NcStudio V10 Engraving and Milling Control System Operation Manual (Dedicated for Four-axis and Dual-Z Three-axis Machine in Metal 3C Industries)

Version: 2021.09.14 1st edition
Author: Product Testing Department

Contents

1 System Overview

NcStudio V10 Engraving and Milling Control System is *an* engraving machine control system developed by **Shanghai Weihong Electronic Technology Co.,Ltd**, who owns its copyright.

It can be used on different types of 3D engraving machine in complex mold machining, advertisement, decoration, and glass edge grinding businesses.

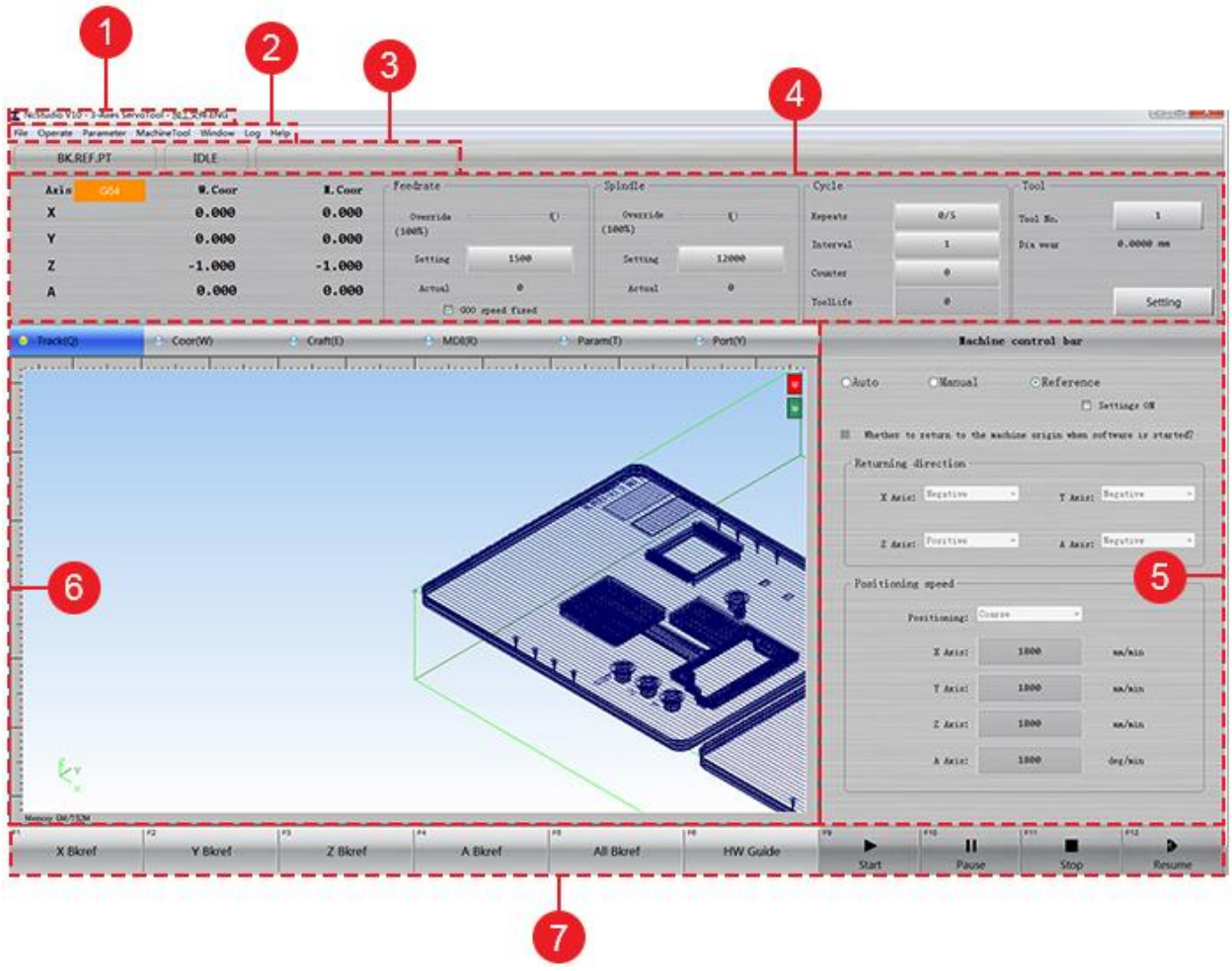
1.1 Software Type

NcStudio V10 Engraving and Milling Control System supports several types of software that are applicable to machine with different numbers of axes. This manual will introduce details on the following two types of software:

Software Name	Supported Axis
NcStudio V10 general four-axis software	Axis X, Y, Z, and A/B/C
NcStudio V10 three-axis dual Z-axis software	Axis X, Y, Z1, and Z2

1.2 Software Interface

The interface of the *NcStudio V10 Engraving and Milling Control System* software is shown below:



1. Title bar
   It shows the software name, current configuration, and name of the currently loaded machining program.
2. Menu bar

Menu	Description
File	Includes commands such as loading and unloading programs, loading objects, generating installation package, choosing configuration, and closing the software or system.
Operate	Includes common machining commands, such as starting, pausing, breakpoint resume, single block machining, going back to machine origin.
Parameter	Includes parameter setting commands.
Machine Tool	Includes commands controlling machining and machining ports, such as axis error compensation and main spindle commands.
Window	Includes commands for switching between screens of different machining modes, switching between sub-tabs of the **Coor** tab, adjusting the display sequence of the function tabs.
Log	Includes the command to open the system log.
Help	Includes commands for viewing NcStudio V10 information and registration.

3. Status bar

Info Type	Description
Current operation mode	E.g. Auto, Manual or Reference
Current system status	E.g. Running, Idle, or Emergency Stop
Whether running normally	E.g. Normal running or Exceptional termination
System prompt or alarm	E.g. The system ongoing or next step, errors, and alarms

4. Info region

Area	Description
Current position	Shows the name of the current workpiece coordinate system, and the mechanical and workpiece coordinates of the axes.
Feed rate	Shows the current feed rate override (multiplier), feed rate setting value, and the actual feed rate of the spindle. The override and setting value can be manually adjusted.
Spindle	Shows the current rotational speed override (multiplier), rotational speed setting value, and the actual rotational speed of the spindle. The override and setting value can be manually adjusted.
Cycle	Shows adjustable cycle machining parameters.

Area	Description
Tool	Shows the current tool No., tool length and diameter worn amount, and tool compensation setting icon.

5. Machine control region

Operation mode	Description
Auto	The machine executes the program automatically. Shows the machining file and operation buttons.
Manual	Shows the axis direction buttons, feed options, and operation buttons. In this mode, you can control machine movement with a hand wheel (HW).
Reference	The default operation mode. Shows the **Returning Direction** parameters (for going back to machine origin) and **Positioning speed** parameters (during going back to machine origin).

6. Function tabs

Tab	Description
Track	Shows real-time machining path for your inspection.
Coor	Include three sub-tabs: **Offset**, **Calibrate**, **Center**.
Craft	Shows adjustable cycle machining and array machining parameters, and part compensation icon.
MDI	You can click the **Edit** buttons and enter commands in the blank fields, and then click the **Execute** buttons to execute the commands.
Param	Shows the list of common parameters. You can add a parameter into the list by going to **Parameter** > **Set Parameters** in the menu bar, selecting the target parameter, and clicking **Set Common On**.
Port	Shows the I/O port status, and status of communication between the software and hardware.

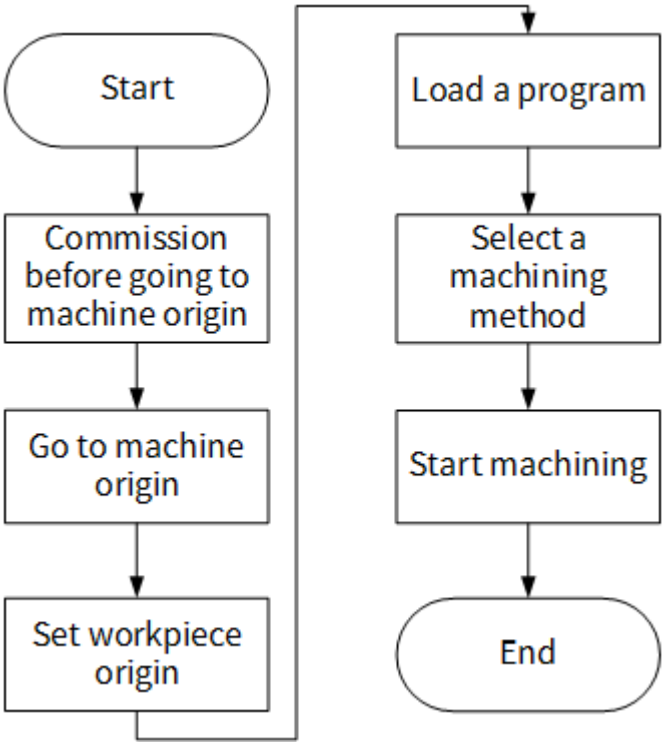
7. Operation button bar

Button	Description
F1–F6	The functions of the buttons vary based on the operation mode.
F9–F12	Have fixed functions: **F9**: start; **F10**: pause; **F11**: stop; **F12**: resume from break point.

