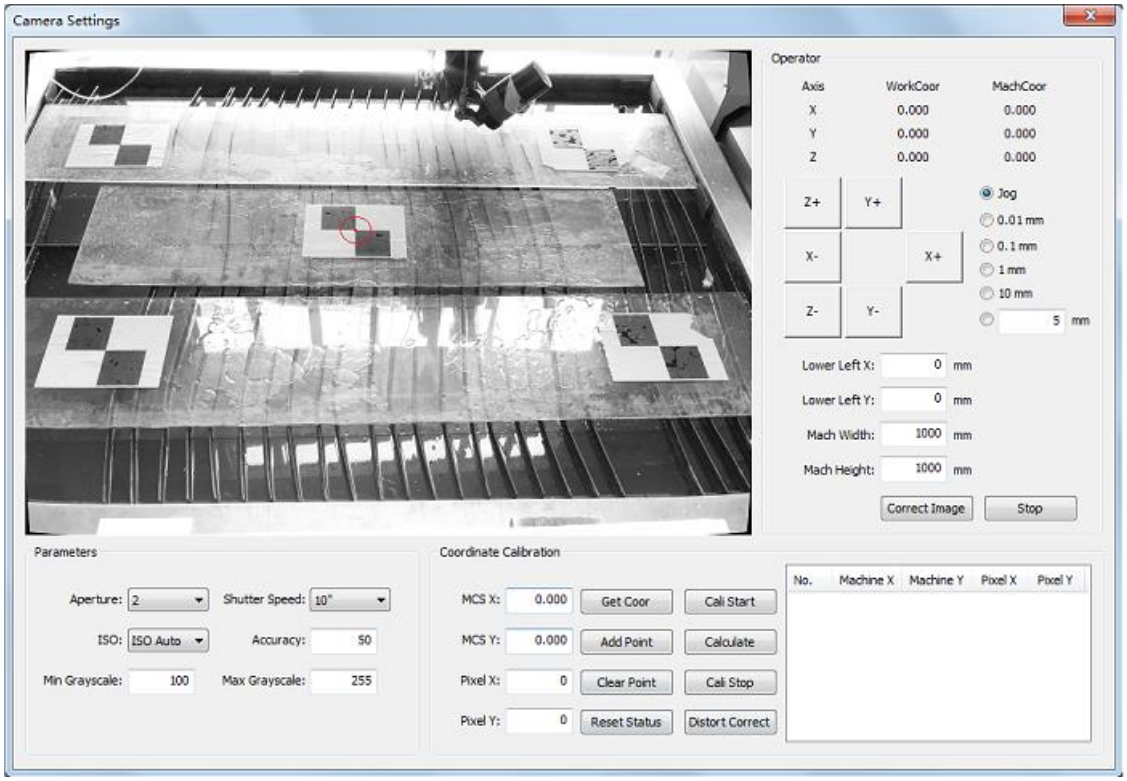
Note: If exceptions occur, such as power is off, the position of calibration block changes, the camera shakes, during calibration,please click **Reset Status** and calibrate coordinates again.

4.13.1.2.2 Verify the Calibrating Result

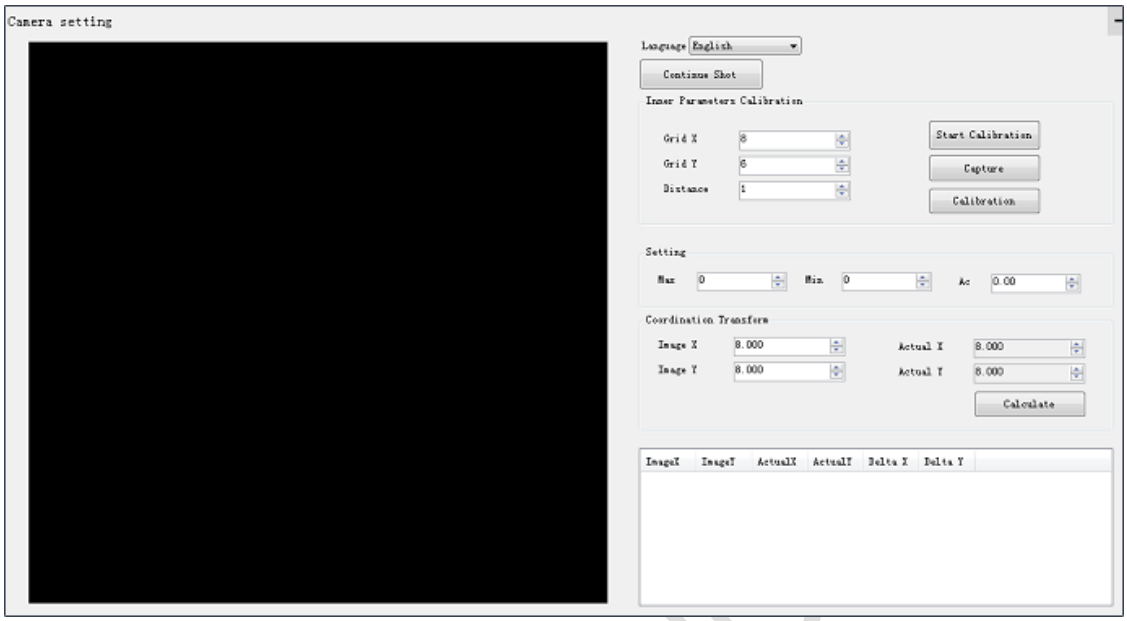
This operation is used to judge if the positioning error is within 1mm~2mm.

To verify the calibrating result, do the following:

1. Follow step 1 ~ 3 of **Execute Calibration**, get machine coordinates and pixel coordinates of one another calibration point:



2. Click **Distort Collect**. **Distort Collect** dialog box pops up:



3. In **Coordination Transform** area, input pixel coordinates of this calibration point into **Image X** and **Image Y** input boxes.

4. Click **Calculate**. **NcEditor** automatically calculates actual machine coordinates.

5. Compare actual machine coordinates and the machine coordinates from step 1, and judge if the positioning error is within 1mm~2mm.

   If not, calibrate coordinates again.

4.13.1.3 Correct the Image

In **Operator** area of **Camera Settings** dialog box, to correct the image, set the following and click **Correct Image**:

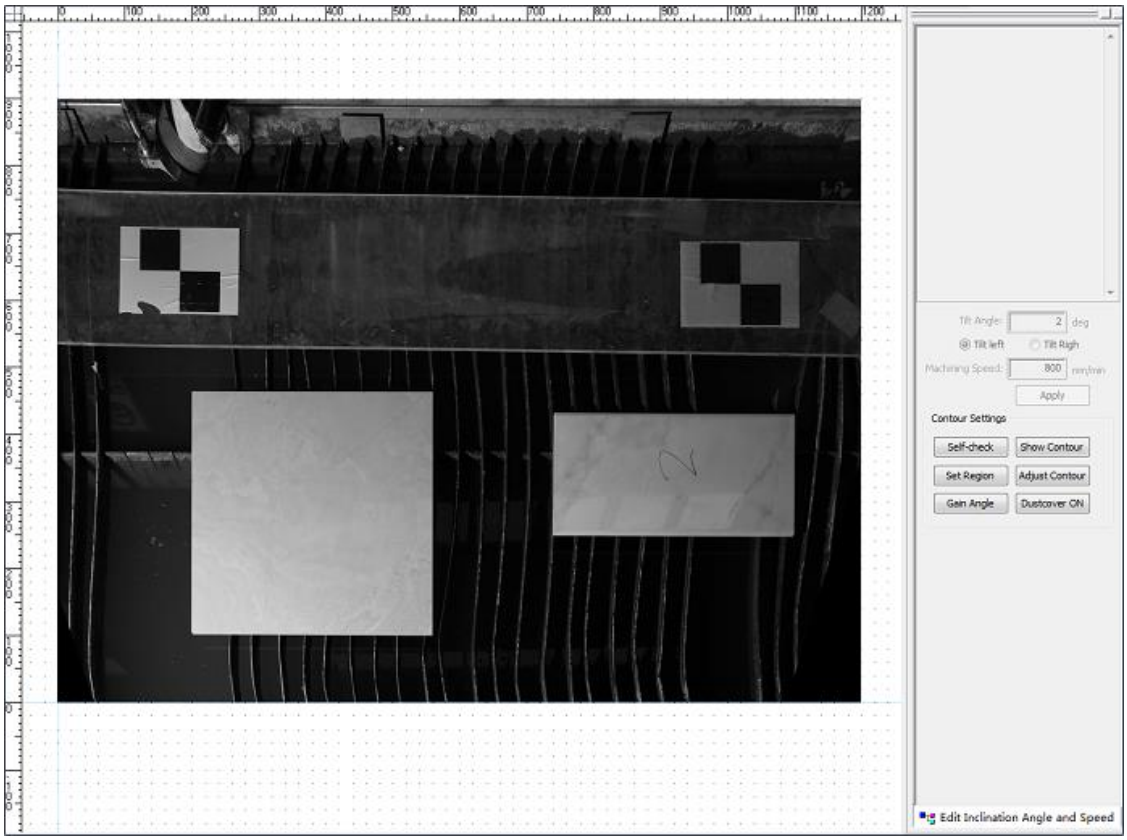
- Lower Left X / Y: The lower limit of workbench travel.
- Mach Width: The length of workbench in X-axis direction.
- Mach Height: The length of workbench in Y-axis direction.

4.13.2 Extract the Contour

This operation is used to extract the contour of workpiece, so as to get the tilt angle of workpiece relative to the workbench. And after setting the type of objects on the workpiece, objects can be rotated to the same angle.

To extract the contour, do the following:

1. In **NcEditor**, click **Dustcover On**. The cylinder of dust cover and light turns on.
2. Click **Self-check**. The camera automatically shoots and import the image into **NcEditor** according to proportional scale 1:1:



3. To select the target region to recognize, do one of the following:
   - Click **Set Region**. Valid regions in the image are automatically selected.
   - Every time you click **Self-check**, only the workpiece contour within the valid region can be extracted.
   - Click **Contour Correct**, manually select regions to recognize and extract the workpiece contour.

4. Click **Show Contour** / **Hide Contour**. The extracted workpiece contour shows / hides in the image.

If you are not satisfied with the result, to extract the contour again, click **Gain Angle** and select two points on the lower edge of workpiece (that is parallel to X-axis). The system automatically extracts the workpiece contour again.

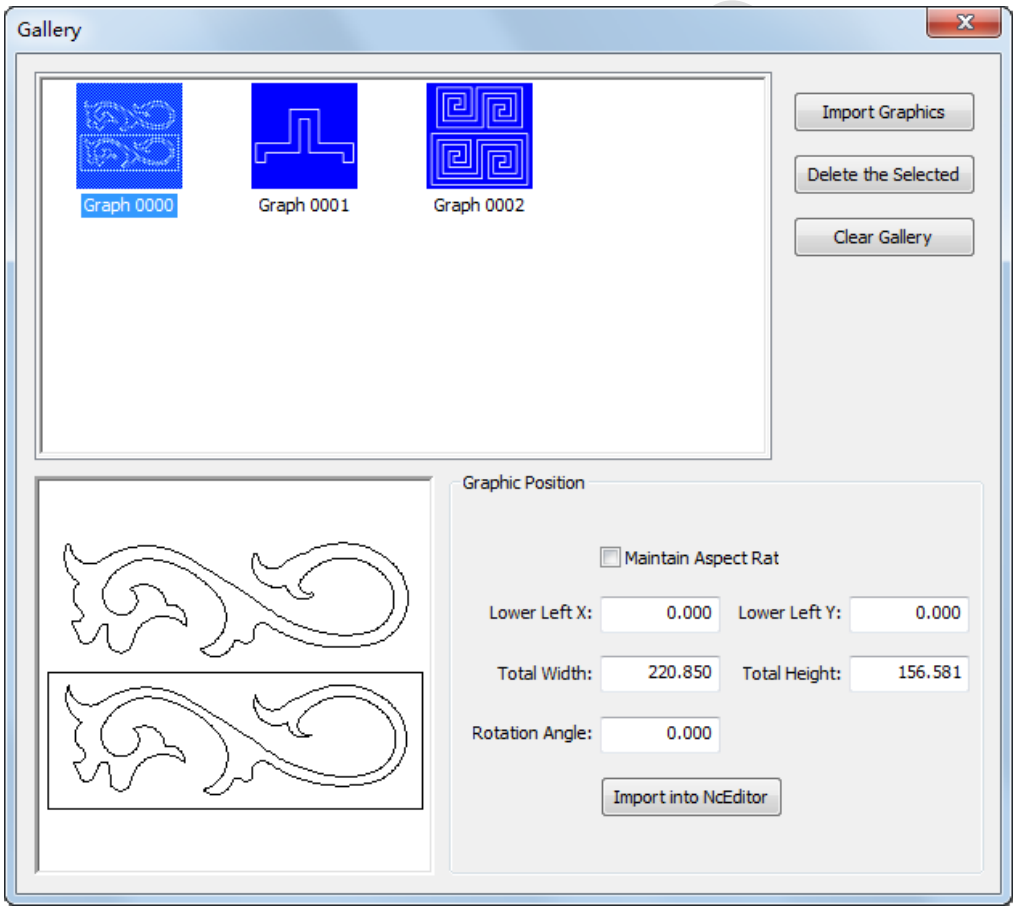
4.13.3 Use the Gallery and Lay out Objects

This operation is used to import program files into the gallery of **NcEditor** and lay out these objects on the workpiece.

The supported format of program file includes `G`, `NC` and `DXF`.

To use the gallery and lay out objects, do the following:

1. In **NcEditor**, click **Draw** → **Custom Gallery**. **Gallery** dialog box pops up:



2. To import the program file of target object into the gallery, click **Import Graphics**.

3. In **Graphic Position** area, set position parameters for the object and click **Import to NcEditor**.

4. In **NcEditor**, manually lay out the target object, save the program and start machining.

In **Gallery** dialog box, according to the actual situation, do one of the following:

- To delete the object in the gallery, select the target object and click **Delete the Selected**.
- To clear all imported objects, click **Clear Gallery**.

4.14 Adjust Five-axis Tool Head Errors

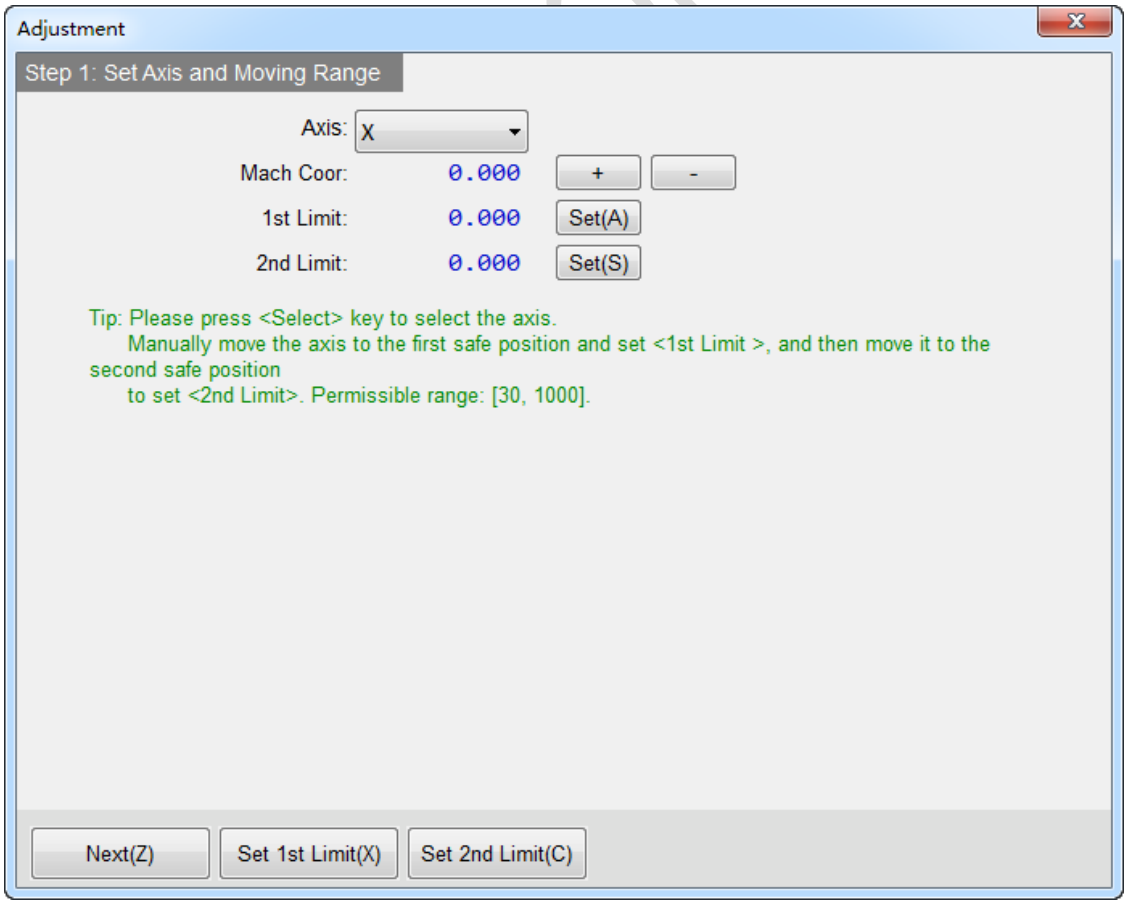
This operation is used to reduce errors and improve cutting precision.

Before adjusting five-axis tool head errors, do the following:

- Return A-axis and C-axis to machine origin.
- Prepare an extended testing rod and a dial indicator with a flat head for measuring.

