Manufacturers' Manual of Ncstudio CNC System for Cutting Machines

1   System Overview

1.1   Overview

WEIHONG **NcStudio CNC System for Cutting Machines** is a CNC solution to cutting machines in woodworking industry.

The system consists of the following:

Hardware

- Industrial computer: NC60A
- Lambda terminal board:
  - o Lambda 21A: applicable to the non-bus control system.
  - o Lambda 21B: applicable to the bus control system.
- Extended terminal board: EX31A
- Servo drive
- Servo motor
- Inverter
- Handwheel

For the connection of each hardware, see Connection Diagram.

Software

- **NcConfig** software: mainly used for commissioning.
- **NcTune** software: mainly used for commissioning.
- **NcStudio** software: mainly used for machining.

For the layout of the main software interfaces of **NcStudio CNC System for Cutting Machines**, see Main Software Interface.

1.2   Connection Diagram

The connection diagram differs in the type of control system:

Connection diagram for the non-bus control system

The connection diagram for the non-bus control system is as follows:
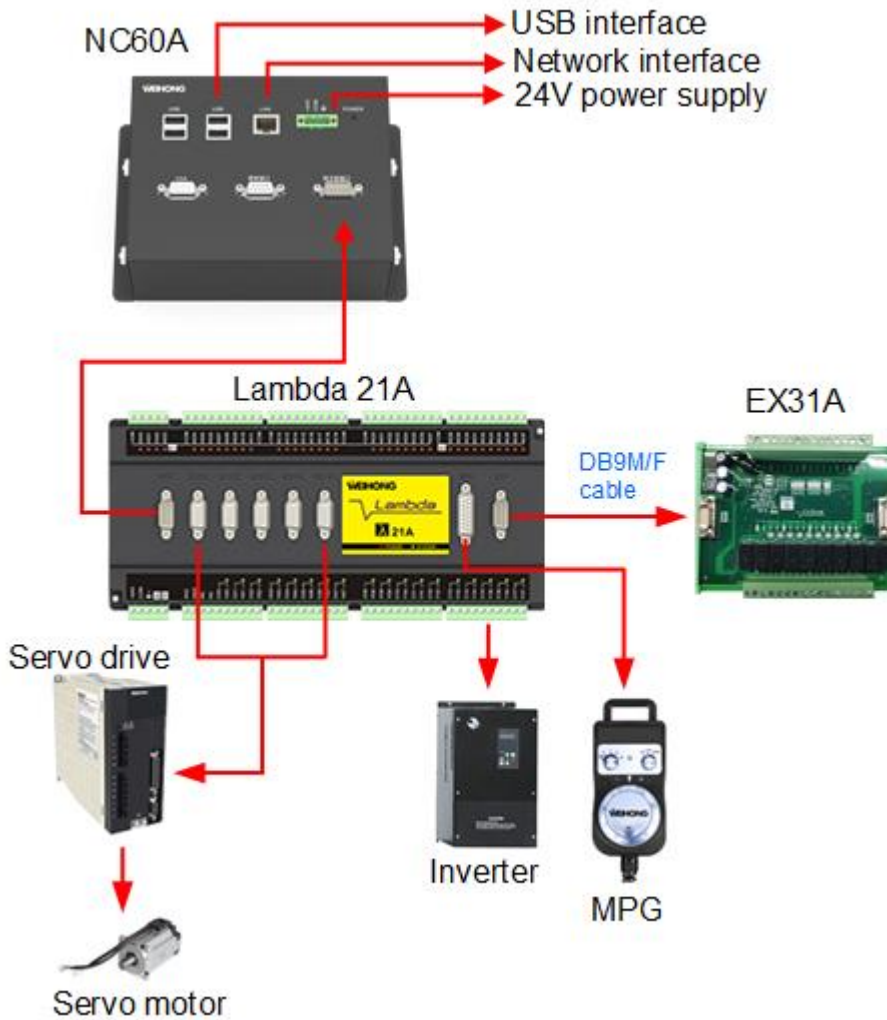
Connection diagram for the bus control system