2 Quick Start

This chapter provides an introduction of the *NcStudio V10 Engraving and Milling Control System* to help you get a quick start.

The machining process using the *NcStudio V10 Engraving and Milling Control System* is shown below:



1. Commission before going to machine origin
2. Go to machine origin or set datum
3. Set workpiece origin
4. Load a program
5. Select a machining method
6. Start machining

2.1 Commission Before Going to Machine Origin

This step mainly includes the following three operations:

1. Adjust the pulse equivalent.
2. Check the machine axis directions.
3. Set the workbench travel limits.

Note: The pulse equivalent, machine axis direction, and workbench travel limit parameters cannot be viewed or adjusted before entering the manufacturer password.

2.1.1 Adjust Pulse Equivalent

The smaller the pulse equivalent, the higher the machining accuracy and workpiece surface quality. The larger the pulse equivalent, the larger the machine maximum feed rate.

Therefore, set the pulse equivalent to a value as small as possible under the condition that the feed rate meets the requirement.

Prerequisite:

The **N50000 Control System Type** parameter is set to **0**.

Follow the steps below to adjust pulse equivalent:

1. Find the **N10010 AXIS_PULSE_FACTOR** parameter in the **Parameter** dialog box.
   For details, see section 2.3 Set Workpiece Origin.
2. Set the parameter to a small value.

After setting the parameter, restart the system.

2.1.2 Check Axis Direction

Prerequisite:

Axis movement is consistent with the values of the **N10010 AXIS_PULSE_FACTOR** parameters.

Follow the steps below to check the axis directions:

1. Check the value of the **N10000 AXIS_DIR** parameter.
   For details, see section 2.3 Set Workpiece Origin.
2. In the **Machine control region**, select **Manual** mode.
3. Select one from the following methods to control the machine to move in the directions of the axes:
   - Select **Jog**.
     Click and hold the axis direction buttons (X+, X-, Y+, Y-, Z+, Z-...) or press and hold the keyboard shortcuts to make the machine move along the axes at a low speed.
     Release the buttons to stop movement.
     Click and hold the axis buttons and the **Turbo** button at the same time to make the machine move along the axes at a high speed.
   - Select **HW**.
     Rotate the axis dial and gear dial of the hand wheel to select an axis and gear. Rotate the biggest control dial by certain blocks. The selected axis moves by a certain distance and then stops.
   - Select **0.01**, **0.1**, **1**, **10**, or enter a value as the step (the distance the machine moves per click of the axis direction button).
     Click the axis direction buttons.
4. Check to see if the axis movement directions are consistent with the direction set by the **N10000 AXIS_DIR** parameter.
   - Yes: The **N10000 AXIS_DIR** parameter is set correctly.
   - No: Set **N10000 AXIS_DIR** to the opposite of its current value.

2.1.3 Set Workbench Travel Limits

You need to set the workbench travel limit based on the machine's actual size to make the soft position limits function properly.

Follow the steps below to set the workbench travel limits:

1. Find the **N10020 WORKBENCH LOWER LIMIT** and **N10030 WORKBENCH UPPER LIMIT** parameters in the **Parameter** dialog box.
   For details, see section 3.3 Modify Parameters.
2. Set the **N10020 WORKBENCH LOWER LIMIT** and **N10030 WORKBENCH UPPER LIMIT** values based on the machine's actual size.
3. After setting the parameters, restart the system.

2.2 Go to Machine Origin or Set Datum

The software supports incremental encoders and absolute value encoders. Methods of going to the machine origin are different when using the two types of encoders.

Select the encoder type by setting the **ENCODER_TYPE** parameter.

- When using an incremental encoder, set the parameter to **0**.
  In this case, make the machine **go to the machine origin**.
- When using an absolute value encoder, set the parameter to **1**.
  In this case, **set the machine origin datum**.

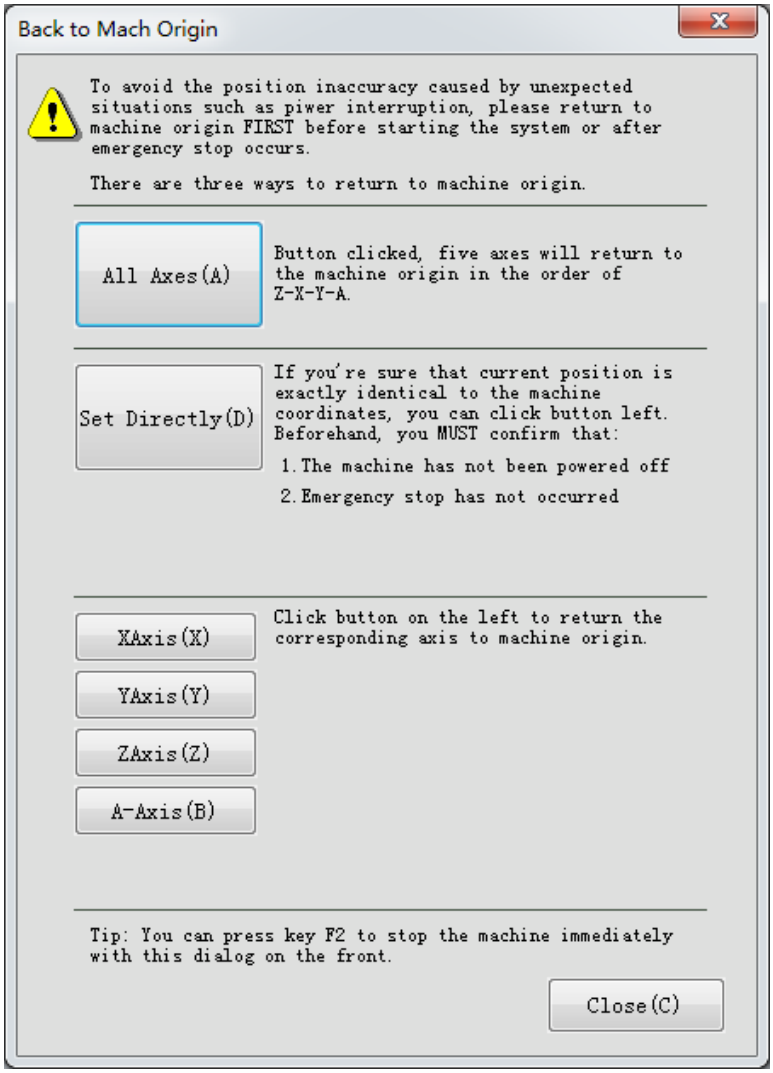
2.2.1 Go to the Machine Origin

Prerequisite:

Machine origin related parameters are set based on actual situation.

Follow the steps below to go to the machine origin:

1. In the **Machine control region**, select **Reference** mode.
2. Tick **Settings ON**. Enter the manufacturer password.
3. Decides whether to tick the **Whether to return to the machine origin when the software is started?** and set the machine origin related parameters (**Returning direction** and **Positioning speed** parameters) based on actual situation.
4. Select one of the following methods to make a single axis or all axes go to the machine origin (the default sequence is Z→X→Y→A):
   - In the **Operation button bar**, click **X Bkref/Y Bkref/Z Bkref/A Bkref/All Bkref** or press corresponding keyboard shortcuts.
   - In the menu bar, go to **Operate** > **Back to Mach Origin**. In the opened **Back to Mach Origin** dialog box, select one of the three methods to make a single axis or all axes go to the machine origin.



Note: Select **Set Directly** only when you are sure that the current position is correct.