To adjust five-axis tool head errors, do the following:

1. Click **Machine Tool** → **Five Axis Error Compensation**. **Five Axis Error Compensation** pops up:



2. **Optional:** If it is not your first time to adjust five-axis tool head errors, uncheck **Enable Compensation** at lower left corner and click **Yes**.
3. Click **Guide** at the upper right corner. **Error compensation guide** dialog box pops up.
4. Execute adjustment according to steps in **Error compensation guide** dialog box.
5. To make the compensation result effective, check **Enable Compensation** at the lower left corner and click **Yes**.

5 Common Operations

Through this section, you can quick know common operations of **NcStudio V10 Waterjet Cutting Control System**.

Common operations include the following:

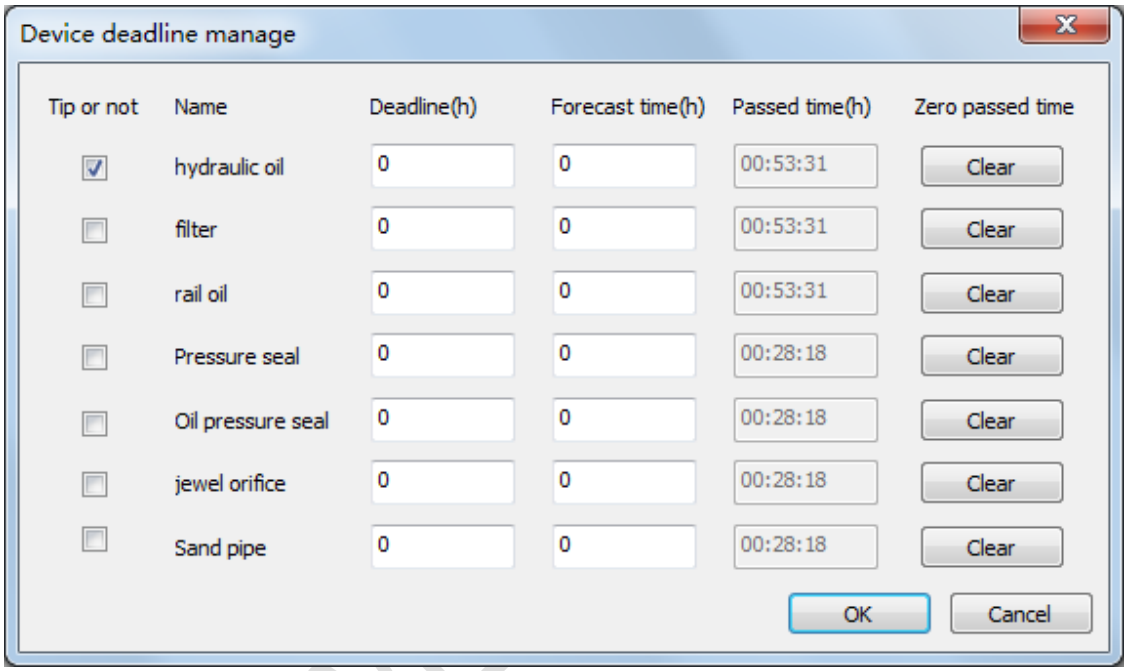
- Manage Wearing Parts
- Rotate or Mirror Machining Objects
- Execute Breakpoint Resume
- Control Abrasive Flow
- Set Valve On and OFF Order
- Use Customized Commands
- Use NcGateway
- Check Machining Information
- Check Ports
- Check Logs

5.1 Manage Wearing Parts

This operation is used to count the service life and used time of wearing parts, and show a prompt before the service life runs out, so as to help to replace wearing parts in time.

To manage wearing parts, do the following:

1. Click **Machine Tool** → **Deadline Manage**. **Device deadline manage** dialog box pops up:



2. Set the following parameters:
   - Deadline: The service life of each wearing part.
   - Forecast time: The time to show a prompt that the service life will run out.
3. Select wearing parts that need to show a prompt in **Tip or not** column:
   - When the set forecast time is up, the system sends a yellow warning in CNC status bar.
   - When the set deadline is up, the system sends a red error in CNC status bar.
4. **Optional:** To reset the used time of wearing parts after replacing them, click **Clear** in **Zero Passed Time** column.

   The time in **Passed time** column is accumulated and cannot automatically be cleared even after replacing wearing parts. So, please manually clear it.

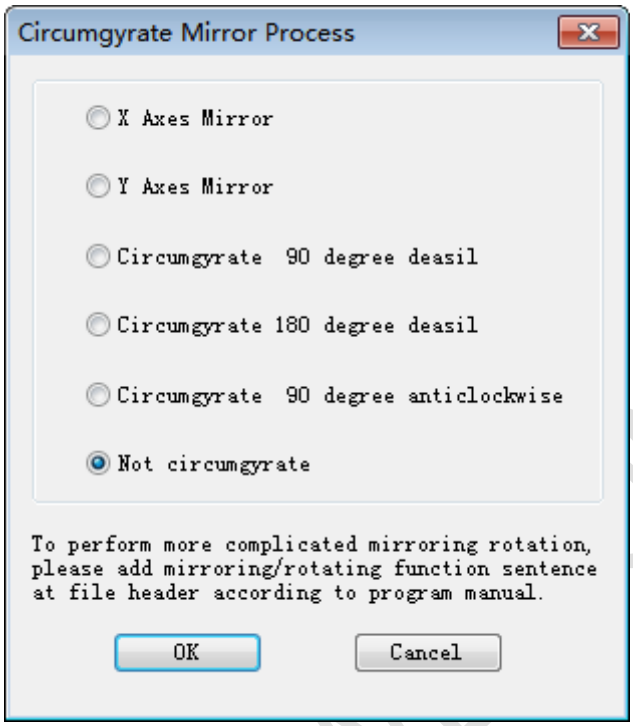
Note: Alternatively, you can manage wearing parts through the **WebChat** version of **NcCloud**.

5.2 Rotate or Mirror Objects

This operation is used to rotate or mirror objects for a program file.

To rotate or mirror objects, do the following:

1. Click **Operate** → **Circumgyrate Mirror Process**. **Circumgyrate Mirror Process** dialog box pops up:



2. To select a rotation or mirroring strategy, do one of the following:
   - To mirror objects along X-axis, select **X Axes Mirror**.
   - To mirror objects along Y-axis, select **Y Axes Mirror**.
   - To rotate objects 90º in CW direction, select **Circumgyrate 90 degree deasil**.
   - To rotate objects 180º in CW direction, select **Circumgyrate 180 degree deasil**.
   - To rotate objects 90º in CCW direction, select **Circumgyrate 90 degree anticlockwise**.

   No circumgyrate means the system does not rotate and mirror objects.

3. Click **OK**.

The system automatically executes the selected operation for the loaded program file.

5.3 Execute Breakpoint Resume

This operation is used to resume machining on the breakpoint if power off or E-stop occurs during machining, so as to save the machining time.

Before executing breakpoint resume, ensure workpiece coordinates are accurate. Otherwise, please return to the machine origin first.

In the menu bar, click **Operate** → **Breakpoint Resume**. The cutting resumes from the position where it stopped last time.

5.4 Control Abrasive Flow

This operation can be divided into the following:

- Control abrasive flow by analog quantity

  It is used to control abrasive flow by adjusting the analog voltage output of Lambda terminal board that connects to the motor of abrasive valve.

- Control abrasive flow by spindle

  It is used to control abrasive flow by bus control system sending pulse to control the speed of stepping motor that connects to the abrasive valve.

  It has more precise control and higher cutting precision.

5.4.1 Control Abrasive Flow by Analog Quantity

Before controlling abrasive flow by analog quantity, ensure parameter **Abrasive Value Control Mode** has been set to **0**.

The relation among the actual output voltage of abrasive valve, the set output voltage and rate of analog output voltage is as follows:

```
Actual Output Voltage = Set Output Voltage × Rate of Analog Output Voltage
```

To control abrasive flow by analog quantity, do the following:

1. Click **Machine Tool** → **System Parameters**. **System Parameters** dialog box pops up.
2. Click **Param** → **Operation** → **Manufacturer**, input the manufacturer password, find and set the parameter **Abrasive Value Output Voltage**.
3. To adjust the rate of analog output voltage, in **Auto** mode, drag **S. R. (N)** slider in machine tool control bar.

5.4.2 Control Abrasive Flow by Spindle

Before controlling abrasive flow by spindle, ensure the following:

- The manufacturer parameter **Abrasive Value Control Mode** has been set to **1**.
- The drive subdivision is the same with the default value of internal parameter **Pulse Output**.

The relation among the actual motor speed, the set motor speed and the rate of motor speed is as follows:

```
Actual Motor Speed = Set Motor Speed * Rate of Motor Speed
```

To control abrasive flow by spindle, do the following:

1. Click **Params** → **Machine Tool**, check **Manufactory** at lower left corner, find and set the parameter **Stepper Motor Speed**.
2. To adjust motor speed, in **Auto** mode, drag **S. R. (N)** slider in machine tool control bar.

5.5 Customize Valve ON / OFF Order

This operation is used to customize the order of turning on / off valves according to your need.

To customize valve ON / OFF order, do one of the following:

- Chick **Machine Tool** and do the following:
    1. For valve ON order, click **Open Valves Order**, and select an order.
    2. For valve OFF order, click **Close Valve Order**, and select an order.
- Click **Machine Tool** → **Open/Close Booster** / **Open/Close H/L** / **Open/Close H.p.** / **Open/Close Cut** / **Open/Close Abrasive**.

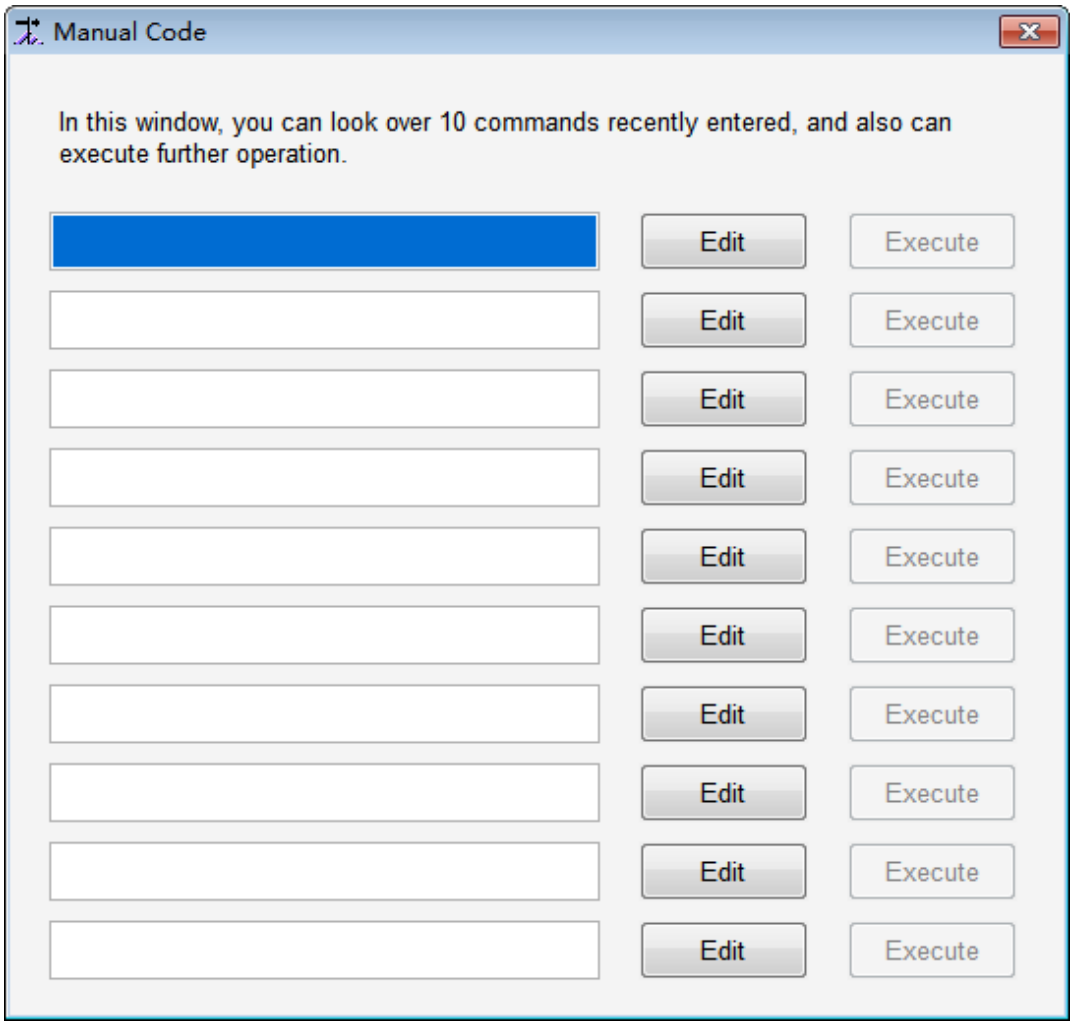
During machining, the system turns on / off these valves according to the set order.

5.6 Use Customized Commands

This operation is used to input and execute at most 10 simple commands, so as to make machine tools move quickly or execute simple operations.

To use customized commands, do the following:

1. Click **Window** → **Manual Code**. **Manual Code** dialog box pops up:



2. Click **Edit**, input commands in the pop-up dialog box and click **OK**.

   All commands that have been input will be automatically saved for later use.

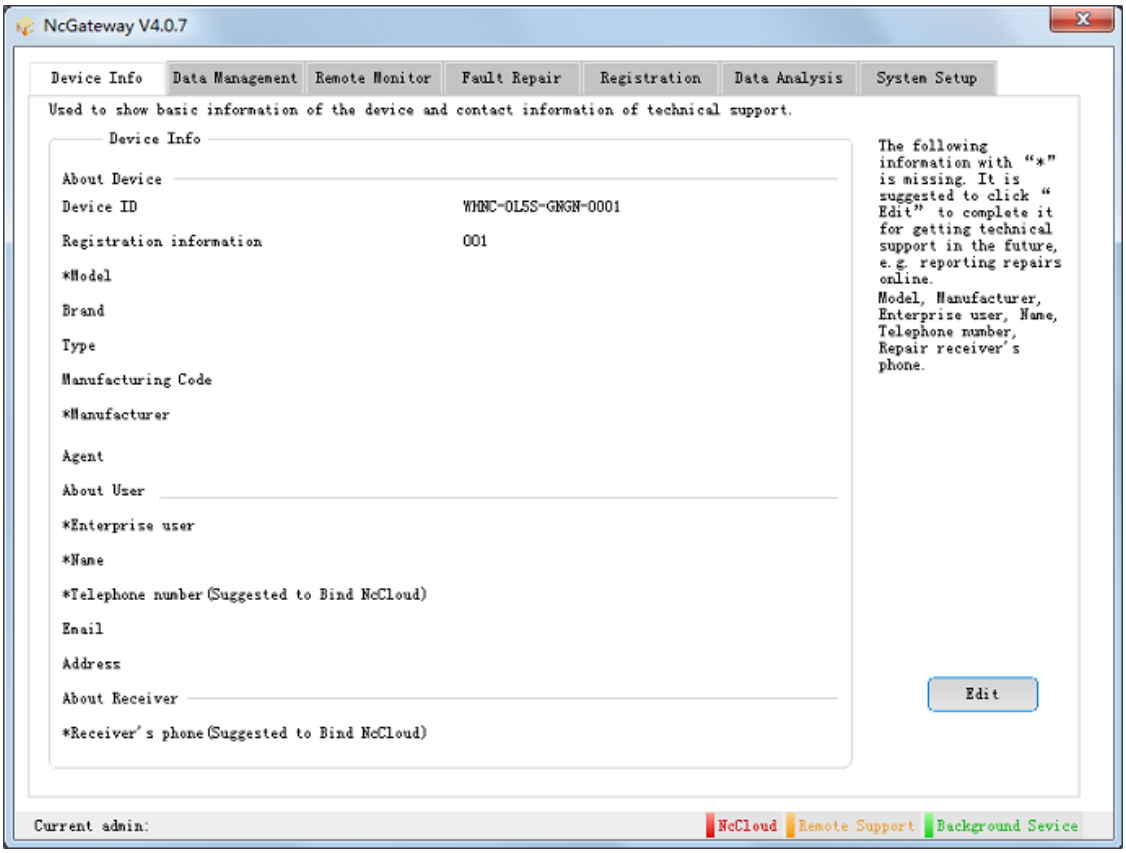
3. To execute the target command, click **Execute** after the corresponding command.

5.7 Use NcGateway

This operation is used to manage the CNC system by **NcGateway**, which is a component of Weihong IIoT solution and can provide data interface and communication protocols.

About **NcGateway**, see NcGateway for details.

To use **NcGateway**, click **Help** → **WeihongCloud**. **WeihongCloud** dialog box pops up:



- Device information: It is used to check the basic information of the host and telephone number of technical support and edit its basic information.
- Data management: It is used to import an authorization file for enabling MTConnect, OPC UA protocols and local data backup.
- Remote monitoring: It is used to remotely monitor the running data and control the CNC system.
- Online repair: It is used to report a repair to the maintainer and check the repair record online.
- Device registering: It provides the QR code, device number and registration information for registering the CNC system.