The connection diagram for the bus control system is as follows:
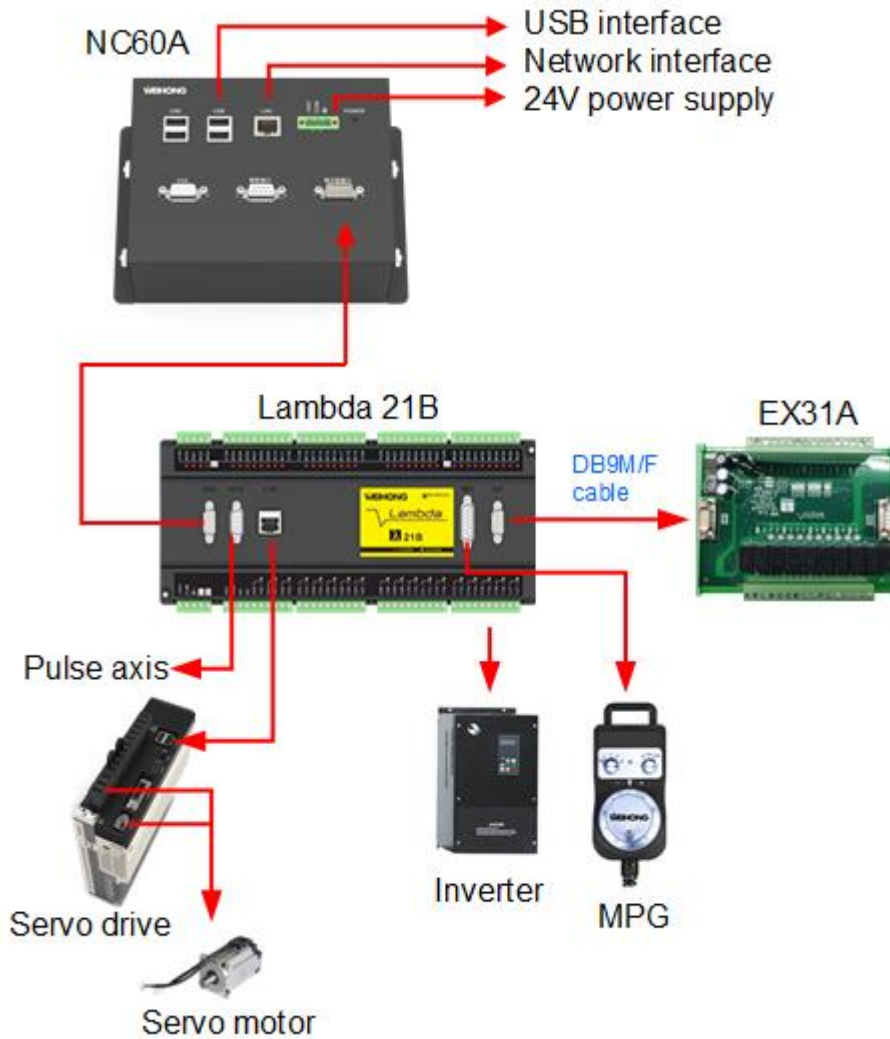
1.3 Main Software Interface

Through this part, you can quickly know the main software interfaces of WEIHONG **NcStudio CNC System for Cutting Machines**.

1.3.1 Main Interface of NcConfig Software

The main interface of **NcConfig** software is as follows:
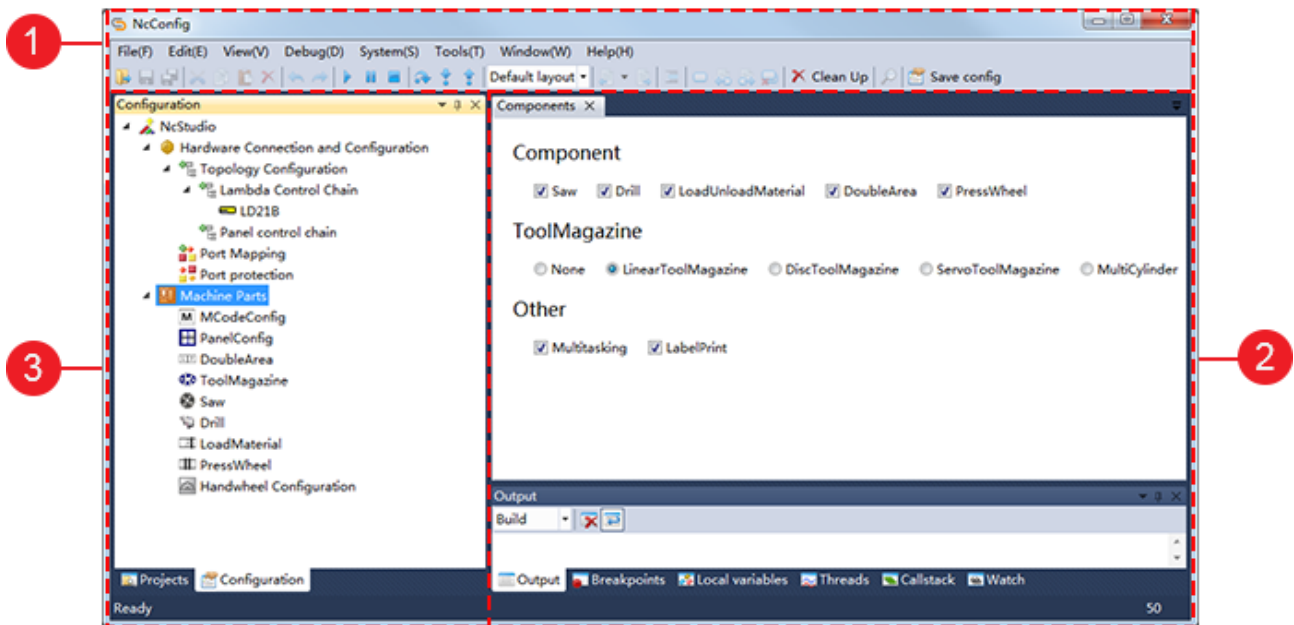
1. Menu bar
2. Function display bar
3. Configuration bar

1.3.1.1 Function Display Bar

You can do function commissioning in this bar after selecting the target function in the configuration bar.