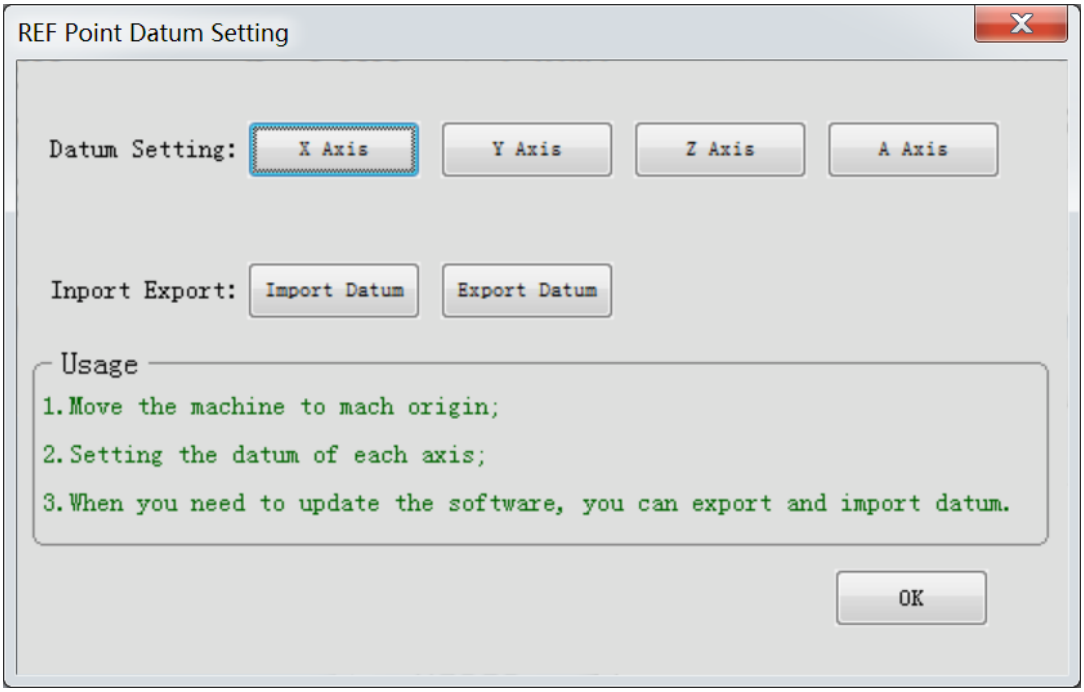
Result

After an axis goes to the machine origin, the icon will be displayed before the axis in the **Current position** region.

2.2.2 Set the Machine Origin Datum

Follow the steps below to set the machine origin datum

1. Manually control the machine to move to the machine origin.
2. In the menu bar, go to **Machine Tool** > **Mechanical Origin Datum Setting**. Enter the manufacturer password. The **REF Point Datum Setting** window is displayed.



3. Two methods are provided: **Datum Setting** and **Import Export**. Select one based on actual situation.
    - **Datum Setting**: Click the **X Axis**, **Y Axis**, **Z Axis**, and **A Axis** buttons in sequence to set datum for them. The system will read the encoder data and record the mechanical coordinates of each axis.
        - If the icon is displayed before axes in the **Current position** region, the datum is set.
        - A prompt requiring software restart will be displayed. Click **OK** and manually restart the software to make the datum setting take effect.
    - **Import Export**: Click the **Export Datum** button to export the datum file to the local path. Edit the datum file and save the modification. Click the **Import Datum** button to import the modified datum file. Manually restart the software.

Note: If the encoder battery box is powered off and on, you need to set the datum again.

2.3 Set Workpiece Origin

You can use two methods to set the workpiece origin in the *NcStudio V10 Engraving and Milling Control System*.

- Clearing
  Used to clear the workpiece coordinates of the current position when the machining accuracy requirement is not high.
- Centering
  Used to determine the workpiece origin for regularly-shaped workpieces and set the workpiece offset.
- Setting workpiece coordinate
  Used to write the mechanical coordinates of the current position into the workpiece offset.

2.3.1 Clearing

Follow the steps below to execute clearing:

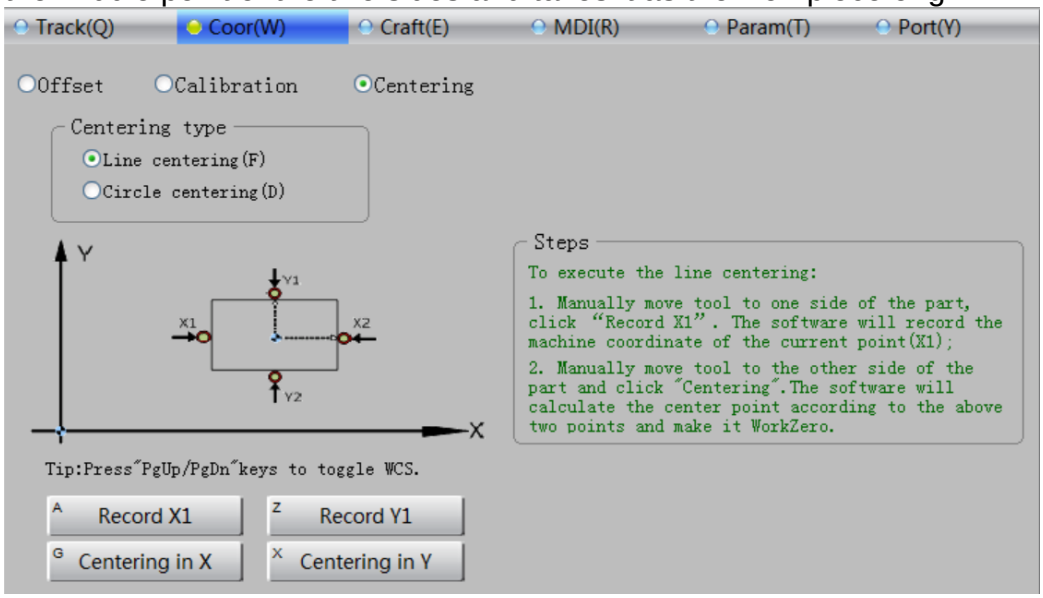
1. In the **Machine control region**, select **Manual** mode.
2. Move the machine axes to the position you want to set as the workpiece origin.

3. Select one from the following methods to execute clearing:
    - Clearing for a single axis or multiple axes: In the **Operation button bar**, click **X Clear**, **Y Clear**, **Z Clear**, **or XY Clear**.
    - Clearing for all axes: In the **Operation button bar**, click **All Clear**, or in the menu bar, go to **Operate** > **Set Workpiece Origin**.
4. A prompt asking for your confirmation will be displayed. Click **Yes**. The workpiece coordinates of the current position will be set to zero.

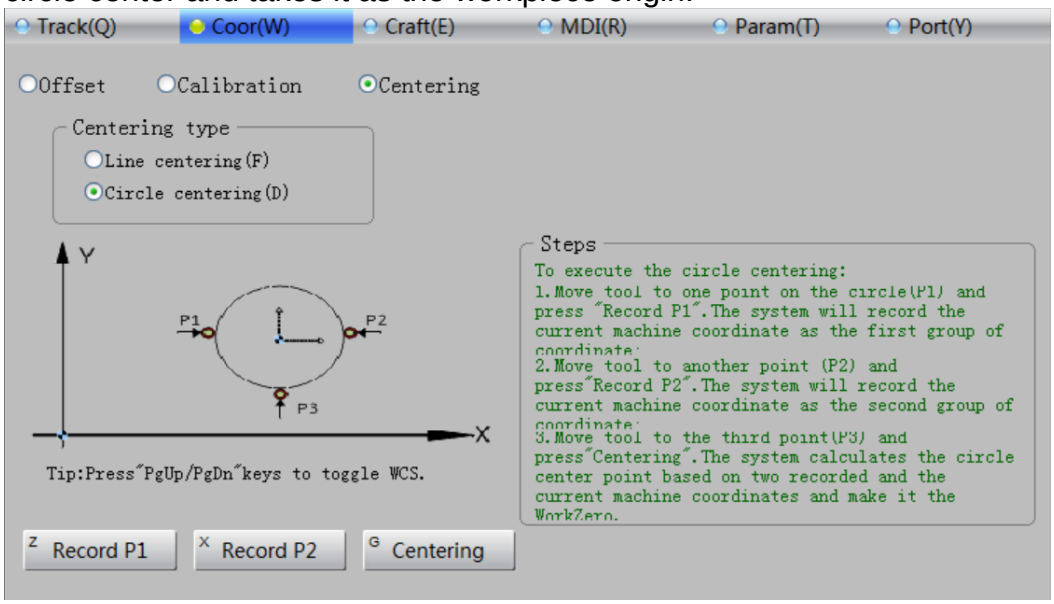
2.3.2 Centering

Follow the steps below to execute centering (taking X-axis centering as an example):

1. Click the **Coor** tab. Select **Centering**.
2. Select a centering method:
    - Select **Line centering** for regular rectangle workpieces. This method finds the middle point of the two sides and takes it as the workpiece origin.



    - Select **Circle centering** for circular workpieces. This method records the coordinates of three points on the circle and calculates the coordinates of the circle center and takes it as the workpiece origin.



3. In the **Machine control region**, select **Manual** mode. Select **HW**.
4. Move the tool to one side of the rectangular workpiece and click **Record X1**. This operation makes the software record the X-axis mechanical coordinate of the current point.
5. Move the tool to the other side of the rectangular workpiece and click **Centering in X**. This operation makes the software calculate the X-axis coordinate of the middle point based on the X-axis coordinate of the current position and that recorded in the previous step. The X-axis coordinate of the middle point will be set to zero.

Note: Step 4 and step 5 are operations for line centering. Operations for circle centering are similar. For details, see the step descriptions in the software.

2.3.3 Set Workpiece Offset

Follow the steps below to set workpiece offset:

1. Click the **Coor** tab. Select **Offset**.
2. In the **Parts offset** area, click **Set** behind the X, Y, and Z coordinates to set the X/Y/Z workpiece coordinates of the current point to zero.

2.4 Load Programs

You can use two methods to load programs into the *NcStudio V10 Engraving and Milling Control System*.