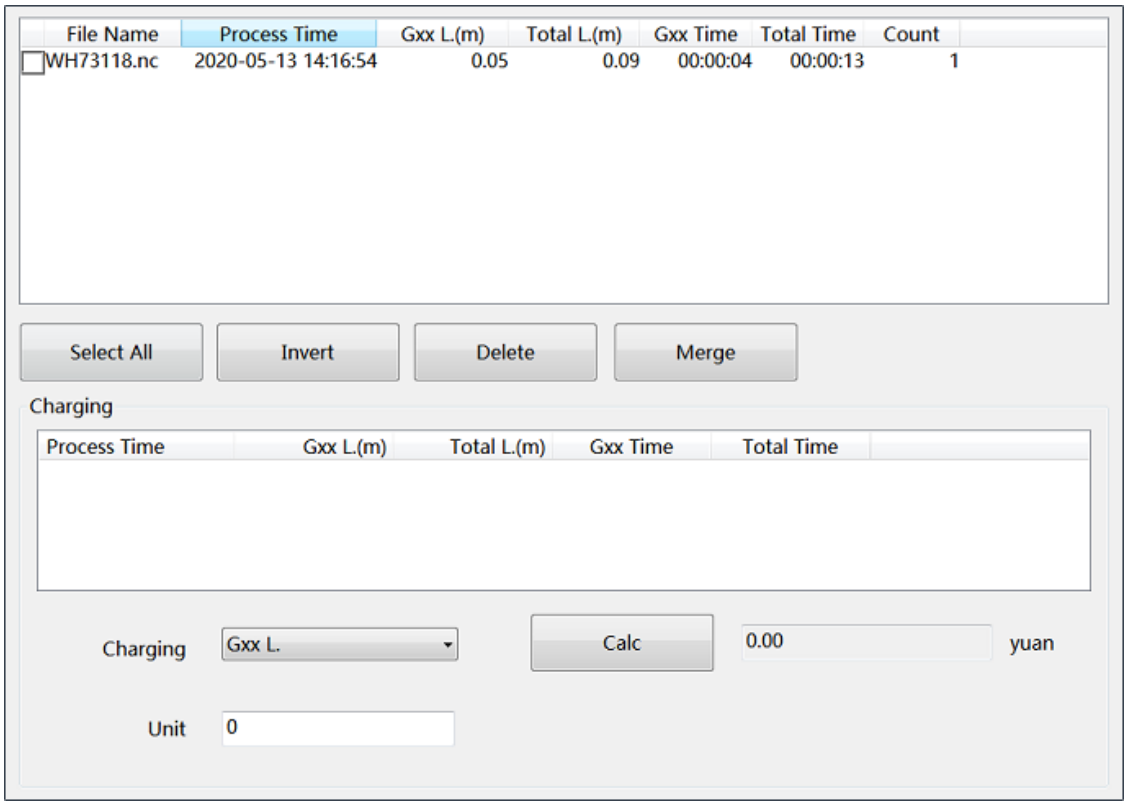
- Data analysis: It is used to check the yesterday's utilization rate and OEE, distribution of today's working time (running, idle, E-stop and unavailable) and counts of today's error, warning and smart information.
- System setup: It is used to set the usage habit, IP or domain name of **xFactory** and related parameter of sampling data.

5.8 Check Machining Information

This operation is used to check machining information, including start time, machined time, cutting length and so on, and calculate machining charge if necessary.

To check machining information, do the following:

1. To enter **Info** window, click **Info**:



2. Check the following machining information:
   - Process Time: Start time
   - Gxx L.: Cutting length
   - Total L.: Machining length
   - Gxx Time: Cutting time
   - Total Time: Machined time
   - Count: Number of units

3. **Optional:** To calculate machining charge, do the following:
    1. Select at least a program file.
    2. Select a charge type from the following:
        - Gxx L.
        - Total L.
        - Gxx Time
        - Total Time
    3. Input unit price (Unit: yuan).
    4. Click **Calc** to start calculation.

5.9 Check Port Information

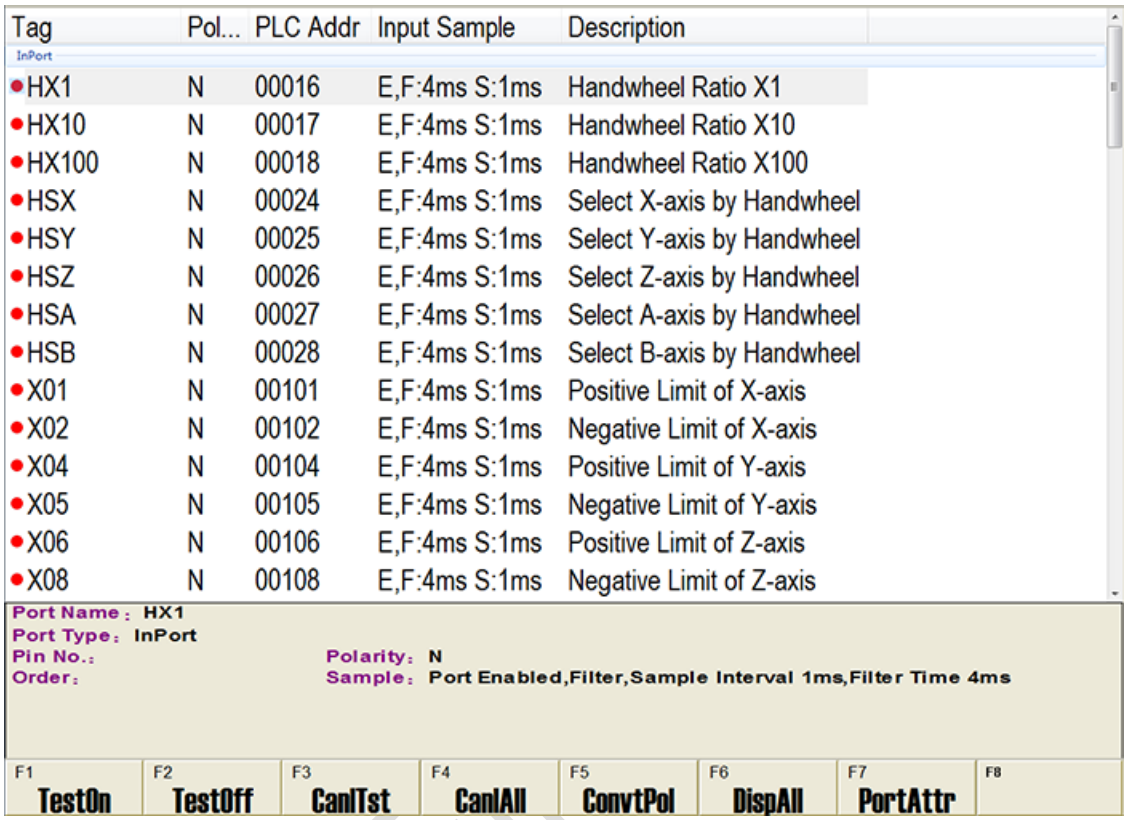
This operation is used to check port information, conduct a simulation test and modify polarity for ports and so on.

The relation between the status of the machine tool and IO ports is as follows:

- Input port: ● means no signal while ● means having a signal.
- Output port: ○ means no signal while ○ means having a signal.

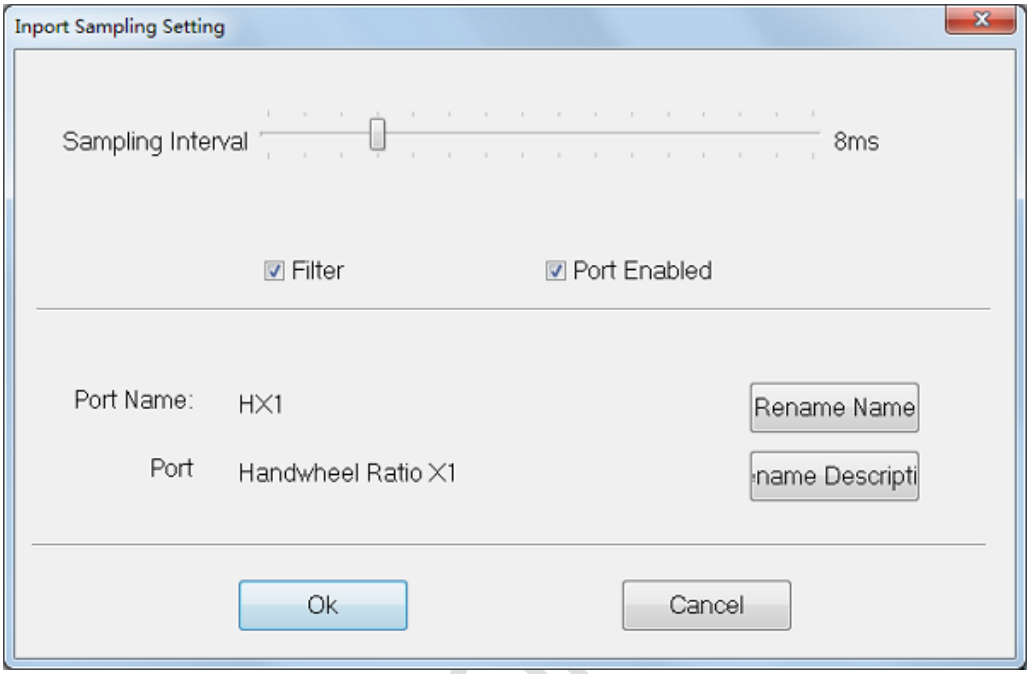
To check port information, do the following:

1. To enter **IO PORT** window, click **IO PORT**:



2. Select the target port and do one of the following:
   - To conduct a simulation test, click **Test On**:
     - Input port:
     - Output port:
   - To close a simulation test, click **Test Off**:
     - Input port:
     - Output port:
   - To cancel a simulation test, click **Canl Tst**.
   - To cancel all simulation tests, click **Canl All**.
   - To modify the port polarity, click **Convt Port**.
   - To show all supported ports, click **Disp All**.

– To set the port attribute, click **Port Attr** and do one of the following in **Inport Sampling Setting** dialog box :



–

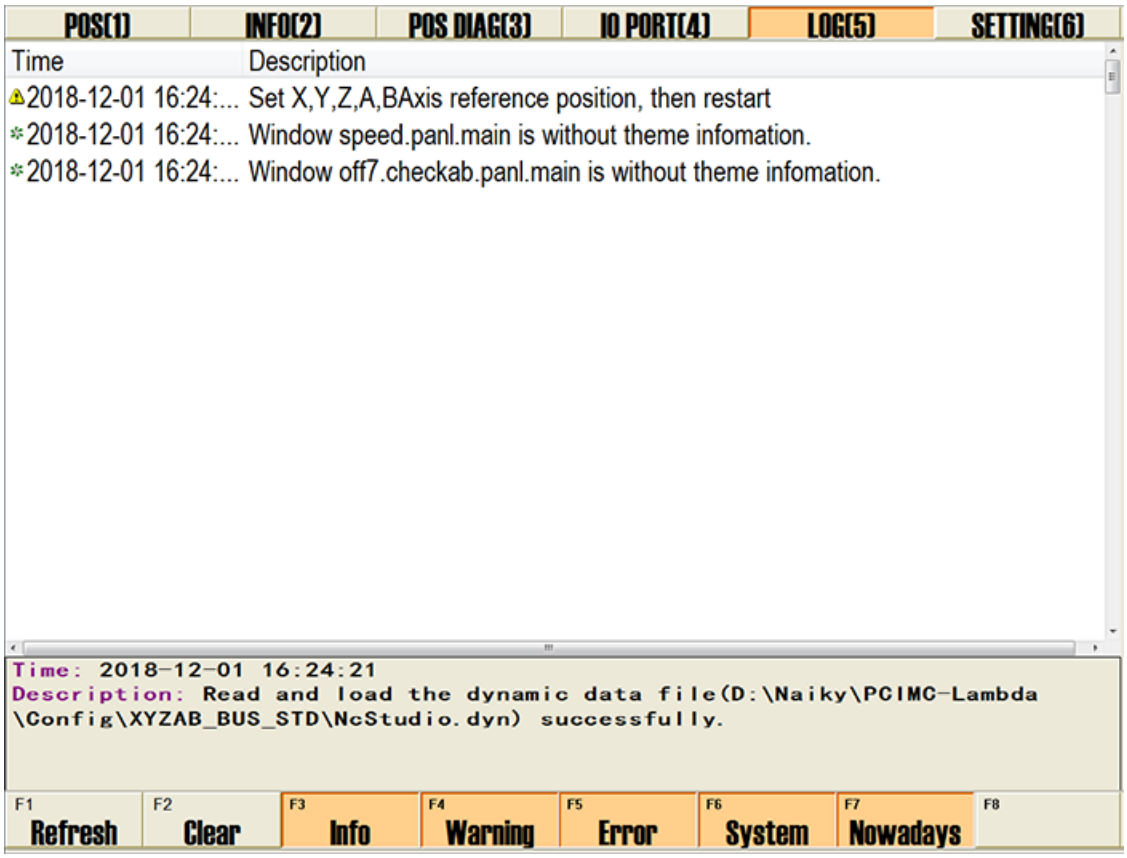
- Set the sampling interval.
- Enable **Filter** and **Port Enabled**.
- Rename the port.
- Modify the port description.

5.10 Check Logs

This operation is used to check different types of logs that record important operations and system events, including logs after this startup and historical logs.

To check logs, do the following:

1. To enter **LOG** window, click **LOG**:



2. To check different types of logs, do one of the following:
   - Click **Info**. Logs with icon about running status show in the window.
   - Click **Warning**. Warnings with icon show in the window.
   - Click **Error**. Errors with icon show in the window.
   - Click **System**. System logs with icon show in the window.
   - Click **Nowadays** and click **Info** / **Warning** / **Error** / **System**. The corresponding logs that generate after this startup show in the window.

   These buttons are clicked by default.

In **LOG** window, you can do one of the following:

- To refresh logs, click **Refresh**.
- To clear all logs, click **Clear**.

  Note: Please regularly clear logs. Otherwise, too many log files will slow down the system.

6 Object Edition

Through this operation, you can know auxiliary drawing operations in **NcEditor**:

- Bridge Objects
- Use Micro Joint
- Use Chamfer
- Set the Groove Direction
- Set Tool Compensation
- Set a Lead Line
- Reverse the Lead Line
- Sort the Machining Order
- Use Vertical Plunge
- Set the Tilt Angle and Cutting Speed

6.1 Bridge Objects

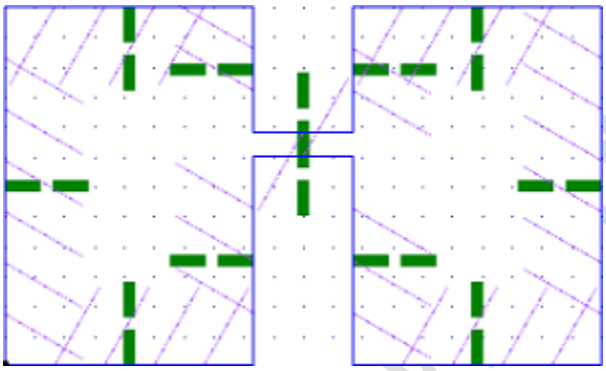
This operation is used to connect two objects close in distance with a bridge, so as to avoid unnecessary operations, such as lifting the tool and turning on / off valves, and reduce wear of wearing parts, like the high-pressure valve and oil pump. As a result, machining efficiency can be improved.

To bridge objects, do the following:

1. In **NcEditor**, click  and select at least two target objects.
2. Click 

in the tool bar. **Bridge Connect** dialog box pops up.
3. Set bridging width and click **OK**.
4. In the drawing area, click the internal area of an object, move the mouse to the internal area of the other object.

The bridging result shows in the drawing window:

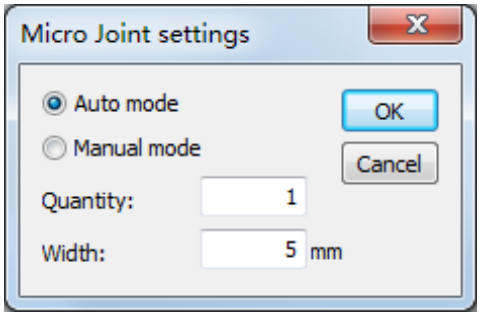


6.2 Use Micro Joint

This operation is used to set reserved segments that are not cut during machining, so as to connect the cutting part with the material and avoid the part falls into the water sink.

To use micro joint, do one of the following:

1. In **NcEditor**, click  and select the target object.
2. To open **Add micro joint mode setting** dialog box, do one of the following:
   - In the menu bar, click **Object** → **Micro Joint**.
   - In the drawing area, right click and select **Micro Joint** in the pop-up menu.



3. Select the mode of micro joint:
   - Auto mode: Set the following and click **OK**:
     - Quantity: The quantity of reserved segments on the object.
     - Width: The width of reserved segments on the object.
   - Manual mode: Set the width of reserved segments, and select the position of reserved segments on the object in the drawing area.

6.3 Use Chamfer

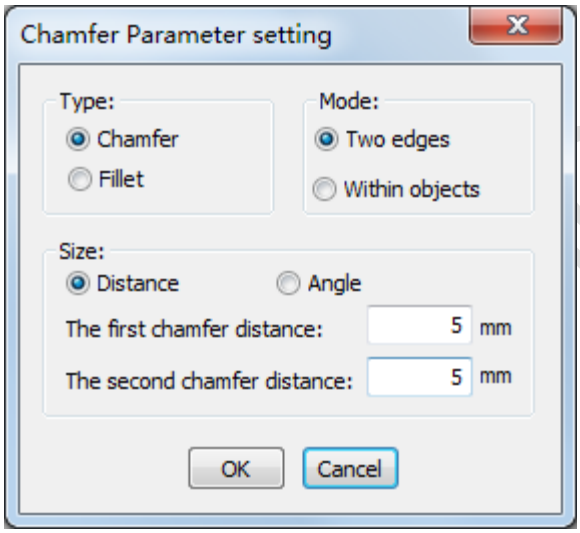
This operation is used to cut the arris into a line or an arc, so as to remove burrs from parts caused by machining, and help part assembly.

Before using chamfer, ensure the following:

- The target object has at least one arris formed by two lines.
- Standard 5-axis machining has been enabled.