- Loading local programs
  Load programs stored in local paths.
- Using Program Wizard
  Load programs generated by the wizard to save preparation time.

The system supports the following program format:

- .nc
- .plt
- .eng
- .dxf

2.4.1 Load Local Programs

Prerequisite:

The needed machining program is prepared and put in a local path.

Use one of the three methods to load a local program:

- In the menu bar, go to **File** > **Open and Load**. Select the target program and click **Open**.
- In the **Machine control region**, select **Auto** mode. In the **Operation button bar**, click **Load** or press **F1** to load a program.
- In the **Machine control region**, select **Auto** mode. Right-click on the blank area and click **Open and Load** to load a local program.

If the following conditions are met:

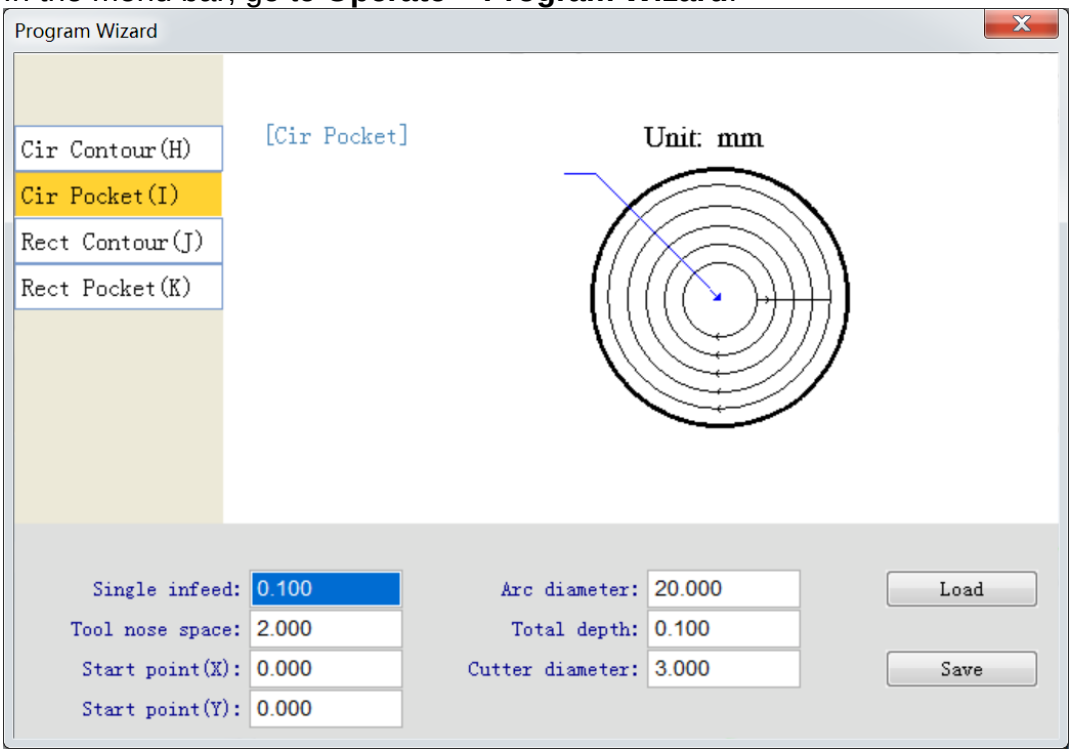
- The **N65203 MachiningEngFileByToolNumber** parameter is set to **Yes**.
- The **N65207 Modify By Tool Number** parameter is set to **Yes**.
- The loaded program is an ENG file.

The **Modify ToolNo for ENG** dialog box will be displayed when loading the file for you to confirm the tool No.:

2.4.2 Use Program Wizard

Follow the steps below to load a program generated by the program wizard:

1. In the menu bar, go to **Operate** > **Program Wizard**.



2. Select a wizard type:
   - Cir Contour

    - Cir Pocket
    - Rect Contour
    - Rect Pocket
3. Set the parameters in the red box.
4. Click **Load**. The old wizard file (.nc) in *D:\\NcFiles\\Wizards* will be overwritten. The new wizard file will be automatically loaded and its content displayed under the **Track** tab.

Note: In the **Program Wizard** window, if you click **Save**, the new file will overwrite the old file in *D:\\NcFiles\\Wizards* but will not be loaded automatically in the software.

2.5 Select Machining Method

The *NcStudio V10 Engraving and Milling Control System* supports three machining methods:

- Auto machining
  The system runs the program automatically for one time.
- Cycle machining
  The system runs the program in a cycle in **Auto** mode.
- Array machining
  The system quickly generates complex machining programs to improve efficiency. This method can be used to process neatly arranged work pieces.

2.5.1 Auto Machining

Follow the steps below to start auto machining:

1. In the **Machine control region**, select **Auto** mode.
2. Click **Start**. The system runs the program until the program end or it being ended manually.

2.5.2 Cycle Machining

Follow the steps below to start cycle machining:

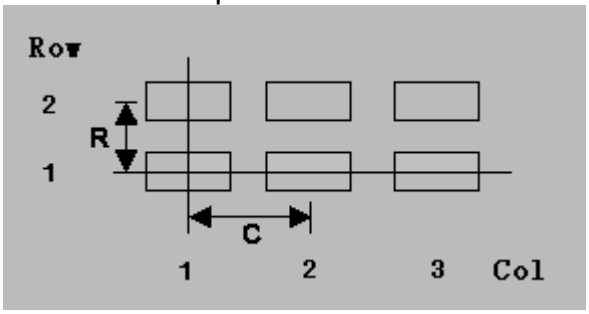
1. Use one of the following methods to find cycle machining parameters:
    - In the **Info region** and find the **Cycle** area.
    - Click the **Craft** tab and find the **Cycle Machining Settings** area.
2. Set the following parameters:
    - **Repeats/CycleTimes**
    - **Interval/CycleInterval**
3. Click **Start**. The system runs the program based on the repeat times and repeat interval set by you in the previous step.

2.5.3 Array Machining

Follow the steps below to start array machining:

1. Click the **Craft** tab.
2. In the **Array machining** area, tick **Array ON**.
3. Set the following parameters:
    - **RowNum**
    - **ColNum**
    - **RowSpace**
    - **ColSpace**
4. Click the **Set comp...** button to open the **Parts Compensation** window.
5. Tick **IsEnabled**. Set the parameters in the **Parameter**, **Value**, **Dir**, **Order**, and **Comp way** areas.

- When **Single part** is selected as the compensation method, there is no need to set the compensation direction (**Dir**).
  In the **Value** area, X01 and Y01 represents the X-axis and Y-axis compensation of the first work piece, X02 and Y02 represents the X-axis and Y-axis compensation of the second work piece, and so on.
  Compensation for up to 12 work pieces is supported.
- When **Array** is selected as the compensation method, select **By row** or **By col** as the compensation direction.



When **By row** is selected, X01 and Y01 represents the compensation for the first-row work pieces.
When **By col** is selected, X01 and Y01 represents the compensation for the first-column work pieces.
Compensation for up to 144 (12×12) work pieces is supported.

6. Close the window and click **Start**. The system runs the program based on the array machining settings.

2.6 Start Machining

Different machining methods of the *NcStudio V10 Engraving and Milling Control System* require different ways to start machining.

- Auto machining
  Used to run the program from beginning to end.
- Single block
  Used to run commands line by line. The system runs a command line and then pauses. You can use this method to check .nc programs for better error diagnosis and recovery.
- Selective machining
  Used to run target commands that may not be adjacent. You can use this method to check tool information, specify the machining range with the beginning and ending row numbers, specify the target track number, or specify the machining range with the beginning and ending track numbers.

2.6.1 Auto Machining

Follow the steps below to use auto machining:

1. Select one of the following methods to start auto machining:
   - In the menu bar, go to **Operate** > **Start**.
   - In the **Operation button bar**, click **Start.**
   - Press **F9**.
2. **Optional:** Select one of the following methods to pause machining:
   - In the menu bar, go to **Operate** > **Pause**.
   - In the **Operation button bar**, click **Pause.**
   - Press **F10**.

3. **Optional:** Select one of the following methods to stop machining:
    - In the menu bar, go to **Operate** > **Stop**.
    - In the **Operation button bar**, click **Stop.**
    - Press **F11**.

2.6.2 Single Block

Follow the steps below to use single block machining:

1. In the menu bar, go to **Operate** > **Single Block**.
2. Click **Start**. The system pauses after running a command block.
3. Click **Start** again to run another command block until all commands are executed.