To use chamfer, do the following:

1. In **NcEditor**, to open **Chamfer Setting** dialog box, do one of the following:
   - In the menu bar, click **Object** → **Chamfer**.
   - In the drawing area, right click and select **Chamfer** in the pop-up menu.



2. Select a chamfer type:
   - Chamfer: to cut the arris into a line.
   - Fillet: to cut the arris into an arc.
3. Select a chamfer mode:
   - Two edges: Add chamfer for two selected adjacent edges.
   - Within objects: Add chamfer for all angles of all objects by batch.

4. To set the chamfer size, according to the chamfer type, do one of the following:
   - Chamfer: Select **Distance** and set chamfer distance of two edges, or select **Angle** and set chamfer length and angle of the first edge.
   - Fillet: Input the chamfer radius.
5. Click **OK**. The cursor shape changes.
6. According to the chamfer mode, do one of the following:
   - Two edges: Select two target edges in the drawing area.
   - Within objects: Select the target object. The system automatically adds chamfer for all included angles.

6.4 Set the Groove Direction

This operation is used to set the tilt angle for the cut surface by setting the groove direction of toolpath, so as to meet the requirement for the tilt angle during machining medallion technics.

It differs in the object type:

- For closed objects
- For unclosed objects

After setting the groove direction, to check the groove direction, click  in the tool bar of **NcEditor**.

6.4.1 For Closed Objects

To set the groove direction for closed objects, do the following:

1. In **NcEditor**, click  and select the target closed object.
2. Click 

in the tool bar and select **Enable standard 5-axis machining** or **Enable Edit Inclination Angle**.
3. Do one of the following in the tool bar:
   - For **Enable standard 5-axis machining**
     - To change the groove direction, click 

.
     - To change the groove direction and valid region, click 

.
   - For **Enable Edit Inclination Angle**
     - To change the groove direction, click 

/ 

.
     - To change the groove direction and machining direction, click 

/ 

.
     - To change the groove direction and valid region, click 

.

6.4.2 For Unclosed Objects

To set the groove direction for unclosed objects, do the following:

1. In **NcEditor**, click  amd select the target unclosed object.
2. Do one of the following in the tool bar:
   - To change the groove direction, click 

/ 

.
   - To change the groove direction and machining direction, click 

/ 

.

6.5 Set Tool Compensation

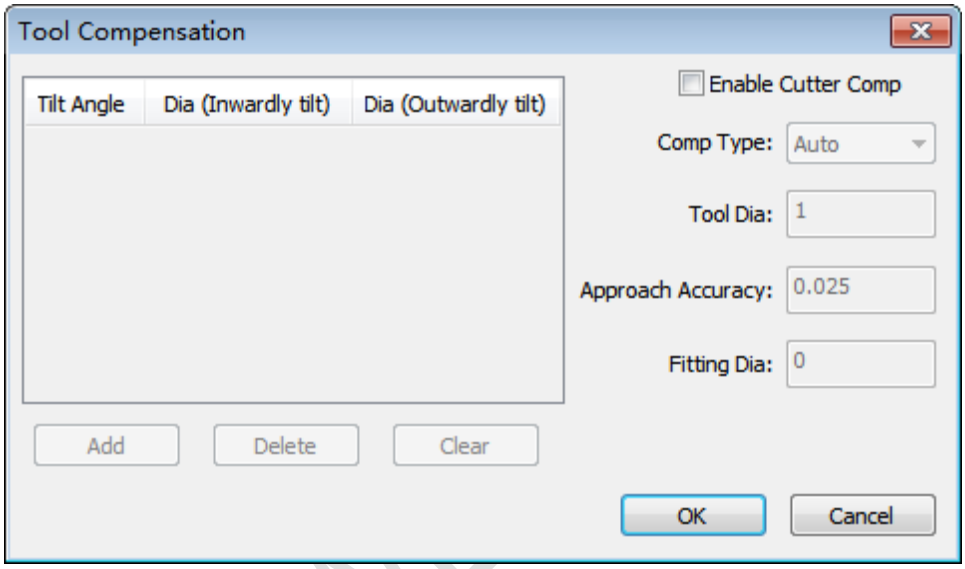
This operation is used to compensate the tool diameter on the designed machining dimension, so as to reduce the offset between the actual machined dimension and designed dimension, which is caused by the tool diameter.

To set tool compensation, do the following:

1. In **NcEditor**, click  and select the target object.

2. Click 

in the tool bar. **Tool Compensation** dialog box pops up:



3. Check **Enable Cutter Comp** and select a type of tool compensation:

   - Auto: The system will automatically set the position of tool compensation according to the groove direction. The default position is at the side of toolpath in the scrap region.
   - Left: Along the machining direction, compensate the tool diameter on the left of workpiece.
   - Right: Along the machining direction, compensate the tool diameter on the right of workpiece.
   - Tilt angle: The kerf width differs in the tilt angle. According to different tilt angles, compensate the corresponding tool diameter.

4. According to the actual tool diameter, set parameter **Tool Dia**.

5. Click **OK**.

To delete added tool compensation, select the target object and do one of the following:

- In the tool bar, click 

.
- In **Tool Compensation** dialog box, uncheck **Enable Cutter Comp** and click **OK**.

6.6 Set a Lead Line

This operation is used to set a lead line in scrap region (invalid region) for closed objects, so as to avoid possible cutting errors or damage to the workpiece caused by the tool dwelling above the start point of machining for a long time.

To set a lead line, do the following:

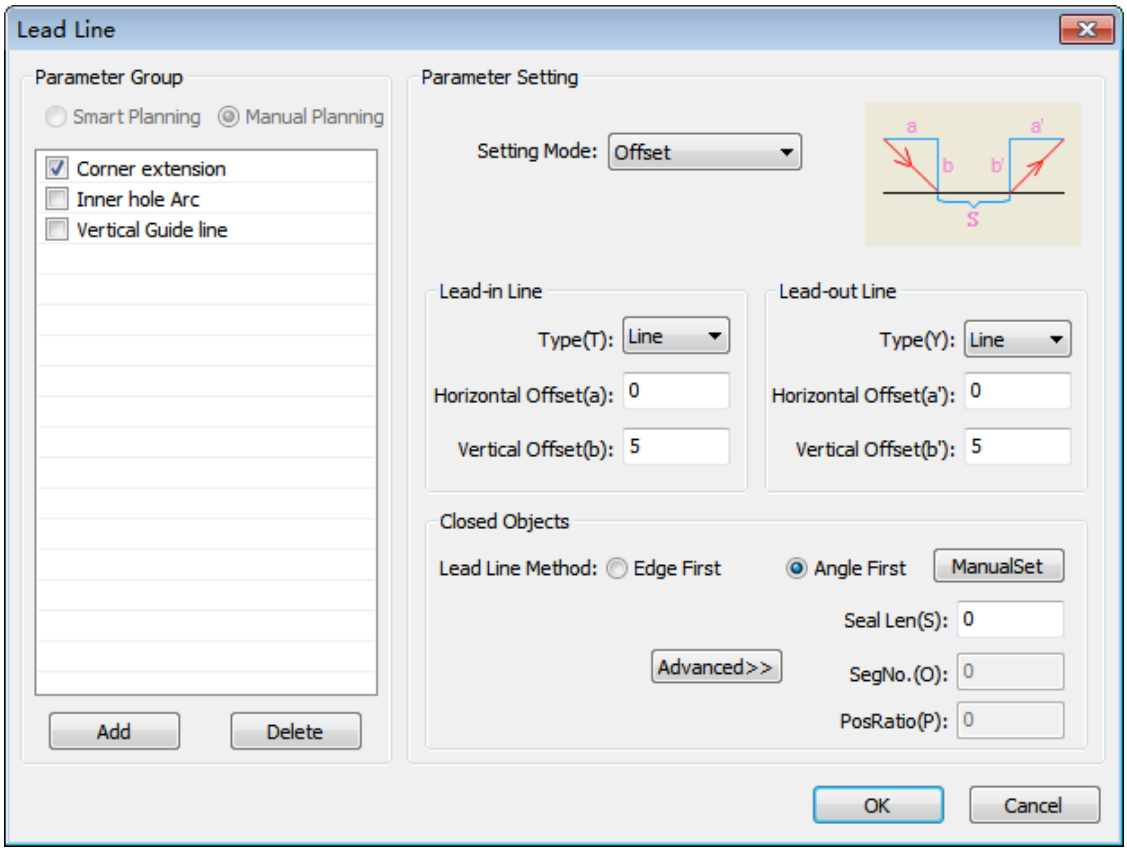
1. In **NcEditor**, click  and select the target object.

   Note: You can check the groove direction by clicking  .

2. To open the **Lead Line** dialog box, do one of the following:

   - In the tool bar, click 

.
   - In the drawing area, right click and select **Set Lead Line** in the pop-up menu.
   - In the menu bar, click **Object** → **Set Lead Line**.

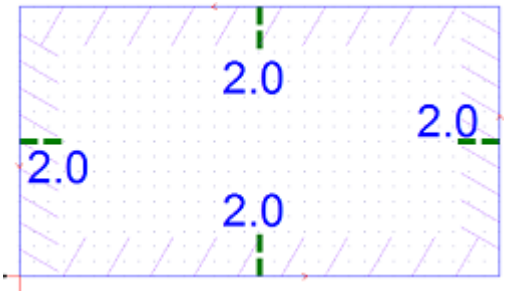


3. Select a lead line group :

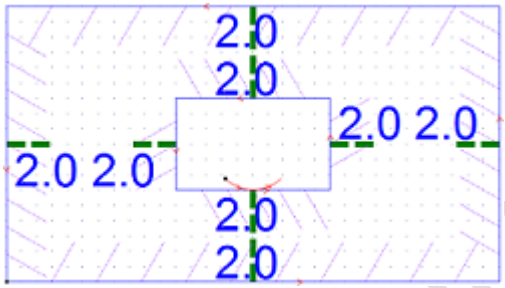
   - Smart Planning: The software automatically selects a group among the preset groups for the target object.
   - Manual Planning: Manually select a group among the preset groups for the target object.

The preset groups include the following:

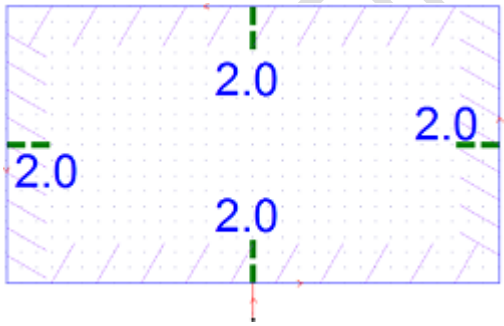
- Corner extension: The extension line of corner.



- Inner hole Arc: With an arc shape, the lead line is at the center position of the longest edge.



- Vertical Guide line



4. Select the following mode of lead line:
   - Angel and length
   - Line and arc
   - Offset
5. Set the type of lead-in line and lead-out line.

6. Select one of the following strategies for the lead line:
   - **Edge first**: Set the lead line at the center position of the longest edge of the closed object.
   - **Corner first**: Set the lead line at the corner with the largest angle in the closed object.
7. Set parameter **Seal Len**.
   - If it is greater than 0, the waterjet tool will over cut at the sealing position.
   - If it is less than 0, the waterjet tool will not cut through at the sealing position.
8. To add a lead line, do one of the following:
   - Click **OK**. Lead lines will be automatically added.
   - Click **ManualSet** and select the position of lead line by your mouse.

To delete the added lead line, select the target object and do one of the following:

- In the tool bar, click 

.
- In **Set Lead Line** dialog box, set the type of lead-in line and lead-out line to **None** and click **OK**.

6.7 Reverse the Lead Line

In general, the groove direction is in the internal region of the object while the lead line is added to the external region of object. This operation is used to reverse the lead line without changing the groove direction, so as to make them both in the internal region of object.

At this time, the scrap region of workpiece (invalid region) is the the internal region of object.

Before reversing the lead line, ensure the lead line has been added.

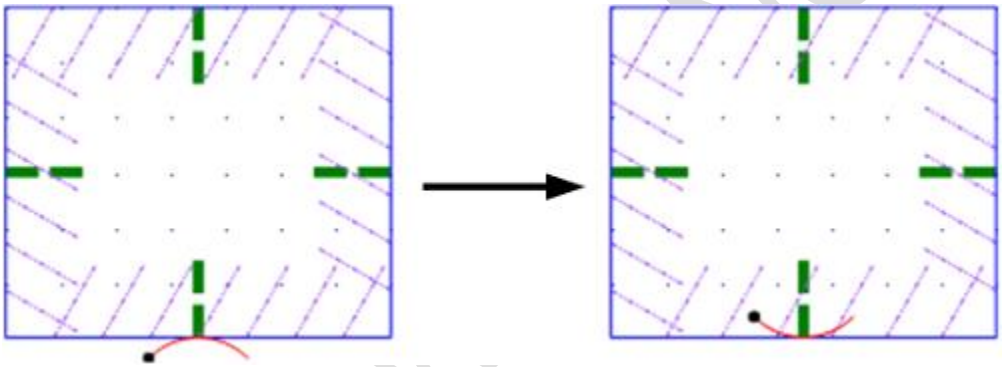
To reverse the lead line, do the following:

1. In **NcEditor**, click  and select the target object.

2. To reverse the lead line, click 

in the tool bar.

The result is as follows:



- Red arc: lead line
- Green dash line: groove direction

6.8 Sort the Machining Order

This operation is used to sort the machining order for several objects in the program file.

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

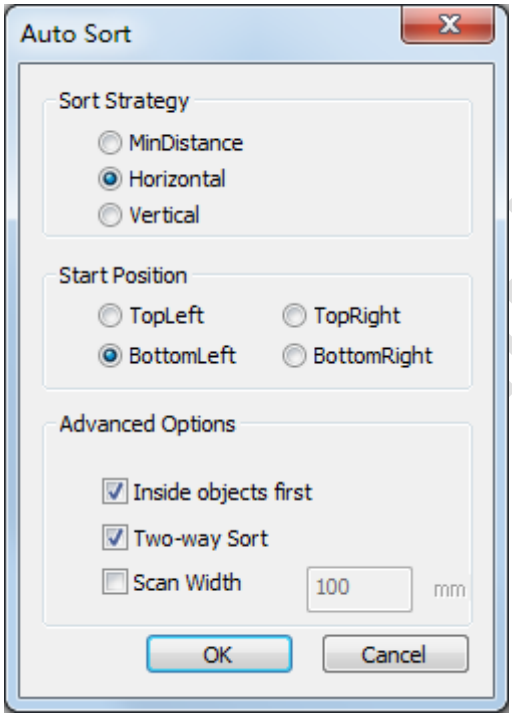
- Automatically sort the machining order.
- Manually sort the machining order.
- Set the machining order in the list.

6.9 Automatically Sort the Machining Order

This operation is used to automatically sort the machining order for several objects.

To automatically sort the machining order, do the following:

1. In **NcEditor**, click  and select the target object.
2. To open **Auto Sort** dialog box, do one of the following:
   - In the tool bar, click  .
   - In the drawing area, right click and select **Auto Set Machining Order**in the pop-up menu.
   - In the menu bar, click **Object** → **Auto Set Machining Order**.



3. Select one of the following strategies in **Sort Strategy** area:
   - Minimum Distance: Sort the machining order by the stipulated minimum distance.
   - Horizontal Direction: Sort the machining order in X-axis direction.
   - Vertical Direction: Sort the machining order in Y-axis direction.

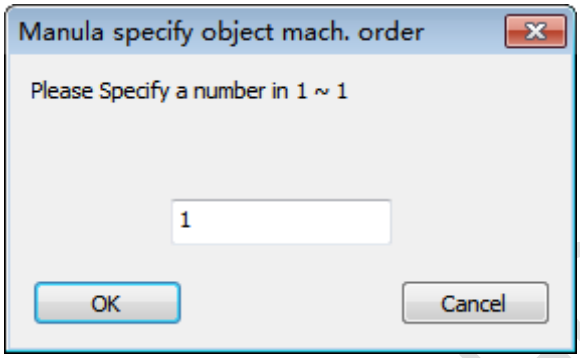
4. Set the start position of sorting in **Start Position** area:
   - Top Left
   - Top Right
   - Bottom Left
   - Bottom Right
5. **Optional:** To set the sorting direction, check one or several option(s) in **Advanced Options** area:
   - Inside object first: The system sorts the machining order from the inside to the outside when several objects are nested.
   - Two-way sort: Based on the selected the sort strategy and start position, the system alternately sorts the machining direction based on one of the following:
     - One row from left to right and another row from right to left.
     - One column from top to bottom and another column from bottom to top.
   - Scan width: Based on the set scanning width, the system sorts the machining direction.

6.9.1 Manually Sort the Machining Order

This operation is used to select one object and manually input its machining order.

To manually sort the machining order, do the following:

1. In **NcEditor**, click  and select one single object.
2. To open **Manual specify object mach. order** dialog box, do one of the following:
   - In the drawing area, right click and select **Manual Set Machining Order**in the pop-up menu.
   - In the menu bar, click **Object** → **Manual Set Machining Order**.



3. Specify a number among 1 ~ n and click **OK**.

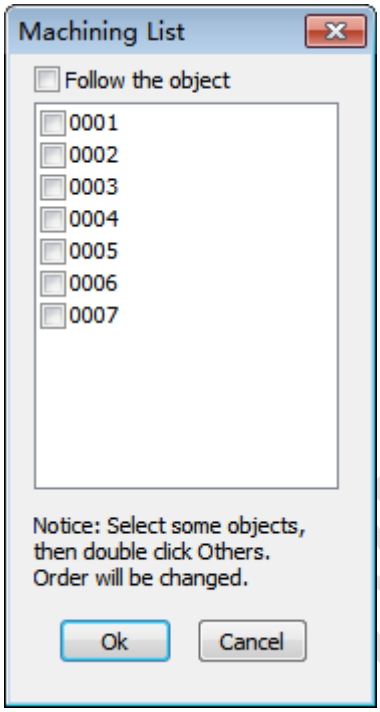
   n is the total of objects and the maximum number.

6.9.2 Set the Machining Order in the List

This operation is used to modify the machining order in the list for at least 2 objects according to the object number specified by the system.

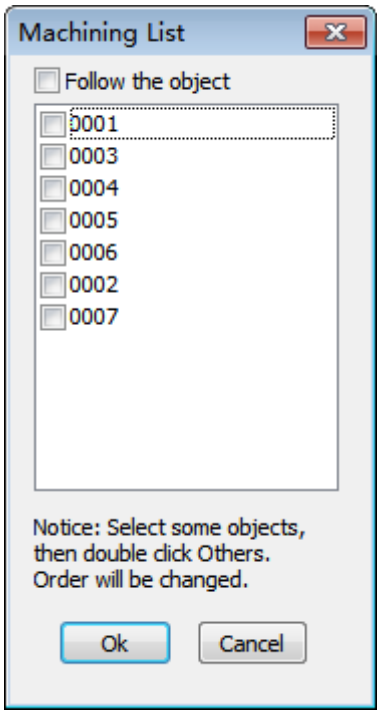
To set the machining order in the list, do the following:

1. In **NcEditor**, to open **Machining List** dialog box, do one of the following:
   - In the menu bar, click **Object** → **Machining Order List**.
   - In the drawing area, right click and select **Machining Order List**in the pop-up menu.



2. To adjust the machining order, check the target object number and double click the other numbers. The former is listed in front of the latter.

   E.g. Check 0002 and double click 0007. Then 0002 is listed in front of 0007.



   With **Follow the object** checked, the former is listed behind the latter. E.g. 0002 is listed behind 0007.

3. Click **OK**. The system cuts objects from top to bottom in the list.

6.10 Use Vertical Plunge

The abrasive tube keeps vertically downward to penetrate workpieces at the start point of object, and then the rotary axis swings to the correct cutting position. As a result, damage to the vulnerable and layered workpieces caused by punching can be reduced.

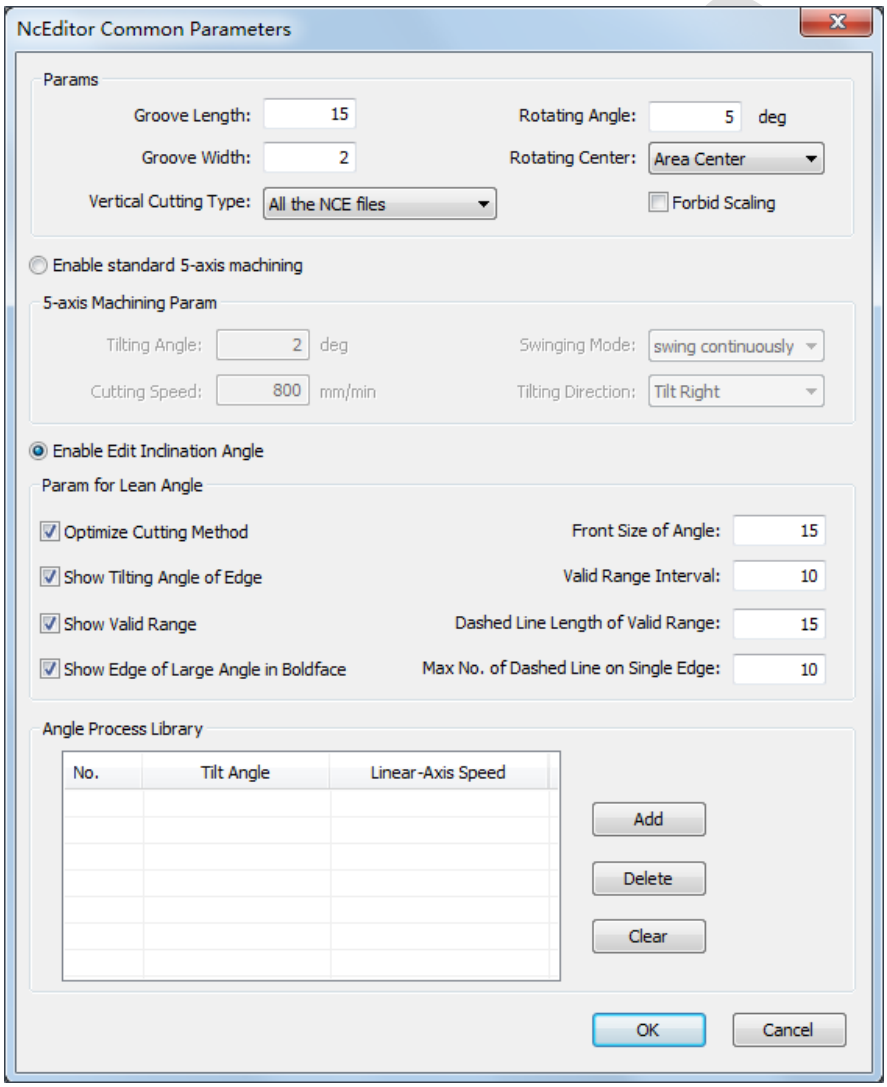
It is only applicable to NCE files.

To use vertical plunge, do the following:

1. In **NcEditor**, to open **NcEditor Common Parameters** dialog box, do one of the following:

   - In the tool bar, click 

.
   - In the menu bar, click **View** → **NcEditor Common Parameters**.



2. To select the position for vertical plunge, select one of the following in the drop-down box of **Vertical Cutting Type**:
   - NCE files with lead lines: Use vertical plunge at the start point of lead line.
   - All the NCE file: Use vertical plunge at the start point of cutting.
3. Click **OK**.

6.11 Set the Tilt Angle and Cutting Speed

This operation is used to set the tilt angle and cutting speed according to the actual situation.

To set the tilt angle and cutting speed, do one of the following:

- Use standard five-axis machining.
- Edit tilt angle.

6.11.1 Use Standard Five-axis Machining

This operation is used to machining all objects with standard five-axis machining parameters.

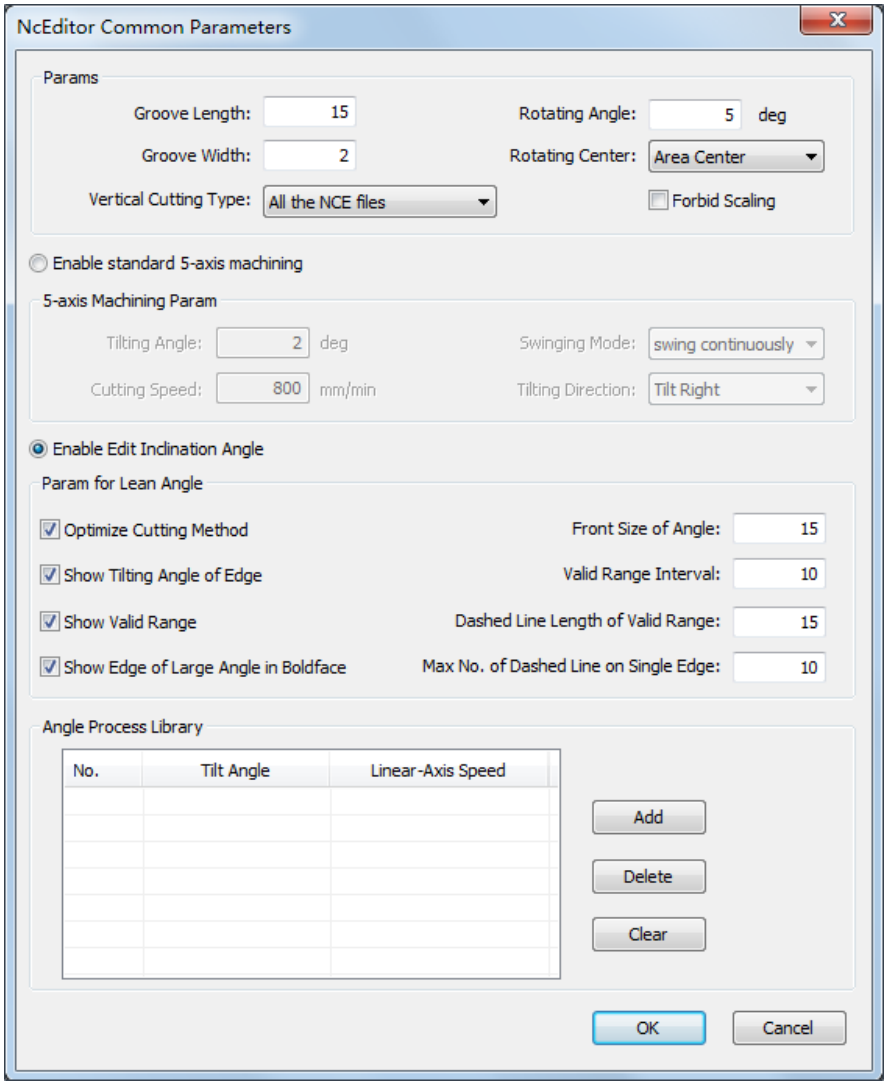
It is applicable to the object whose tilt angle of each edge is the same.

To use standard five-axis machining, do the following:

1. In **NcEditor**, to open **NcEditor Common Parameters** dialog box, do one of the following:

   - In the tool bar, click 

.
   - In the menu bar, click **View** → **NcEditor Common Parameters**.



2. Select **Enable standard 5-axis machining** and click **OK**.

   It is recommended to use default values of parameters **Tilting Angle** and **Cutting Speed**.

6.11.2 Edit Tilt Angle

This operation is used to select different tilt angles and corresponding speed, so as to ensure cutting effect of workpiece surface.

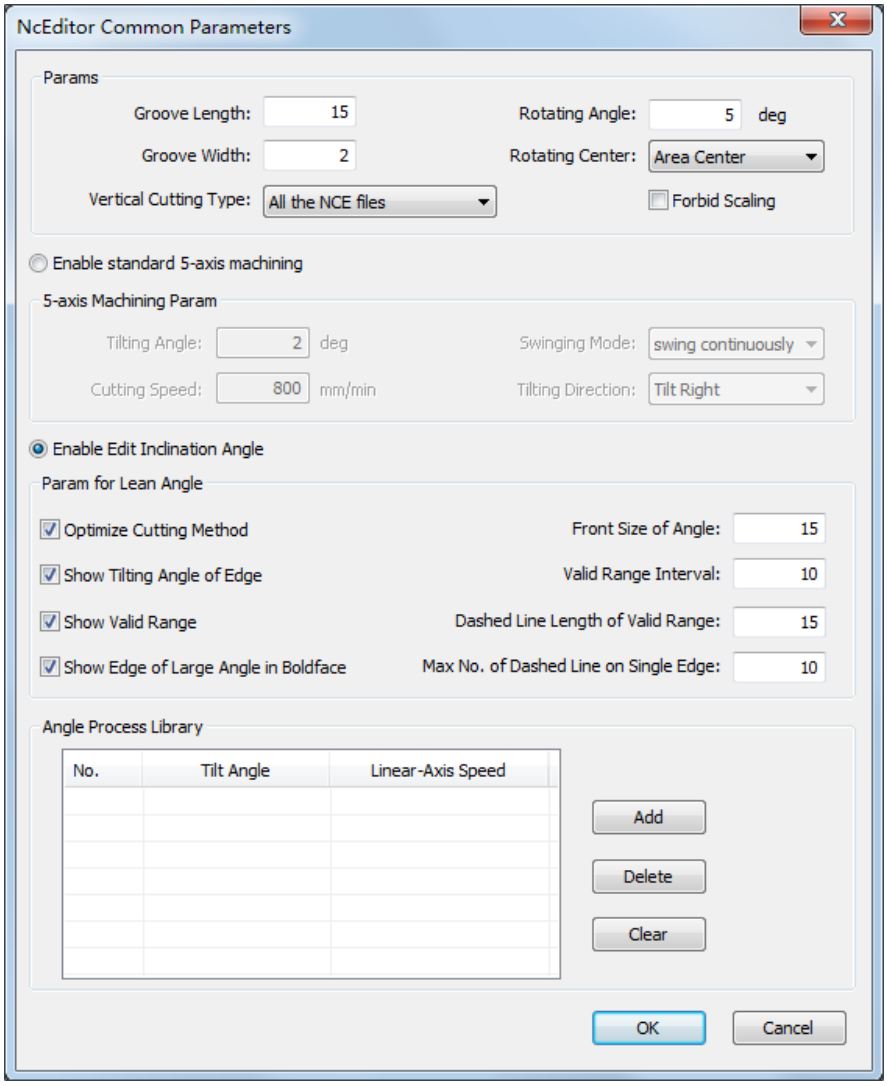
It is applicable to the object whose tilt angle of each edge is different.

To edit tilt angle, do the following:

1. In **NcEditor**, to open **NcEditor Common Parameters** dialog box, do one of the following:

   - In the tool bar, click 

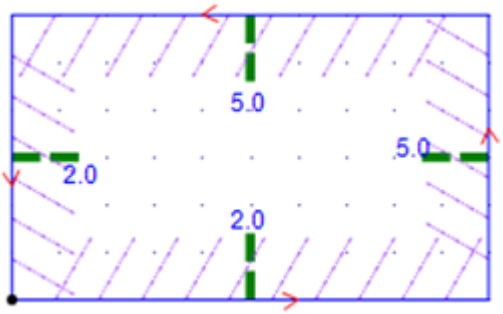
.
   - In the menu bar, click **View** → **NcEditor Common Parameters**.



2. Select **Enable Edit Inclination Angle**, select the following parameters in **Param for Lean Angle** area:

   - Optimize Cutting Method: Optimize cutting by controlling the uncertain factors caused by tool error in scrap region (invalid region).
   - **Note:** Only when the groove direction and valid region are within the internal region of object, this parameter takes effect.

- Show Tilting Angle of Edge: show in figures as follows:



- Show Valid Range: Shows in purple lines.
- Show Edge of Large Angle in Boldface: The larger the tilt angle of edge, the wider the line weight of edge.

3. To add / delete technics for tilt angles, including tilt angle and speed of the linear axis, in **Angle Process Library** area, click **Add/ Delete**.

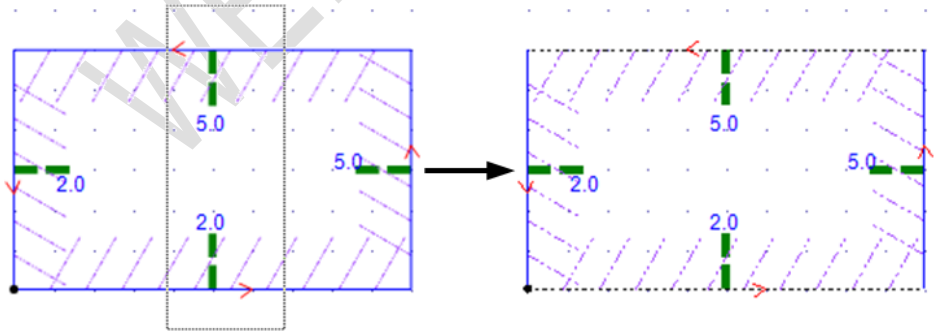
   Note: The added tilt angle cannot be the same with the existing one.

4. To set tilt angle and speed for each edge of object, do one of the following:

   - By selecting the target edge:

     1. Click 

in the tool bar and select the target edge. **Set Tilt Angle** dialog box pops up and the selected edge shows as a dash line:



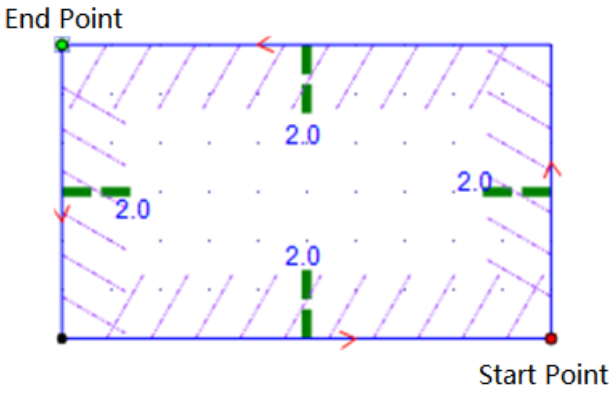
     2. Set the tilt angle and tilt direction of the tool head and click **OK**.

        The speed for the tilt angle corresponds to that in **Angle Process Library** area of **Editor Common Parameters**dialog box.

– By selecting the start point and end point:

1. To open **Set Tilt Angle** dialog box, click 

in the tool bar, select the start point on the object, move along the machining direction and select the end point.



The red arrow represents the machining direction.

2. Set the tilt angle and tilt direction of the tool head and click **OK**.

After selecting **Enable Edit Inclination Angle**, according to your need, do one of the following in the tool bar:

- To change the groove direction, click 

/ 

.

  Alternatively, do related settings in **Set Tilt Angle** dialog box.

- To change the valid region of object, click 

.

7 System Management

Through this section, you can quick know related information about system management of **NcStudio V10 Waterjet Cutting Control System**.

System management includes the following:

7.1 Generate an Installation Package

This operation 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.

According to different purposes, installation packages include the following types:

- Formal installation package: for actual use and simulation.

  It is mainly used for debugging and adjustment.

- Simulative installation package: for simulation.

  It is mainly used for demonstrations to customers.

To generate an installation package, do one of the following:

- To generate an formal installation package, click **File** → **Generate Installation** and select the installation path.
- To generate a simulative installation package, click **File** → **Generate Emute Installation** and select the installation path.