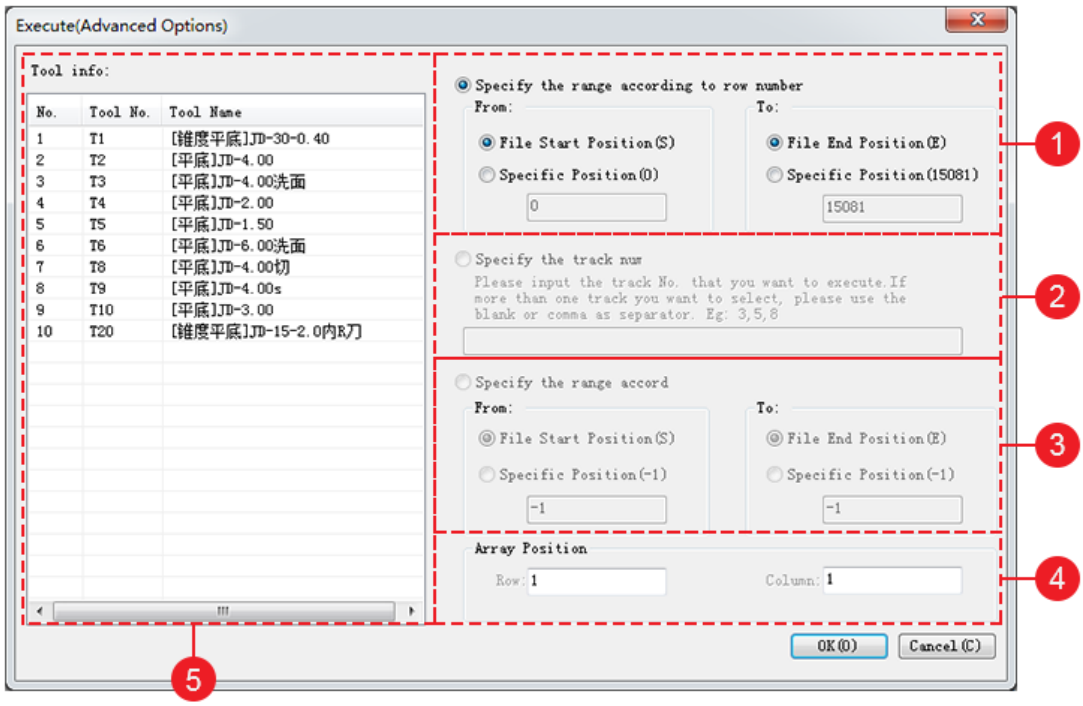
2.6.3 Selective Machining

The command rows selected during selective machining may not include tool change or spindle commands. Therefore, if no tool change or spindle command is inserted, a wrong tool can be used or machining can be started before the spindle starts rotation.

Therefore, before start selective machining, set the **N65300 Insert Tool and M345 Instruction Before Resuming** parameter to **Yes**. The system will automatically insert the tool change or spindle commands set last time during selective machining and machining resumed from the break point.

Follow the steps below to use selective machining:

1. In the **Machine control region**, select **Auto** mode.
2. Select one of the following methods to open the window for setting selective machining parameters:
    - In the menu bar, go to **Operate** > **Select Processing Block**.
    - In the **operation button bar**, click **GOTO**.



3. Specify the machining range:
    - By row range (①):

To start from or/and end at a specific row, select **Specific position** in the **From** or/and **To** areas and enter the desired start row number and end row number in the fields. The system runs the commands between the start and end rows.

- By track number (②):
  The system runs the commands of the specified track number.
- By track range (③):
  To start from or/and end at a specific track, select **Specific position** in the **From** or/and **To** areas and enter the desired start track number and end track number in the fields. The system runs the commands between the start and end tracks.

Note: You can specify machining range of G code files (such as .nc, .g, and .nce files) only by row range. You can specify machining range of .eng files either by row range or track range.

4. **Optional:** Check the tool information in the program in the **Tool info** region (⑤).
5. **Optional:** Set the machining range by specifying the array position (④).

3 Common Operations

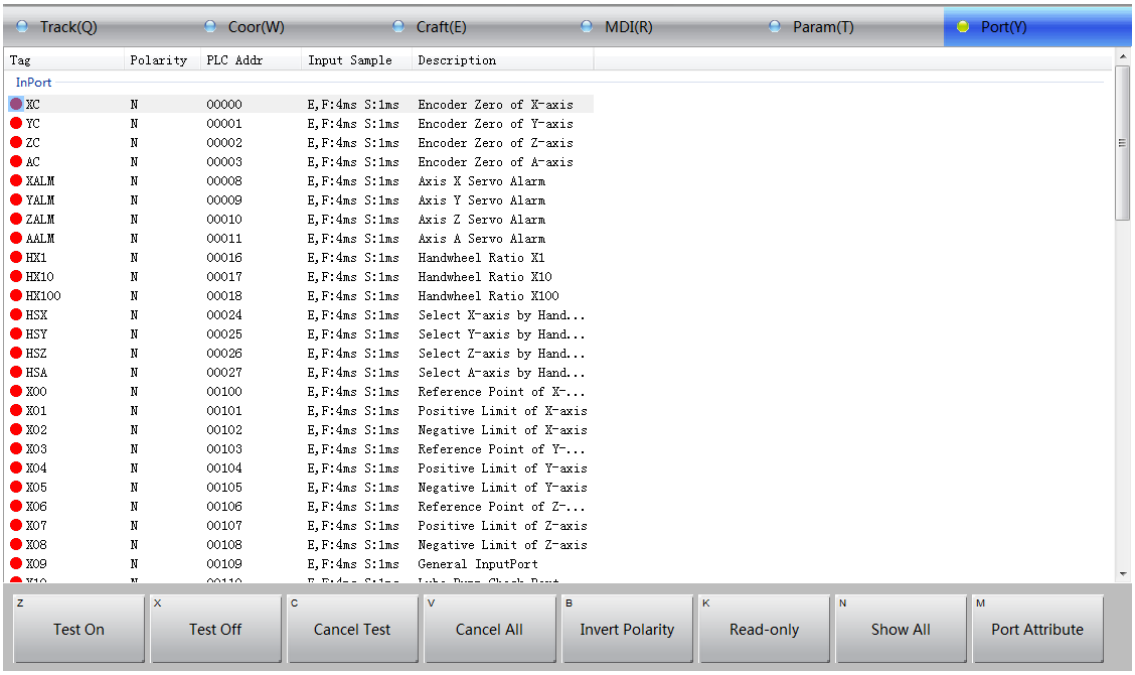
This chapter introduces common operations for using the *NcStudio V10 Engraving and Milling Control System*:

- Modify I/O port setting
- View the log
- Modify parameters
- View machining statistics
- Register the software
- Use NcCloud

3.1 Modify I/O Port Setting

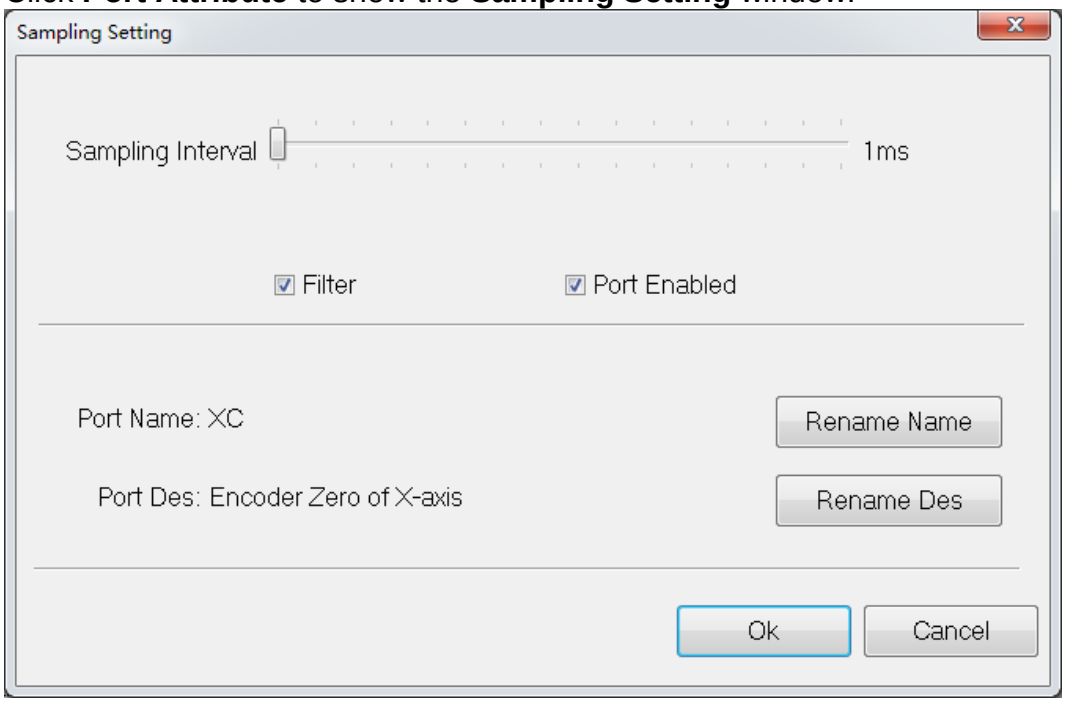
Follow the steps below to modify settings of a port:

1. Click the **Port** tab.



   - For input ports:
     🔴: no signal; 🟢: signal detected
   - For output ports:
     ⭕: no signal; 🟢: signal detected
2. On the **Port** page, you can:
   - Click **Test On** to simulate turning on the port.
     If an input port is selected, the symbol before it becomes 🟢. If an output port is selected, the symbol before it becomes 🟢.
   - Click **Test Off** to simulate turning off the port.
     If an input port is selected, the symbol before it becomes 🔴. If an output port is selected, the symbol before it becomes 🔴.
   - Click **Cancel Test**/**Cancel All** to cancel the test of the selected port/all ports.
   - Click **Invert Polarity** to change the polarity of the selected port to the opposite.
   - Click **Read-only** to set the selected port to read-only.
     The status of a read-only port cannot be changed.
   - Click **Show All** to display all ports supported by the hardware.