The system begins to generate an installation package.

After finishing, you can find the generated installation package under the set path.

7.2 Select a System Language

At present, **NcStudio** supports Chinese and English.

To select a system language, do one of the following:

- During installing the software, select a system language.
- After installing the software, in the menu bar, click **Language** → **Chinese / English**.

7.3 Register the Software

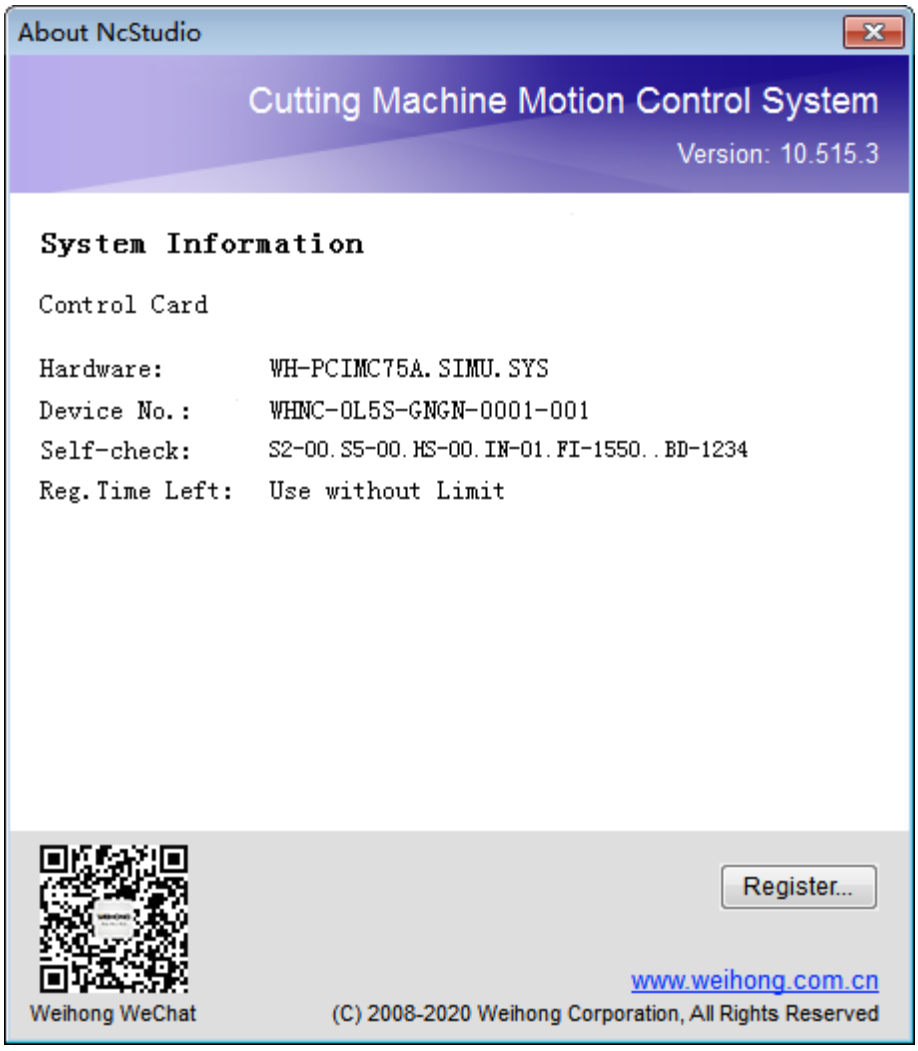
This operation is used to do registration through the board card and restrict the system usage time.

Before registering the software, do the following:

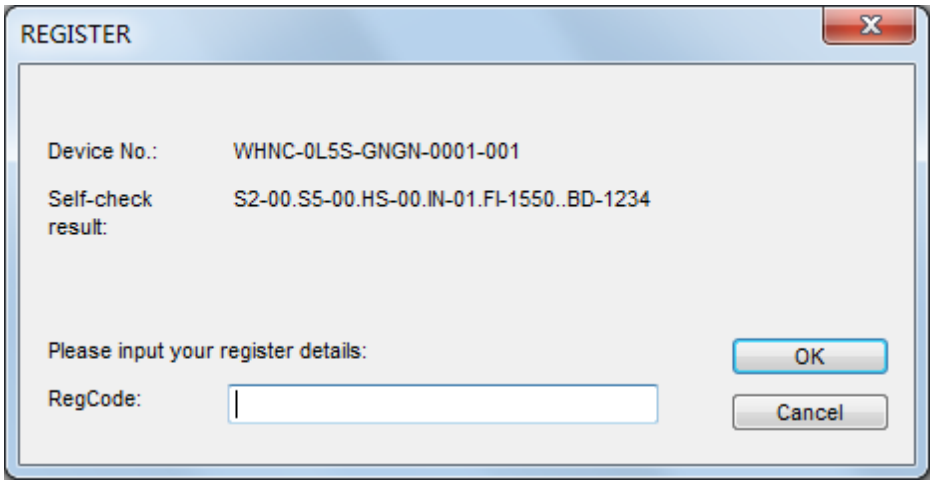
1. The machine tool is in idle or E-stop status.
2. To get a registration code, send the device number to your supplier.

To register the software, do the following:

1. Click **Help** → **About NcStudio**. **About NcStudio** dialog box pops up:



2. Click **Register**. **REGISTER** dialog box pops up:



3. Input the registration code and click **OK**.

   The prompt *Register succeeded. Please restart the software.* appears.

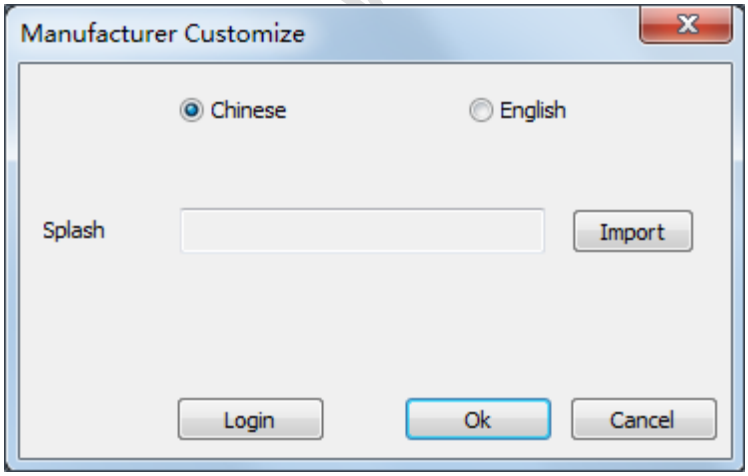
Restart the software to make registering effective.

7.4 Customize the Startup Interface

This operation is used to customize the startup interface for both English and Chinese interfaces.

To customize the startup interface, do the following:

1. Click **Help** → **Manufacturer Customize**. **Manufacturer Customize** dialog box pops up:



2. Select the target language.

3. To activate **Import** button, click **Login**, and input the manufacturer password.

4. Click **Import** and select an image with BMP format.

If the software normally shuts down, the customized startup interface shows after restarting the software.

8 Parameters

Through this section, you can quick know the related parameters of **NcStudio V10 Waterjet Cutting CNC System**.

Including the following:

- Cutting parameters
- Operation parameters
- Axis parameters
- Controller parameters
- Program parameters
- Interface parameters
- Other parameters

8.1 Cutting Parameters

Including the following:

- Delay
- Output Mode of Low Pressure
- Pressure Way
- Cutting Type
- Tilt Angle of Cut Plane
- Tilt Direction of Cut Plane
- Arc Fitting Accuracy
- Max Rotating Angle of C-axis
- Punching
- Velocity
- Conner Controlling
- Drilling

8.1.1 Delay

Open HP Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning on the high-pressure valve.

Close HP Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning off the high-pressure valve.

Open Cutting Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning on the water valve.

Close Cutting Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning off the water valve.

Open Abrasive Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning on the abrasive valve.

Close Abrasive Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning off the abrasive valve.

Pause before Cut

- Unit: ms
- Range: 0~600000
- Default value: 0
- Effective: Immediately
- Description: The delay time before cutting.

Pressure Discharge Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 0
- Effective: Immediately
- Description: The delay time to turn off the water valve after machining.

Pause after Cut Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 0
- Effective: Immediately
- Description: The delay time after cutting.

Close LP Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning off low-pressure valve.

Open LP Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 1000
- Effective: Immediately
- Description: The delay time after turning on low-pressure valve.

After Stop Close Abrasive And Hv Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 0
- Effective: Immediately
- Description: The delay time to turn off the water valve after turning off the high-pressure abrasive valve when “Stop” button is clicked.

Close Booster Delay Time

- Unit: ms
- Range: 0~600000
- Default value: 30000
- Effective: Immediately
- Description: With the high-pressure valve off, the delay time to automatically turn off the water pump.

8.1.2 Output Mode of Low Pressure

LP Output Mode

- Unit: -
- Range: 0;1;2;3
- Default value: 1
- Effective: Immediately
- Description: The output mode of low pressure.
  - 1: turn on the low-pressure valve before turning on the high-pressure valve while turning off the low-pressure valve after turning off the high-pressure valve.
  - 2: turn on the low-pressure valve first. And after delay time, turn off the low-pressure valve and turn on the high-pressure valve.
  - 3: turn on the low-pressure valve before machining while turning off the low-pressure valve after machining.

8.1.3 Pressure Way

Pressure Way

- Unit: -
- Range: 1;2;3
- Default value: 1
- Effective: Immediately
- Description: The way of turning on pressure valves.
  - 1: turn on the high-pressure valve only.
  - 2: turn on the low-pressure valve only.
  - 3: turn on high-pressure valve and low-pressure valve.

8.1.4 Cutting Type

Select Cutting Type

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: Immediately
- Description: It is used to select cutting type.
  - 0: abrasive cutting
  - 1: pure-water cutting

8.1.5 Tilt Angle of Cut Plane

The Lean Angle of Plane Cutted

- Unit: deg
- Range: 0~90
- Default value: 2
- Effective: After restart
- Description: The tilt angle of cut plane.

8.1.6 Tilt Direction of Cut Plane

The Lean Direction of Tool

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: After restart
- Description: The leaning direction of cut plane.
  - 0: seeing from the machining direction, the cut plane leans to the right.
  - 1: seeing from the machining direction, the cut plane leans to the left.

8.1.7 Arc Fitting Accuracy

Arc Fitting Accuracy

- Unit: mm
- Range: 0.0001~1
- Default value: 0.01
- Effective: After restart
- Description: The length of each short line in interpolation.

8.1.8 Max Rotating Angle of C-axis

Max Rotating Angle of C Axis

- Unit: deg
- Range: 360~108000
- Default value: 0
- Effective: After restart
- Description: The maximum rotating angle of C-axis to prevent enlacing during generating toolpath.

8.1.9 Punching

Punch Type

- Unit: -
- Range: 0;1;2
- Default value: 0
- Effective: After restart
- Description: The punching type.
  - 0: none
  - 1: arc punching. That is, circle is punched at the start point of the lead line.
  - 2: swing punching. That is, straight-line is punched at the start point of the lead line.

Punching Strategy

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: After restart
- Description: The punching strategy.
  - 0: single punching. That is, during machining multiple objects, each object is punched and cut before machining the next one.
  - 1: batch punching (only support NCE file). That is, during machining multiple objects, all objects are punched first before cutting them.

Punch Distance

- Unit: mm
- Range: 0.01~10000
- Default value: 2
- Effective: After restart
- Description: The punching diameter for arc punching, and punching interval for swing punching.

Punch Counts

- Unit: -
- Range: 1~1000
- Default value: 3
- Effective: After restart
- Description: The machining times of circle for arc punching, and the swinging times of straight-line for swing punching.

Punch Speed

- Unit: mm/min
- Range: 0.001~100000
- Default value: 1000
- Effective: After restart
- Description: The speed to punch.

Arc Punching Position

- Unit: -
- Range: 0;1;2
- Default value: 0
- Effective: After restart
- Description: The circle position relative to the start point of the lead line for arc punching.
  - 0: left. That is, the circle is on the left of the start point of the lead line.
  - 1: middle. That is, the circle is at the middle of the start point of the lead line.
  - 2: right. That is, the circle is on the right of the start point of the lead line.

Punch Position

- Unit: -
- Range: 1;2
- Default value: 1
- Effective: After restart
- Description: The punching position of waterjet cutter.
  - 1: at the start point of lead line.
  - 2: at start points of all objects.

8.1.10 Velocity

AX Startup Vol

- Unit: mm/min
- Range: 0~100000
- Default value: 0
- Effective: Immediately
- Description: The minimum feedrate during machining.

Default Feedrate

- Unit: mm/min
- Range: 0~100000
- Default value: 800
- Effective: Immediately
- Description: The default speed during machining (It does not refer to the positioning speed).

AX Linear Mach ACC

- Unit: mm/s^2
- Range: 0.001~100000
- Default value: 300
- Effective: Immediately
- Description: The maximum acceleration of each linear axis during machining.

REV Default Feedrate

- Unit: rpm
- Range: 0~100000
- Default value: 15
- Effective: Immediately
- Description: The default speed of each revolving axis during machining (It does not refer to the positioning speed).

8.1.11 Conner Controlling

Whether Use ACC

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: Immediately
- Description: Whether to use corner deceleration.

Conner Speed Percentage

- Unit: -
- Range: 0~100
- Default value: 100
- Effective: Immediately
- Description: The percentage of the corner speed and normal machining speed at the other places.

Corner ACC Distance

- Unit: mm
- Range: 0~100
- Default value: 2
- Effective: Immediately
- Description: The accelerating distance at the corner.

Corner DECEL Distance

- Unit: mm
- Range: 0~100
- Default value: 2
- Effective: Immediately
- Description: The decelerating distance at the corner.

End Section DECEL Distance

- Unit: mm
- Range: 0~100
- Default value: 2
- Effective: Immediately
- Description: The decelerating distance at the last segment of object.

Conner Slow Angle

- Unit: deg
- Range: 0~180
- Default value: 60
- Effective: Immediately
- Description: The corner deceleration is enabled when the corner angle is larger than this parameter.

8.1.12 Drilling

Start Height

- Unit: mm
- Range: 0,9999
- Default value: 5
- Effective: Immediately
- Description: The Z-axis coordinate of the drill at the beginning.

ON Delay

- Unit: ms
- Range: 0,1e+007
- Default value: 3000
- Effective: Immediately
- Description: The delay time to start drilling after the drill moves to the start point.

Max Drilling Time

- Unit: ms
- Range: 0,1e+007
- Default value: 3000
- Effective: Immediately
- Description: The longest time of the cylinder to move down. During drilling, the cylinder pushes out down until trigger the signal of drilling in position, which means drilling ends and the cylinder pulls in.

Pulled in Time

- Unit: ms
- Range: 0,1e+007
- Default value: 3000
- Effective: Immediately
- Description: The duration from detecting the signal of drilling in position to detecting the signal of drill pulling in in position.

Start Machine Coor

- Unit: ms
- Range: -10000,10000
- Default value: 100
- Effective: Immediately
- Description: The machine coordinates of the drill at the start time of drilling.

Falling Distance

- Unit: mm
- Range: 0,10000
- Default value: 100
- Effective: Immediately
- Description: The down depth of the drill before drilling.

Drilling Depth

- Unit: mm
- Range: 0,10000
- Default value: 10
- Effective: Immediately
- Description: The down depth of the drill after drilling.

Drilling Speed

- Unit: mm/min
- Range: 0.01,100000
- Default value: 1000
- Effective: Immediately
- Description: The feedrate during drilling.

8.2 Operation Parameters

Including the following:

- Wait time on E-stop
- Brake control
- Abrasive valve
- Controlling pressure
- Controlling circle velocity
- Manual mode
- Automatic machining
- Reference point
- Other operation parameters

8.2.1 Wait Time on E-stop

Wait Time On Estop

- Unit: sec
- Range: 0.001~10
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer
- Description: The wait time for a complete stop after E-stop.

8.2.2 Brake Control

Brake Control

- Unit: -
- Range: 0;1;2
- Default value: 1
- Effective: After restart
- Permission: Manufacturer
- Description: The way of brake control.
  - 0: by communication data.
  - 1: by brake input signal.
  - 2: by the driver.

8.2.3 Abrasive Valve

Abrasive Valve Control Mode

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: After restart
- Permission: Manufacturer
- Description: The way of controlling the abrasive valve.
  - 0: by analog quantity
  - 1: by pulse spindle

Abrasive Valve Output Voltage

- Unit: mv
- Range: 0~10000
- Default value: 5000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The output voltage of the abrasive valve that is controlled by the analog quantity.

Abrasive Valve Output Voltage Increment

- Unit: mv
- Range: 0~100
- Default value: 5
- Effective: Immediately
- Permission: Manufacturer
- Description: The increment of output voltage in each analog control period of the abrasive valve.

Abrasive Valve Output Voltage Mode

- Unit: -
- Range: 0;1
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer
- Description: The mode of output voltage when the abrasive valve is controlled by the analog quantity.
  - 0: gradual change
  - 1: abrupt change

Abrasive Value Stepper Motor Speed

- Unit: rpm
- Range: 0.001~59999.9
- Default value: 200
- Effective: Immediately
- Permission: Manufacturer
- Description: The speed of stepping motor when the abrasive valve is controlled by the pulse spindle.

8.2.4 Controlling Pressure

Press. Serial Port Address

- Unit: -
- Range: 1~9 (Integer)
- Default value: 1
- Effective: After restart
- Permission: Manufacturer
- Description: 1: COM1; 2: COM2; 3: COM3; 4: COM4; 5: COM5; 6: COM6; 7: COM7; 8: COM8; 9: COM9.

Press. Serial Port Node No

- Unit: -
- Range: 0~99
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer
- Description: The node number for communication when the serial port controls pressure.