- Click **Port Attribute** to show the **Sampling Setting** window:

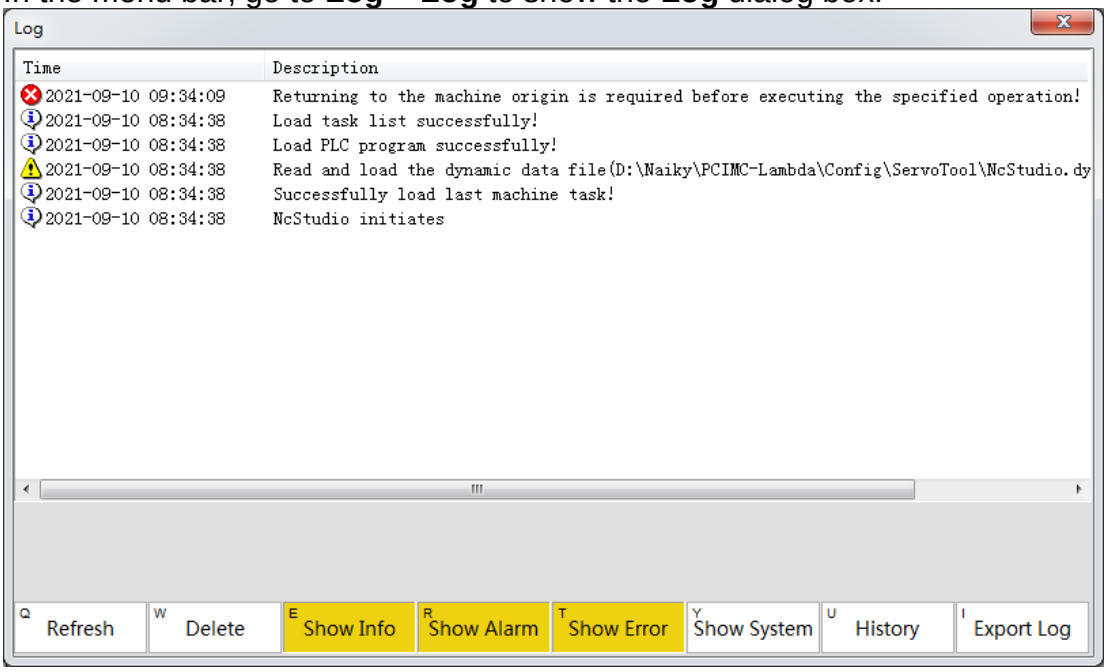


You can alter the port sampling interval, enable/disable the filtering function, and change the port name and description.

3.2 View the Log

Follow the steps below to view the log:

1. In the menu bar, go to **Log** > **Log** to show the **Log** dialog box:



2. **Optional:** In the **Log** dialog box, you can:
   - Click **Refresh** to refresh the log information.
   - Click **Delete** to delete all log information.

- Click **Show Info** (to make the button highlighted in yellow) to show all software running information (whose symbol is). This button is pressed down (highlighted in yellow) by default.
- Click **Show Alarm** (to make the button highlighted in yellow) to show all alarms (whose symbol is). This button is pressed down (highlighted in yellow) by default.
- Click **Show Error** (to make the button highlighted in yellow) to show all errors (whose symbol is). This button is pressed down (highlighted in yellow) by default.
- Click **Show System** (to make the button highlighted in yellow) to show all system information (whose symbol is).
- Click **History** (to make the button highlighted in yellow) to show all past log.
- Click **Export Log** to export the log to a local path.

Note: Delete the log regularly. If the log becomes too large, it can affect system performance and response speed.

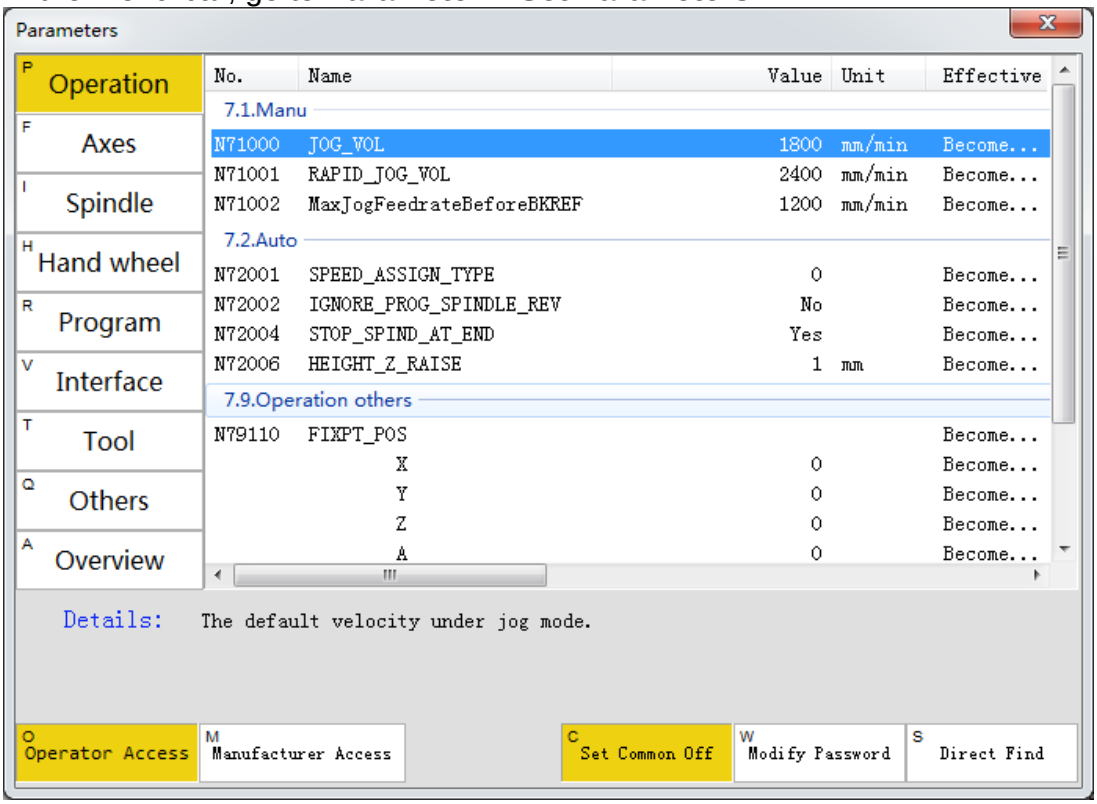
3.3 Modify Parameters

Based on user identity and permissions, the system has operator parameters and manufacturer parameters.

Operator parameters are displayed by default. The manufacturer password is required to access and modify manufacturer parameters.

Follow the steps below to view parameters:

1. In the menu bar, go to **Parameter** > **Set Parameters**.



2. In the **Parameters** window, you can alter parameter values:
   a. Double-click the target parameter line. An input box is displayed.
   b. Enter the desired value in the field and click **F1 OK**.

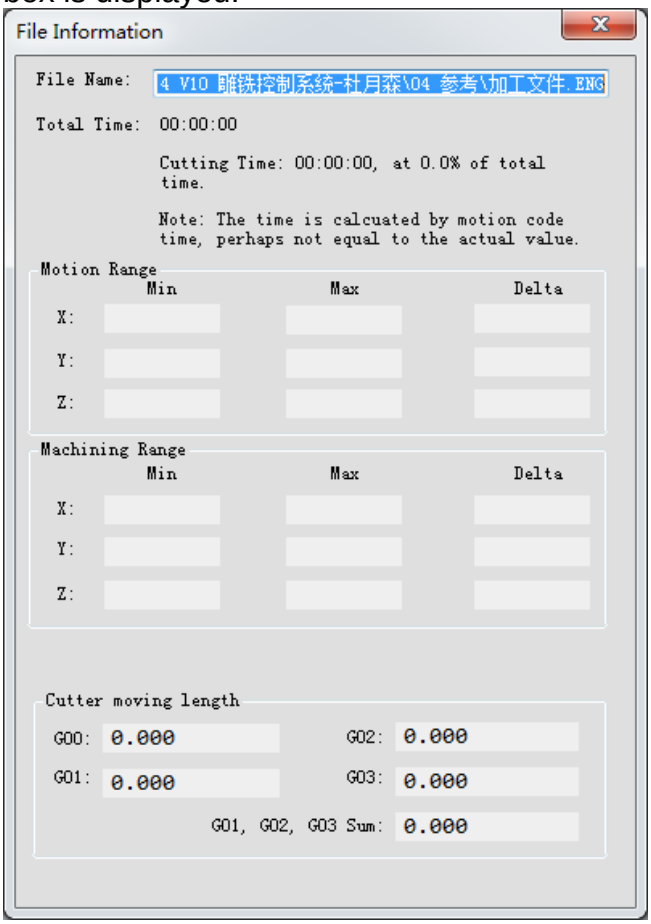
3. **Optional:** Search for the target parameter:
   a. Click **Direct Find**. The **Find Parameter** dialog box is displayed.
   b. Select **No./Name** and **Page/All** to search for the target parameter with its serial number/name in the current page/all pages.
4. **Optional:** Add a parameter to the list of common parameters:
   Select the target parameter and click **Set Common On** to add the parameter into the list of common parameters. The list of common parameters is shown in the **Param** function tab.
   Among the operator parameters, most of the **Operation** parameters and **Axes** parameters are in the list of common parameters.

3.4 View Machining Statistics

You can check the statistics in middle of machining to achieve real-time monitor of the machining process.

Follow the steps below to view machining statistics:

1. In the menu bar, go to **Operate** > **File Information**. The **File Information** dialog box is displayed.



2. You can check the following information:
   - Total time, cutting time and percentage of the cutting time in the total time
   - Axis motion range
   - Machining range
   - Tool movement distance

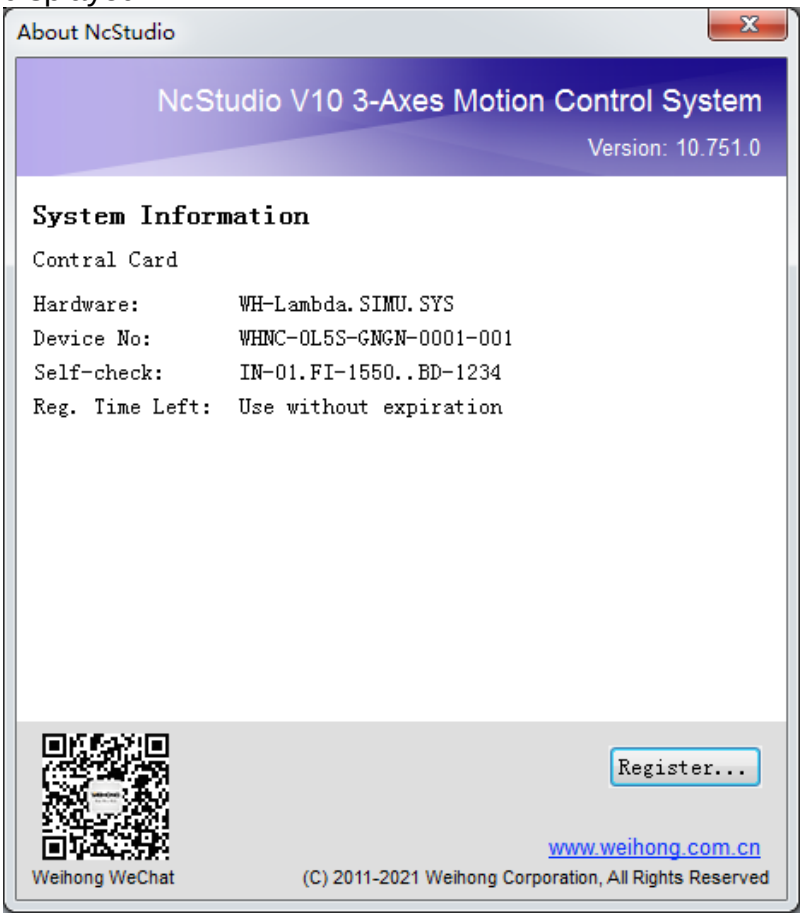
3.5 Register the Software

Prerequisite:

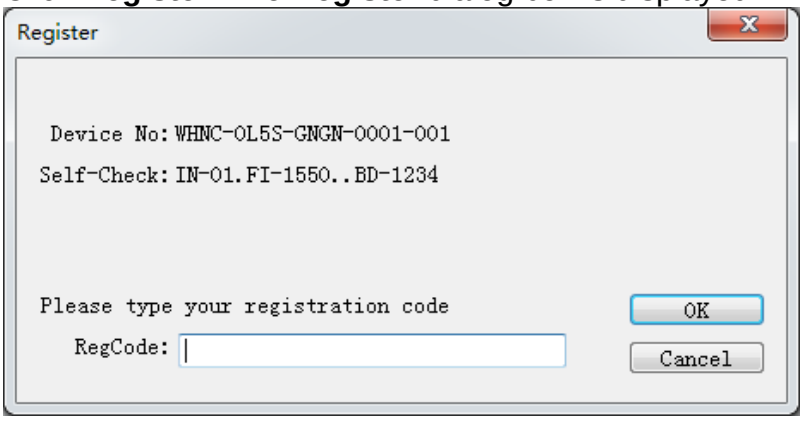
- The machine is not in middle of machining, which means that it is in idle or emergency stop status.
- You have acquired the registration code by sending the equipment No. to the supplier.

Follow the steps below to register the software:

1. In the menu bar, go to **Help** > **About NcStudio**. The **About NcStudio** window is displayed.



2. Click **Register**. The **Register** dialog box is displayed.



3. Enter the registration code in the field. Click **OK**. The registration success prompt is displayed.
4. Restart the software. The software is registered.

3.6 Use NcCloud

Weihong Industrial Internet of Things Solution (**Weihong IIoT Solution**) is a new basic solution for production management, equipment management, and business innovation based on monitor of digital control equipment.

Weihong IIoT Solution includes the following products and services:

- NcGateway
- xFactory
- NcCloud

NcCloud provides remote equipment monitor services based on public clouds. With one NcCloud account, you can use the services on web pages to meet monitor requirements in different production scenarios. Equipment information is synchronized across the two platforms.

Prerequisite:

- The *NcStudio V10 Engraving and Milling Control System* is installed and the software opened.

  After the installation is complete, two icons (

and 

) are displayed on the desktop.
- You have an NcCloud account.
  If you do not have an NcCloud account, refer to the NcCloud user manual to register a new account.
- The computer is connected to the Internet.

Follow the steps below to use NcCloud:

1. Log in to NcCloud: Open the Weihong official website (https://www.weihong.com.cn/en/) and click NcCloud. Click **Login/Register** and enter the information.
2. Click the desktop icons 

> 

, the NcCloud screen is displayed.

3. Add the equipment: Go to **All Equipment** > **Add Equipment**. Enter the equipment number and verification code as shown in box ③. Enter an additional name for the equipment.

   Note: After adding a piece of equipment, you can press **Esc**/**Enter** to exit the screen, which will not affect data transmission between the equipment and the cloud.

After a piece of equipment is added and the production monitor function is enabled, you can check the machine real-time running status on web pages, the information on which is synchronized.

4 Troubleshooting

This chapter lists common problems that may happen in the *NcStudio V10 Engraving and Milling Control System* and corresponding solutions to help you troubleshoot:

- Common problems and solutions
- Common alarms and solutions

4.1 Common Problems and Solutions

Some common problems are listed below:

- The spindle does not rotate
- An axis does not move
- The Z-axis servo motor brake cannot be turned on
- Going to machine origin exception
- Tool moves upward after reaching the tool sensor position during tool calibration

4.1.1 Spindle not Rotate

Troubleshoot as follows:

In the menu bar, go to **MachineTool** > **Main Spindle**. Check to see if the spindle indicator of the controller is turned on:

- ON: Use a multimeter to measure if the spindle ON ports (Y01–C01) are conductive and if the analog voltage output between AVC and GND is normal.
  If the ports are conductive and the voltage output is normal, check to see if the inverter parameters are set correctly, if the spindle and inverter work normally and if the spindle and inverter are connected properly.
- OFF: Turn off the computer and power off the machine. Disconnect and reconnect the controller power cable.
  If the indicator is still off, replace the Lambda controller or NC65C control card.

4.1.2 Axis not Move

Troubleshoot as follows:

1. Click the **Port** function tab. Check to see if the polarity of the output port **Axis X/Y/Z/A Servo Enable** is **N**.
2. Check to see if the parameters of the servo drive are set correctly: the control method should be set to position control, pulse input should be pulse plus direction, the electronic gear ratio should be set correctly, etc.
3. Check to see if the axis servo cable is securely connected to the controller and servo drive.
4. Check to see if the motor is enabled.
5. Manually control the axis to move and see if the drive receives pulses.
   - If the drive receives pulses but the machine does not move, check to see if the movement transmission mechanism is loose.
   - If the drive does not receive pulses, replace the upper computer or drive.