8.2.5 Controlling Circle Velocity

Ref Circle Max Velo

- Unit: mm/min
- Range: 0.001~100000
- Default value: 800
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The maximum velocity of circle with 10mm diameter.

Circle Min Velo

- Unit: mm/min
- Range: 0.001~100000
- Default value: 100
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The minimum velocity of circle motions.

8.2.6 Manual Mode

Jog Vol

- Unit: mm/min
- Range: 0.001~100000
- Default value: 1800
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default velocity of X-axis or Y-axis in jog mode.

Rapid Jog Vol

- Unit: mm/min
- Range: 0.001~Max Speed of Each Axis
- Default value: 5000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The velocity of X-axis or Y-axis in high-speed jog mode.

Rev Jog Vol

- Unit: rpm
- Range: 0.001~100000
- Default value: 2
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default velocity of rotary axis in jog mode.

Rev Rapid Jog Vol

- Unit: rpm
- Range: 0.001~100000
- Default value: 3
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The velocity of rotary axis in high-speed mode.

Lock Dir After Limited

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: When the soft limit is triggered, the machine tool can only move toward direction not limited, otherwise, it may damage the machine tool.

Jiggle Feedrate

- Unit: mm/min
- Range: 0.001~100000
- Default value: 400
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The speed of fine-tuning.

Jiggle Step Length

- Unit: mm
- Range: 0.001~2
- Default value: 0.5
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The step length of machining corresponding to each fine-tuning.

Jog Vol Z

- Unit: mm/min
- Range: 0.001~100000
- Default value: 800
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default velocity of Z-axis in jog mode.

Rapid Jog Vol Z

- Unit: mm/min
- Range: 0.001~100000
- Default value: 1000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default velocity of Z-axis in high-speed jog mode.

The Step Range of Z axis

- Unit: mm
- Range: 0~10000
- Default value: 10000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The safe range of step size for Z-axis.

8.2.7 Automatic Machining

Speed Assign Type

- Unit: -
- Range: 0;1;2
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The mode to specify feedrate.
  - 0: use the speed specified by the program file.
  - 1: use the default speed.
  - 2: use the speed specified according to the proportion.

Height Z Raise

- Unit: mm
- Range: -10000~10000
- Default value: 5
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The lifting height of Z-axis during dry run.

8.2.8 Reference Point

REFPT Coor

- Unit: -
- Range: -
- Default value: 0
- Effective: After restart
- Permission: Manufacturer
- Description: The machine coordinate of the machine origin. And its default value is 0.

Axis Type Of Bkref

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: Whether to return to the machine origin before machining.

Driving Spindle Back Distance

- Unit: mm
- Range: -1000~1000
- Default value: 100
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The retract distance of driving spindle when double Y-axes return to the machine origin or during checking the machine origin.

The Speed of Going to Machine Origin

- Unit: mm/min
- Range: 0~1000
- Default value: 200
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The speed to return to the machine origin.

Tool Lift Speed

- Unit: mm/min
- Range: 0~1000
- Default value: 50
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The lifting speed during height measurement.

8.2.9 Other Operation Parameters

Safe Height

- Unit: mm
- Range: 0~10000
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The safe height to avoid collisions in the workpiece coordinate, which is used after returning to the machine origin and breakpoint resume.

Near Point Rang

- Unit: -
- Range: 0~500
- Default value: 100
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Adjacent point machining can be used within the machining range of adjacent point.

Use Special Control of Z axis or not

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to adopt optimized toolpath in NCE file during G00 positioning.

Enable Edit Inclination Angle

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: :Whether editing inclination angle of each edge is allowable in NcEditor.

Show Edge Inclination Angle

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to show inclination angle of each edge in NcEditor.

Action After Programming

- Unit: -
- Range: 0;1;2;3
- Default value: 3
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The tool action every time machining ends.
  - 0: do not move.
  - 1: back to the fix point.
  - 2: back to the workpiece origin.
  - 3: back to the set point (workpiece coordinate).

SETPT POS

- Unit: -
- Range: -99999~99999
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The workpiece origin of the set end point.

FIXPT POS

- Unit: -
- Range: -99999~99999
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The machine coordinate of the fixed point.

Display Simu Out Range

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to prompt if simulation is out of machine travel range.

Clear Cycle Times at Cycle Stop

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to clear the cycle times when cycle stops or software is restarted or files are reloaded.

Y-Axis Work Range

- Unit: -
- Range: 1;2
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer
- Description: It is used to specify the Y-axis work range.
  - 1: positive work range
  - 2: negative work range

Axis of Rotation is Valid or not

- Unit: -
- Range: 0;1
- Default value: 1
- Effective: After restart
- Permission: Manufacturer;operator
- Description: Whether the rotary axis is valid.
  - 0: valid
  - 1: invalid

8.3 Axis Parameters

Including the following:

- Axis
- Position feedback
- Compensation
- Velocity limit
- Bus drive
- Setting double Y
- Controlling dry run
- Laser head travel

8.3.1 Axis

Axis Dir

- Unit: -
- Range: 1;-1
- Default value: 1
- Effective: After restart
- Permission: Manufacturer
- Description: The direction of axes.
  - 1: positive
  - The actual direction is the same with the set direction of machine tool.
  - -1: negative
  - The actual direction is opposite to the set direction of machine tool.

Workbench Lower Limit

- Unit: -
- Range: -99999~Machine Origin
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: The machine coordinate of the lower limit of workbench travel range that is checked effective.

Workbench Upper Limit

- Unit: -
- Range: Machine Origin~99999
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: The machine coordinate of the upper limit of workbench travel range that is checked effective.

Check Workbench Range

- Unit: -
- Range: Yes;No
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to check the workbench travel range.

Axis Type

- Unit: -
- Range: 1;2;3
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: The axis type.
  - 1: linear axis
  - 2: rotary axis
  - 3: rotary axis with the shortest path

Enable Driver Registration

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to enable drive registration function.

8.3.2 Position Feedback

Leadscrew Pitch

- Unit: -
- Range: 0.001~99999
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: The actual moving distance or degree of corresponding axis per revolution of motor.

  It is related with the specified machine tool. So it is recommended to set this parameter according the actual measuring result.

Power Err

- Unit: -
- Range: 0~99999
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer
- Description: If the position difference between power and reboot is greater than this value, the alarm occur. And it is only for the absolute encoder.

Mechanical Reduction Ratio (Numerator)

- Unit: -
- Range: 1~99999
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: The numerator of mechanical reduction ratio.

Mechanical Reduction Ratio (Denominator)

- Unit: -
- Range: 1~99999
- Default value: 1
- Effective: After restart
- Permission: Manufacturer
- Description: The denominator of mechanical reduction ratio.

8.3.3 Compensation

Enable Leadscrew CMPN

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to enable compensations for the leadscrew errors, including compensations for backlash errors and pitch errors.

Only Backlash CMPN

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description:
  - Yes: the data is read from the parameter **Backlash** to execute compensation.
  - No: the data of backlash errors and pitch errors is read from error file to execute comprehensive compensation.

Backlash

- Unit: -
- Range: 0~1
- Default value: 0
- Effective: After restart
- Permission: Manufacturer
- Description: The backlash of each axis.

  This parameter takes effect only when the parameter **Only Backlash CMPN** is set to **Yes**.

Maximum Angle Allowed for Correction

- Unit: deg
- Range: 0~1
- Default value: 0.8
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum allowable angle of axis offset in perpendicularity correction.

8.3.4 Velocity Limit

AX MAX VOL

- Unit: mm/min
- Range: 0.001~100000
- Default value: -
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum allowable velocity of each axis.

AX MAX CTRL VOL

- Unit: mm/min
- Range: 0.001~100000
- Default value: -
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum allowable velocity of control point on each axis during machining.

8.3.5 Bus Drive

Driver Station Address

- Unit: -
- Range: 0;1;2;3;4;5
- Default value: -
- Effective: After restart
- Permission: Manufacturer
- Description: It should be the same with rotary switch setting of drive station address. 0 represents the invalid address.

Driver Station Address Setting Switch

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether it should be the same with dial switch setting of drive station address.

Encode Digit

- Unit: -
- Range: 10~30
- Default value: 23
- Effective: After restart
- Permission: Manufacturer
- Description: The digit number of encoder of the servo motor.

Electronic Gear Ratio (Numerator)

- Unit: -
- Range: 1~1.07374e+009
- Default value: 23
- Effective: After restart
- Permission: Manufacturer
- Description: It should be the same with electronic gear ratio (numerator) of the drive.

Electronic Gear Ratio (Denominator)

- Unit: -
- Range: 1~1.07374e+009
- Default value: 23
- Effective: After restart
- Permission: Manufacturer
- Description: It should be the same with electronic gear ratio (denominator) of the drive.

Servo Type

- Unit: -
- Range: 0;1;2
- Default value: 2
- Effective: After restart
- Permission: Manufacturer
- Description: The type of servo drive.
  - 0: Yaskawa Σ5 servo drive
  - 1: Yaskawa Σ7 servo drive
  - 2: WISE servo drive

Brake Output

- Unit: -
- Range:
- Default value: NA
- Effective: After restart
- Permission: Manufacturer
- Description: It specifies the PLC address for output port of each brake.

  This parameter takes effect only when the parameter **Brake Control** is set to **0**.

8.3.6 Setting Double Y

Is Double Y System or not

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to use the double Y-axes configuration.

Check Y1Y2 Encoder Error

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to check the encoder error between feedback value of Y1-axis and Y2-axis.

Double Y Static Tolerance

- Unit: -
- Range: 0.001~999999
- Default value: 5
- Effective: After restart
- Permission: Manufacturer
- Description: When Y1-axis and Y2-axis are static, if the difference between the feedback and output value is bigger than this value, corresponding alarm will occur. This parameter is effective for Double Y configuration.

Double Y Dynamic Tolerance

- Unit: -
- Range: 0.00~999999
- Default value: 5
- Effective: After restart
- Permission: Manufacturer
- Description: When Y1-axis and Y2-axis are dynamic, if the difference between the feedback and output value is bigger than this value, corresponding alarm will occur. This parameter is effective for Double Y configuration.

8.3.7 Controlling Dry Run

Rapid Travel Feedrate

- Unit: mm/min
- Range: 0.001~100000
- Default value: 5000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default speed of linear axis during positioning. It is not the machining speed.

Rev Rapid Travel Feedrate

- Unit: rpm
- Range: 0~Max Speed of Each Axis
- Default value: 26
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default speed of rotary axis during positioning. It is not the machining speed.

Ax Linear Post Acc

- Unit: mm/s^2
- Range: 0.001~100000
- Default value: 400
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum acceleration of each linear axis during positioning.

8.3.8 Laser Head Travel

Laser Upper Limit

- Unit: mm
- Range: -99999~99999
- Default value: 0
- Effective: After restart
- Permission: Manufacturer
- Description: The machine coordinate of the upper limit of laser head travel during height measurement.

Laser Lower Limit

- Unit: mm
- Range: -99999~99999
- Default value: -150
- Effective: After restart
- Permission: Manufacturer
- Description: The machine coordinate of the lower limit of laser head travel during height measurement.

8.4 Controller Parameters

Including the following:

- General controller
- Handwheel and keyboard
- Operation panel
- Valve state at pause

8.4.1 General Controller

Pausing Time

- Unit: -
- Range: 0.01~10
- Default value: 0.5
- Effective: After restart
- Permission: Manufacturer
- Description: The duration time from running to pause or stop state. And if the value is too small, rock to the machine will occur.

Show REF Point Sign If Soft Limit Triggered

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to show the sign of reference point if the soft limit is triggered.
  - Yes: the soft limit takes effect after returning to the machine origin.
  - No: the soft limit immediately takes effect.

8.4.2 Handwheel and Keyboard

HW Count Strictly

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: If count pulses strictly, the machine tool will move the exact pulses generated by handwheel; Otherwise, the machine tool moves only when the handwheel starts turning.

HW Direction

- Unit: -
- Range: 1;-1
- Default value: -1
- Effective: After restart
- Permission: Manufacturer
- Description: The relation between the handwheel direction and the moving direction of axis.

HW Multiple_1

- Unit: -
- Range: 0.0001~10
- Default value: 0.001
- Effective: After restart
- Permission: Manufacturer
- Description: The handwheel multiple*1.

HW Multiple_10

- Unit: -
- Range: 0.0001~10
- Default value: 0.01
- Effective: After restart
- Permission: Manufacturer
- Description: The handwheel multiple*10.

HW Multiple_100

- Unit: -
- Range: 0.0001~10
- Default value: 0.1
- Effective: After restart
- Permission: Manufacturer
- Description: The handwheel multiple*100.

HW Gear Factor

- Unit: -
- Range: 1~1000
- Default value: 1
- Effective: After restart
- Permission: Manufacturer
- Description: The ratio of the rotational speed of handwheel to the feeding speed of axis. The greater the value is, the bigger the feeding speed is.

Forbid Handwheel Gear Track Back Function

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to forbid the track back function during handwheel guide.

Hw Acc

- Unit: mm/s^2
- Range: 1~1000
- Default value: 200
- Effective: After restart
- Permission: Manufacturer
- Description: The smaller the value is, the smoother handwheel movements are.

Support NK Ex Keyset

- Unit: -
- Range: 0;1;2
- Default value: 0
- Effective: After restart
- Permission: Manufacturer
- Description: The type of extended keyword.
  - 0: null
  - 1: Mini keyboard
  - 2: Naiky extended keyboard

Install Extended Keyboard Hook

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to install the extended keyboard hook.

Wireless Handwheel Connection Prompt

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to prompt that the wireless handwheel is successfully connected.

Bar Code Enable

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to support barcode scanning.

Where to Load Toolpath in Scanning

- Unit: -
- Range: 0;1
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: It is used to select where to load toolpath during scanning with a barcode scanner.
  - 0: NcStudio
  - 1: NcEditor

Max Interval for Scanning

- Unit: ms
- Range: 50~5000
- Default value: 150
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The maximum interval to scan barcode of toolpath with a barcode scanner.

Priority Loading File Format

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: During scanning the code, if there is a toolpath file with the same name and different extensions in the folder, the priority file format to load.
  - 0: DXF、NC、G format
  - 1: NCE format

Shortest Response Time of Keyboard

- Unit: ms
- Range: 5~120
- Default value: 30
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The barcode scanner responds within the shortest response time of keyboard. Otherwise, the keyboard responds.

8.4.3 Operation Panel

Min Frp

- Unit: -
- Range: 0~Max Frp
- Default value: 0
- Effective: After restart
- Permission: Manufacturer
- Description: The minimum allowable feedrate.

Max Frp

- Unit: -
- Range: Min Frp~120
- Default value: 120
- Effective: After restart
- Permission: Manufacturer
- Description: The maximum allowable feedrate.

Manual Feedrate is Effective or not

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: When the feedrate in manual mode is not effective, the default feedrate is 100%.

Granularity Feedrate

- Unit: -
- Range: 1~120
- Default value: 1
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The granularity of the feedrate slider bar.

Granularity Abrasive Ctrl

- Unit: -
- Range: 1~100
- Default value: 1
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The granularity of the abrasive control slider bar.

Granularity G00 Feedrate

- Unit: -
- Range: 1~100
- Default value: 25
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The granularity of the G00 feedrate slider bar.

Turn Off Ports When Tracking Back

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to turn off ports during automatically tracking back.
  - Yes: if ports are turned on previously, turn off.
  - No: do not change the previous state.

8.4.4 Valve State at Pause

Pause Port Close Style

- Unit: -
- Range: 1;2;3
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The mode to turn off valves at pause.
  - 1: keep turning on valves
  - 2: immediately turn off valves
  - 3: delay to turn off valves

Start Open Port Style

- Unit: -
- Range: 1;2;3
- Default value: 3
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The mode to turn on valves at pause.
  - 1: keep turning off valves
  - 2: immediately turn on valves
  - 3: delay to turn on valves

8.5 Program Parameters

Including the following:

- G code options
- Controlling track
- Controlling the velocity
- DXF file translation

8.5.1 G Code Options

RT DEC LEN

- Unit: mm
- Range: 1~999
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: During rapid positioning, the distance to decelerate when the spindle gets close to the target position and then moves at the approach speed.

Speed While Approaching to Workpiece

- Unit: mm/min
- Range: 0.001~99999
- Default value: 120
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The feeding speed of the tool when it gets close to the workpiece during positioning.

8.5.2 Controlling Track

Track Corner Tol

- Unit: mm
- Range: 0~0.1
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer
- Description: To improve the finish of machined workpiece, the tool may not move to the exact set position at two-segment connection. When difference of current position and set position reaches value specified by this parameter, the system takes it machining is completed.

Smooth Time

- Unit: s
- Range: 0~1
- Default value: 0.03
- Effective: Immediately
- Permission: Manufacturer
- Description: The longer the time, the smoother the surface of the workpiece. But some of the details may be weakened.

Optimize Cutting Method

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to optimize the cutting method.

Max Inclination Angle Difference

- Unit: -
- Range: 0~360
- Default value: 20
- Effective: After restart
- Permission: Manufacturer
- Description: When the parameter **Optimize Cutting Method** is set to **Yes**, angles exceeding set value will be sectioned.

Vertical Cutting Type

- Unit: -
- Range: 0;1;2
- Default value: 2
- Effective: After restart
- Permission: Manufacturer
- Description: The type of vertical cutting at the start point of object.
  - 0: never
  - 1: only NCE files with lead lines
  - 2: all the NCE files

8.5.3 Controlling the Velocity

Ax Startup Vol

- Unit: mm/min
- Range: 0~100000
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The minimum velocity during machining.

Min Smooth Angle during Lines Connect

- Unit: deg
- Range: 0~180
- Default value: 5000
- Effective: After restart
- Permission: Manufacturer;operator
- Description: When the angle between two lines is smaller than the value of this parameter, the angle is smoothed.

Rev Default Feedrate

- Unit: rpm
- Range: 0.001~100000
- Default value: 15
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The default feeding speed of rotary axis during machining. It is not the positioning speed.

Angle Acceleration

- Unit: deg/s^2
- Range: 0.001~100000
- Default value: 300
- Effective: Immediately
- Permission: Manufacturer
- Description: The angular acceleration of the rotary axis.

Z Linear Acc

- Unit: mm/s^2
- Range: 0.001~100000
- Default value: 400
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum linear acceleration of Z axis.

Axis Max Acc

- Unit: mm/s^2
- Range: 0.001~100000
- Default value: -
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum acceleration of each axis.

Ax Con Acc

- Unit: mm/s^2
- Range: 0.001~100000
- Default value: 1200
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum resultant feed acceleration of adjacent two axes. The recommended value is 1~2 times of the single axis acceleration.

Ax Acc Acc

- Unit: mm/s^3
- Range: 0.001~100000
- Default value: 5000
- Effective: Immediately
- Permission: Manufacturer
- Description: The change rate of acceleration for each axis.

Velo Slide Time

- Unit: s
- Range: 0~99999
- Default value: 0.05
- Effective: Immediately
- Permission: Manufacturer
- Description: The longer the time, the smoother the speed. This parameter has no effect on track precision.

Limit Arc Velo

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer
- Description: Whether to limit the arc-motion velocity.

Arc Forward Length

- Unit: mm
- Range: 0~100
- Default value: 2
- Effective: Immediately
- Permission: Manufacturer
- Description: The forward length of limiting the arc-motion velocity.

Slide Speed for Small Lines

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: Immediately
- Permission: Manufacturer
- Description: Whether to eliminate speed wave during machining small lines.

Reference Length of Slide Speed for Small Lines

- Unit: mm
- Range: 0.001~200
- Default value: 5
- Effective: Immediately
- Permission: Manufacturer
- Description: It is used to adjust the speed wave for the line whose length is shorter than reference length.

Speed Slide Length

- Unit: mm
- Range: 0~300
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer
- Description: The forward length of smoothing speed.

Arc Speed Limit Type

- Unit: -
- Range: 1;2
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer
- Description: The type of arc speed limit.
  - 1: reference circle speed limit
  - 2: linear arc speed limit

Maximum Arc Radius

- Unit: mm
- Range: 0~5000
- Default value: 100
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum radius of linear arc speed limit.

Maximum Arc Speed

- Unit: mm/min
- Range: 1.66667~100000
- Default value: 1800
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum speed of linear arc speed limit.

Minimum Arc Radius

- Unit: mm
- Range: 0~Maximum Arc Radius
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer
- Description: The minimum radius of linear arc speed limit.

Minimum Arc Speed

- Unit: mm/min
- Range: 0~Maximum Arc Speed
- Default value: 100
- Effective: Immediately
- Permission: Manufacturer
- Description: The minimum speed of linear arc speed limit.

8.5.4 DXF File Translation

Tool Up Height

- Unit: mm
- Range: 0~99999
- Default value: 10
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The lifting height of the tool during dry run.

Depth Layer

- Unit: mm
- Range: -99999~0
- Default value: 0
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The machining depth of each tool layer in 2D machining.

First Point as Origin

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: Whether to set the first point of DXF file as zero point.

Enable Machine Bottom

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The valve operation is performed only when cutter arrives at workpiece surface in 3D machining.

Force to Recognize Dxf File As Metric Sys

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: Whether to ignore programming Unit in DXF file.

Ignore Height in DXF

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer;operator
- Description: Whether to ignore height in DXF file.

8.6 Interface Parameters

Including the following:

- Object operation
- Position interface
- Setting interface showing

8.6.1 Object Operation

Interval Distance

- Unit: -
- Range: 0~100000
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The interval distance between two dashes during drawing the groove direction.

Dash Length (Groove Direction)

- Unit: -
- Range: 0~100
- Default value: 15
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The length of the dash that represents groove direction.

Front Size of Angle

- Unit: -
- Range: 1~1000
- Default value: 15
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The pixel size of angle during drawing the leaning angle.

Show Valid Material Range

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to show valid material range when the parameter **Enable Edit Inclination Angle** is set to **Yes**.

Space for Valid Material Range

- Unit: -
- Range: 1~1000
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The space for drawing valid material range with equal distance.

Valid Material Range Dash Length

- Unit: -
- Range: 1~1000
- Default value: 15
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The length of the dash that represents valid material range.

Max No. of Dashed Line on Single Edge

- Unit: -
- Range: 1~100
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The maximum number of dashed lines on a single edge in valid range.

Show Edge of Large Angle in Boldface

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: When **Enable Edit Inclination Angle** is set to **Yes**, the edges will become bold if the angle becomes larger.

Dash Width (Groove Direction)

- Unit: -
- Range: 0.1~100
- Default value: 2
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The width of the dash that represents groove direction.

8.6.2 Position Interface

Auto Load Track

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to automatically parse the machining track after loading the program file.

Auto Load Track Limit

- Unit: KB
- Range: 0~100000
- Default value: 1000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: A machining file can be automatically loaded only when the file size is less than this value.

Whether Auto Load Files

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to automatically load the previous program file when the software starts.

Whether Lock Edit After Machining

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: Immediately
- Permission: Manufacturer
- Description: Whether to lock editing the position and size of objects after machining.

Whether Lock Edit After Import

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to lock editing the position and size of objects after importing the file.

NcEditor Forbid Scaling

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to forbid zooming objects in NcEditor.

8.6.3 Setting Interface Showing

Font Show Adjust Factor

- Unit: -
- Range: 1~100
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The factor to adjust the front size of paragraph number.

8.7 Other Parameters

Including the following:

- Cylinder
- Software configuration type
- Application
- Laser height measurement
- Scanning track
- Shooting positioning

8.7.1 Cylinder

Red Light When Finish

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer;operator
- Description: Whether to prompt by the red light before input with a mouse or keyboard after machining.

8.7.2 Software Configuration Type

Software Configuration

- Unit: -
- Range: 0;1
- Default value: 1
- Effective: After restart
- Permission: Manufacturer;operator
- Description: The type of software configuration.
  - 0: normal
  - 1: with height measurement

8.7.3 Application

Use Metric

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to use metric.

Change Value When the Conversion of the Metric System and British Standard

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to automatically modify parameter values during the conversion of the metric system and british standard.

Enable Rotary Workbench Mode

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to enable the mode of rotary workbench.

8.7.4 Laser Height Measurement

Laser Sensor Station Address

- Unit: -
- Range: 1~16
- Default value: 1
- Effective: After restart
- Permission: Manufacturer
- Description: The station address of laser sensor.

Coarse Position Speed

- Unit: mm/min
- Range: 0.001~10000
- Default value: 800
- Effective: Immediately
- Permission: Manufacturer
- Description: The speed of Z-axis downwards the upper limit of laser sensor during getting the datum.

Sensor Clearance Height

- Unit: mm
- Range: 65~105
- Default value: 85
- Effective: Immediately
- Permission: Manufacturer
- Description: The clearing height of laser sensor.

Altimetry Result Processing Method

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer
- Description: The method to process the result of height measurement.
  - 0: linear area method
  - 1: linear distance method

Sample Period

- Unit: us
- Range: 200;500;1000;2000
- Default value: 2000
- Effective: After restart
- Permission: Manufacturer
- Description: After changing the sampling period, the sensor needs to be initialized.

Average Times

- Unit: us
- Range: 1;4;16;64;256;1024
- Default value: 64
- Effective: After restart
- Permission: Manufacturer
- Description: After changing the average times, the sensor needs to be initialized.

Max Ignorable Distance

- Unit: mm
- Range: 0~200
- Default value: 50
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum continuous distance of getting height errors during laser height measurement.

Tolerance of Height Measurement

- Unit: mm
- Range: 0.01~10
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer
- Description: The maximum deviation of height when XY-plane moves 10mm.

Delay to Push out Dustcover

- Unit: ms
- Range: 0~100000
- Default value: 1000
- Effective: Immediately
- Permission: Manufacturer
- Description: The delay time to push out dust cover during laser height measurement.

Buffer Length during Resume

- Unit: mm
- Range: 0~200
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer
- Description: The extra scanning length to eliminate measurement errors during breakpoint resume.

Ignore Large Segment Abnormal

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: Immediately
- Permission: Manufacturer
- Description: Whether to ignore the parameter **Max Ignorable Distance** during laser height measurement.

8.7.5 Scanning Track

Take Direction

- Unit: -
- Range: 0;1
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer
- Description: The start direction for automatically picking points.
  - 0: the positive direction of X-axis
  - 1: the positive direction of Y-axis

Take the Horizontal Spacing

- Unit: mm
- Range: 1~10000
- Default value: 50
- Effective: Immediately
- Permission: Manufacturer
- Description: Take the preset value of the point Horizontal spacing.

The Buffer Length

- Unit: mm
- Range: 1~100
- Default value: 10
- Effective: Immediately
- Permission: Manufacturer
- Description: The buffer length of inflection point.

Scan Speed

- Unit: mm/min
- Range: 0.001~100000
- Default value: 6000
- Effective: Immediately
- Permission: Manufacturer
- Description: The scanning speed during automatic measurement.

Take the Longitudinal Spacing

- Unit: mm
- Range: 1~10000
- Default value: 200
- Effective: Immediately
- Permission: Manufacturer
- Description: Take the preset value of the point longitudinal spacing.

Cutting Edge Length

- Unit: mm
- Range: 1~1000
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer
- Description: During picking points, precut the edge length.

8.7.6 Shooting Positioning

Enable Positioning by Shooting

- Unit: -
- Range: Yes;No
- Default value: No
- Effective: After restart
- Permission: Manufacturer
- Description: Whether to position by shooting.

Show Shooting Result in NcEditor

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to show shooting result in NcEditor.

Picture Display Mode

- Unit: -
- Range: 1;2
- Default value: 1
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The mode of picture display.
  - 1: by effect
  - 2: by performance

Custom Tool Path Color

- Unit: -
- Range: Yes;No
- Default value: Yes
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: Whether to customize the color of toolpath.
  - Yes: use customized tool path color.
  - No: use the color set in the system.

Red in RGB

- Unit: -
- Range: 0~255
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The customized value of red in RGB.

Green in RGB

- Unit: -
- Range: 0~255
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The customized value of green in RGB.

Blue in RGB

- Unit: -
- Range: 0~255
- Default value: 255
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The customized value of blue in RGB.

Dustcover ON Delay

- Unit: ms
- Range: 0~100000
- Default value: 3000
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The delay time to turn on dust cover.

Deviation X between Cutter Head and Laser Sensor

- Unit: mm
- Range: -99999~99999
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The offset X between the center position of the water cutter head and the laser projection point in the vertical state.

Deviation Y between Cutter Head and Laser Sensor

- Unit: mm
- Range: -99999~99999
- Default value: 0
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The offset Y between the center position of the water cutter head and the laser projection point in the vertical state.

Machine Position when Shooting

- Unit: mm
- Range: -99999~99999
- Default value: -
- Effective: Immediately
- Permission: Manufacturer;operator
- Description: The machine coordinate of cutter head that does not affect the shooting result during shooting.

9 FAQs

Through this section, you can quickly find solutions to common questions that you may encounter during using **NcStudio V10 Waterjet Cutting Control System**.

Common questions include the following:

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

9.1 Questions during Returning to the Machine Origin

It includes the following:

- Cannot detect origin signal
- Axis direction is incorrect
- Speed of coarse positioning is too slow

9.1.1 Cannot Detect Origin Signal

Cause

Error in the origin switch.

Solution

1. Touch the origin switch, check and observe if related ports haves signals in **IO Port** window:
   - Yes: Contact us.
   - No: Proceed to the next step.

   See Check Port Information for details.
2. Check if indicator lights of **X00**, **X03**, **X06**, **X09** and **X12** on the terminal board are on.
   - Yes: Check if the electric circuit between the board card and the terminal board loosens:
     - Yes: Tighten it.
     - No: Contact us.
   - No: Check if the electric circuit between the origin switch and the terminal board loosens:
     - Yes: Tighten it.
     - No: Replace the origin switch.

9.1.2 Axis Direction is Incorrect

Cause

1. Incorrect polarity of port **Reference Point**.
2. Incorrect value of parameter **Axis Direction**.

Solution

1. Modify its polarity.

   See Check Port Information for details.

2. Modify its value to the opposite value.

   See Adjust Axis Direction for details.

9.1.3 Speed of Coarse Positioning is Too Slow

Cause

1. Too small value of the parameter **Coarse Locating Velo**.
2. Polarity of the port **Reference Point** in **NcStudio** does not match with type of the origin switch.

Solution

1. Modify the parameter **Coarse Locating Velo** to a larger value.
2. Modify its polarity.

   See Check Port Information for details.

9.2 Questions about Warnings

It includes the following:

- Simulation result shows running range exceeded machine travel. Enter LOG window to learn more
- Cannot execute the operation. The system has not returned to the machine origin
- The parser is busy. Cannot execute the operation

9.2.1 Simulation Result Shows Running Range Exceeded Machine Travel. Enter Log Window to Learn More

Cause

The workbench travel in the program file exceeds the set upper limit and lower limit of workbench travel.

Solution

1. Check if the workpiece origin is set reasonably:
   - Yes: Proceed to next step.
   - No: Set it again.
   - See Set the Workpiece Origin for details.
2. Check if the workbench travel in the program file is reasonable:
   - Yes: Modify parameters **Workbench Upper Limit** and ***Workbench Lower Limit** to enlarge the workbench travel.
   - No: Modify the workbench travel in the program file.

9.2.2 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 Return to the Machine Origin for details.

9.2.3 The Parser is Busy. Cannot Execute the Operation

Cause

Cannot execute the operation in the current machining status.

Solution

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

9.3 Questions about Alarms

It includes the following:

- X (Y/Z/A/B/C) positive (negative) limit alarm
- X (Y/Z/A/B/C) servo alarm
- ESTOP alarm
- Failed to execute machining
- The terminal board is disconnected

9.3.1 X/Y/Z/A/B/C Positive (Negative) Limit Alarm

Cause

1. Incorrect polarity of port **Negative Limit** / **Positive Limit**.
2. X-axis / Y-axis / Z-axis / A-axis / B-axis / C-axis collided with the limit switch.
3. Failure of the limit switch.

Solution

1. Modify the polarity of the corresponding port.

   See Check Port Information for details.
2. Manually move the corresponding axis away from the limit switch.
3. Replace the limit switch.

9.3.2 X/Y/Z/A/B/C Servo Alarm

Cause

1. Incorrect polarity of port **Servo Alarm**.
2. The alarm of X-axis / Y-axis / Z-axis / A-axis / B-axis / C-axis in the servo drive occurs.

Solution

1. Modify the polarity of the corresponding port.

   See Check Port Information for details.
2. Check servo connecting wires of X-axis / Y-axis / Z-axis / A-axis / B-axis / C-axis.

9.3.3 ESTOP Alarm

Cause

1. The **ESTOP** button is pressed.
2. Incorrect polarity of port **Emergency Stop**.

Solution

1. Rotate **ESTOP** button clockwise.
2. Modify the polarity of port **Emergency Stop**.

   See Check Port Information for details.

9.3.4 Failed to Execute Machining

Cause

The software has expired.

Solution

Register the software again.

See Register the Software for details.

9.3.5 The Terminal Board is Disconnected

Cause

1. Incorrect polarity of port **Terminal Board Disconnection / Extended Terminal Board 1 Disconnection / Extended Terminal Board 2 Disconnection**.
2. Cables of terminal board have not been firmly connected.
3. Failure of Lambda terminal board.

Solution

1. Modify the polarity of corresponding port.

   See Check Port Information for details.

2. Re-plug cables of terminal board and restart the software.
3. Check if SYSTEM indicator light of Lambda terminal board is on:
   - Yes: contact us.
   - No: replace a new terminal board.

10 Legal Notices

Important—Read Carefully before Using This Product:

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.

10.1 Description of Further Rights and Restrictions:

- You may install for use one copy of the Software Product on a single machine;
- 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;
- 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;
- 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;
- 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;
- You may NOT sublicense, assign or transfer the license agreement;
- You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
- You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
- 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.

10.2 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 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.

10.3 After-sales Guarantee:

Weihong Company guarantees that for 90 days from the date of shipment the software carrier will be free from defects in materials and workmanship. When such a defect has been confirmed, our only responsibility is to replace the software carrier. This remedy is your exclusive remedy. This after-sales guarantee is invalid for any carrier defect caused by accidents, abuses or maloperation. The replaced software carrier enjoys the remaining guarantee time of the original software carrier or of a 30-day guarantee time, whichever is longer. Except as the after-sales guarantee made above, the Software Product does not enjoy any other form of after-sale guarantee.

10.4 Limitation of Liability:

The above guarantee, whether made explicitly or by implication, constitutes the entire contents of the guarantee, including guarantee for the commerciality and applicability of special application aim. Whether you follow other terms in this agreement or not, Weihong Company, as well as its agents and sales staff, will not be responsible for any profits loss, availability loss, business break-off or any forms of indirect, special, accidental or inevitable damage or claim made by any third party, generated from the using of the Software Product, even if Weihong Company has been informed of the possible occurrence of such events in advance.

10.5 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.

10.6 Applicable Law:

Copyright Law, Regulations for the Protection of Computer Software, Patent Law and other relevant laws and regulations. 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

Shanghai Weihong Electronic Technology Co., Ltd.

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

Hot-line: 400 882 9188

Website: www.weihong.com.cn/en