4.1.3 Z-axis Servo Motor Brake Cannot be Turned on

Troubleshoot as follows:

1. Check to see if the **Brake of Z-axis** port has signal.
   - No signal:
     Check to see if the servo drive is enabled and if the brake output parameters of the servo drive are set correctly.
   - Signal detected:

Remove the connection cables of the brake output ports (Y00–C00). Start the system and power on the machine (rule out system alarm signals). Use a multimeter to measure if the ports are conductive.
Not conductive: check the system of the upper computer.
Conductive: the brake output ports are normal.

2. Power off the machine and reconnect the removed cables (and connect the 24V power supply of the original loop). Use a multimeter to measure if the voltage of the two ends of the motor-side brake cable is 24V.
   If the voltage is 24V, the motor is damaged.
3. If the fault persists, replace the Lambda controller.

4.1.4 Going to Machine Origin Exception

- Symptom 1: Position limit alarms or servo drive alarms are reported when going to machine origin.

Troubleshoot as follows:

1. Click the **Port** function tab. Ensure that the polarity of the **Reference Point of X-axis/Y-axis/Z-axis** input port is consistent with the origin switch signal type (N: normally open; P: normally closed).
2. Manually control the axis to move to the origin switch position. On the **Port** page, check to see if 🟢 is displayed before the **Reference Point of X-axis/Y-axis/Z-axis** input port.
   If 🟢 is not displayed, the software cannot receive signals from the origin.
   Check to see if the origin switch and the origin switch cable connection are normal.
   Use a conducting wire to connect the controller origin signal to the COM port (the origin switch is 24V when active HIGH). On the **Port** page, check to see if 🟢 is displayed before the **Reference Point of X-axis** input port.
3. In the **machine control region**, select **Reference** mode. Check to see if the **Returning direction** parameters are opposite to the direction set by the **N74080 BACK_DISTANCE** parameter.
4. Check the following items:
   - Whether the origin switch position is proper
   - Whether the origin switch is too close to the position limit switch
   - Whether the origin switch is behind the position limit switch
   - Whether the origin switch is out of the machine mechanical travel range

- Symptom 2: When going back to origin, the axis keeps moving in one direction at a small speed (one tenth of the coarse positioning speed) until triggering position limit alarm.

Troubleshoot as follows:

Click the **Port** function tab. Check to see if the **Reference Point of X-axis/Y-axis/Z-axis** input port has signal input:

- Signal input detected: 🟢
- No signal: 🔴
- Symptom 3: When going to machine origin, the axis moves at a small speed in the opposite direction for a long distance or keeps moving in the opposite direction after coarse positioning.

The reason is that the system cannot detect the axis encoder zero signal.

Troubleshoot as follows:

1. Manually control the axis to move. Click the **Port** tab, check to see if the **Encoder Zero of X-axis/Y-axis/Z-axis/A-axis** input port has signals.
2. Check to see if the axis servo cable is properly connected to the Lambda controller and servo drive.
3. Check to see if the drive, motor, encoder cable, servo cable, and the control system are normal, such as exchanging the servo cable or servo drive with that of axes that can go to machine origin normally.

4.1.5 Tool Moves Upward After Reaching Tool Sensor During Tool Calibration

Troubleshoot as follows:

1. Click the **Port** tab, check to see if the polarity of the **Cut Signal of Z-axis** input port is consistent with the tool sensor signal type.
2. Manually press and hold the tool sensor. Check to see if the **Cut Signal of Z-axis** input port has signal input (●).
   If the port has no signal, the tool sensor is damaged.

4.2 Common Alarms and Solutions

Exception reports in the *NcStudio V10 Engraving and Milling Control System* include:

- Warnings 
- Alarms 

4.2.1 Warnings

4.2.1.1 Simulation Result Shows that the Program Range Exceeds the Mechanical Travel Limits

Cause:

The machining range in the program exceeds the workbench travel limits set by the **N10020 WORKBENCH_LOWER_LIMIT** and **N10030 WORKBENCH_UPPER_LIMIT** parameters.

Solution:

1. Check to see if the workpiece origin position is proper.
2. Check the program.
3. Modify the values of the **N10020 WORKBENCH_LOWER_LIMIT** and **N10030 WORKBENCH_UPPER_LIMIT** parameters to expand the workbench travel range.

4.2.1.2 Returning to the Machine Origin is Required Before Executing the Specified Operation

Cause:

The axes are not at the machine origin (the **N74001 NEED_REFPT_BEFORE_MACHINING** parameter is set to **Yes**).

Solution:

Make the axes go to the machine origin first.

4.2.1.3 Code Interpreter is Busy. Cannot Execute the Operation in Current Status.

Cause:

Illegal actions are performed during machining, such as modifying a parameter.

Solution:

Stop machining and perform the actions when the machine is idle.

4.2.1.4 No Machining File

Cause:

No program is loaded.

Solution:

Load a machining file.

4.2.2 Alarms

4.2.2.1 X/Y/Z Axis Positive/Negative Limit Alarm

Cause 1:

Polarity of the **Positive/Negative Limit of X-axis/Y-axis/Z-axis** port is wrong.

Solution:

Change the port polarity (on the **Port** page).

Cause 2:

The X/Y/Z axis collides with the position limit switch during movement.

Solution:

Manually control the axis to move away from the position limit switch.

Cause 3:

The position limit switch has exception.

Solution:

Check to see if the position limit switch is normal.

4.2.2.2 X/Y/Z Axis Servo Alarm

Cause 1:

Polarity of the **Axis X/Y/Z/A Servo Alarm** port is wrong.

Solution:

Change the port polarity (on the **Port** page).

Cause 2:

The X-axis/Y-axis/Z-axis/ servo drive reports an alarm.

Solution:

Check to see if the axis servo connection cable is normal.

4.2.2.3 Emergency Stop Alarm

Cause 1:

The emergency stop button is pressed down.

Solution:

Rotate the button to release it.

Cause 2:

Polarity of the **Emergency Stop** port is wrong.

Solution:

Change the port polarity (on the **Port** page).

4.2.2.4 Spindle Alarm

Cause 1:

Polarity of the **Spindle Alarm** port is wrong.

Solution:

Change the port polarity (on the **Port** page).

Cause 2:

The inverter reports an alarm.

Solution:

Locate the problem source based on the inverter alarm type.

4.2.2.5 Terminal Board Disconnection Alarm

Cause 1:

The terminal board is not connected properly or the Lambda controller has hardware faults.

Solution:

1. Remove and reconnect the cable and restart the software to see if the fault disappears.
2. Change the port polarity and restart the software to see if the port polarity was wrong.
3. Locate the problem source based on the status of the Lambda controller SYSTEM indicator.
4. Replace the Lambda controller.

Cause 2:

Polarity of the **Terminal Block Not Connect Alarm** port is wrong.

Solution:

Change the port polarity (on the **Port** page).

Note: Some alarms are added based on the machine mechanical structure and not explained on the user interface. If you have any questions, please contact the machine manufacturer.

5 Legal Notices

The term “Software Product” includes all copies of the licensed software and its documentation. This license agreement is a legal agreement between You (either an individual, a legal entity or any affiliated companies or other entities) and Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). By installing, copying, or otherwise using the Software Product, you agree to be bound by the terms and conditions of this license. Unless otherwise stated in this agreement, you shall not use, copy, revise, rent, or transfer the Software Product or any part of the Software Product for any other purposes.

5.1 Requirements of Use

1. You may install for use one copy of the Software Product on a single machine;
2. You may make a copy of the Software Product for archive or backup purposes and the copy is to be used on this machine only;
3. You may transfer the Software Product and the license agreement to a third party, provided that the third party accepts the terms and conditions stated in this agreement, with prior express permission from Weihong Company;
4. When transfer confirmed, you shall transfer all the copies of the original documents and the supplementary documents to the third party or destroy all the copies un-transferred;
5. You can use the Software Product on a network server or intranet server only if it is stipulated in explicit terms that you are allowed to use the Software Product on a network server or intranet server, or you have purchased license for each node and terminal using the Software Product;
6. You may NOT sublicense, assign or transfer the license agreement;
7. You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
8. You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
9. The license agreement shall be terminated automatically upon you transfer the Software Product or copies of all or part of the Software Product to a third party.

5.2 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 Chinese copyright law and international treaty provisions as well as on the basis of other laws and agreements regarding intellectual property. You are not allowed to remove the copyright statement made in the Software Product, and guarantee that you shall copy the copyright statement in all copies of the Software Product or of any part of the Software Product. You are obliged to stop any form of illegal copying of the Software Product and accompanying materials.

Weihong Company can update the Software Product at any time. You can pay real-time attention to Weihong Company's official website.

5.3 Termination

This license may be terminated by Weihong Company at any time once you violate any terms or conditions made in this agreement. Once the license is terminated, you are obliged to destroy all the copies of the Software Product or return them to Weihong Company.

Now, we affirm that you have already read through this agreement and understood it thoroughly and agreed to comply with all of the terms and conditions of this agreement strictly.

Weihong Electronic Technology Co., Ltd.

SPECIALIZED / CONCENTRATED / FOCUSED

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