1.3.1.2 Configuration Bar

In this bar, you can do the following:

- Configuring devices
- Setting machine parts
- Setting port mapping
- Setting port protection

1.3.2　Main Interface of NcTune Software

The main interface of **NcTune** software is as follows:
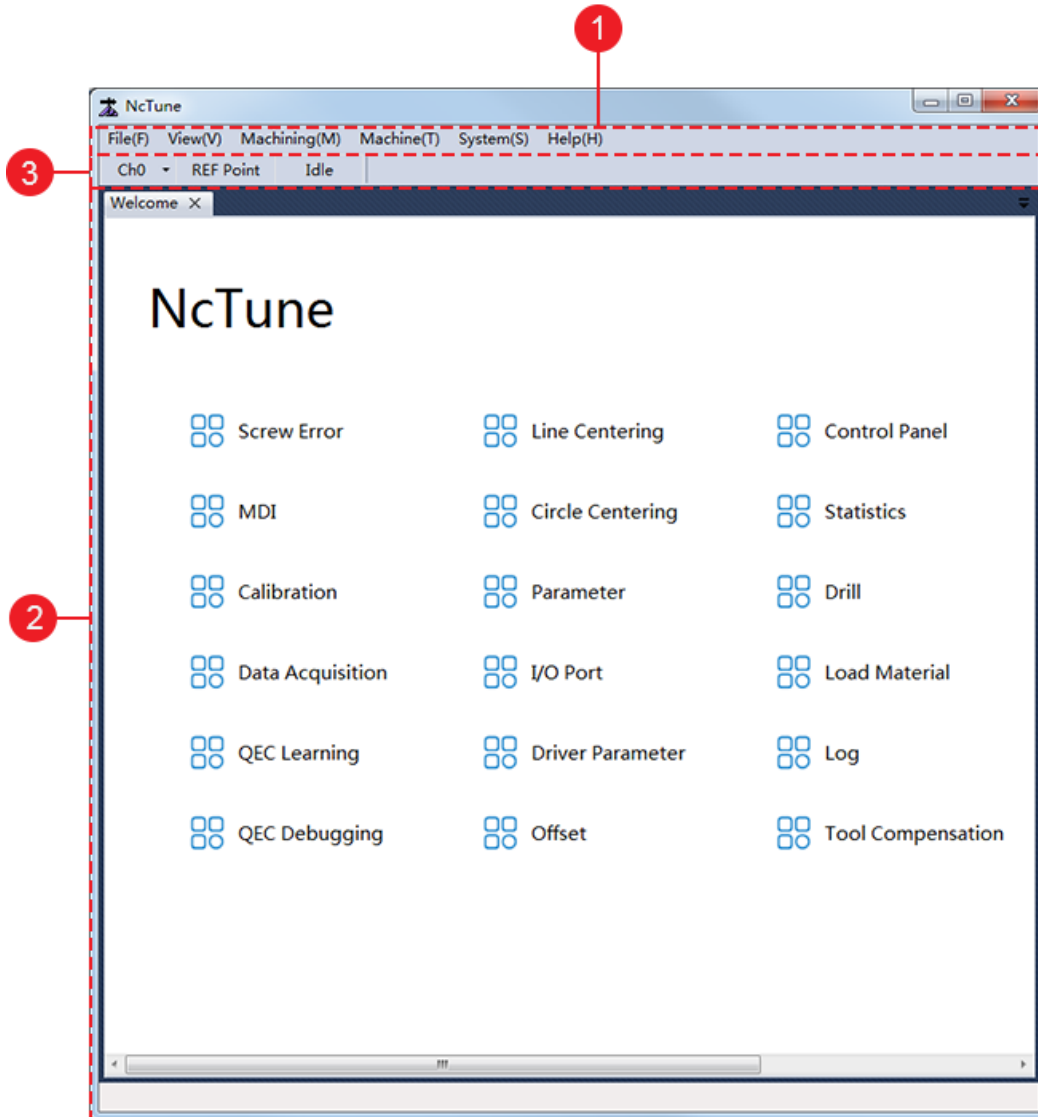
1. Menu bar
2. Welcome page

   In the welcome page, you can do the following:
     o Setting screw error compensation
     o Inputting MDI
     o Executing tool calibration
     o Executing data acquisition
     o Executing learning for the circularity test
     o Doing commissioning for the circularity test
     o Executing centering
     o Setting parameters
     o Setting port polarity
     o Setting drive parameters
     o Managing offsets
     o Checking control panel
     o Checking machining statistics

- o Setting gang drilling
- o Setting parameters for loading and unloading
- o Checking logs
- o Doing commissioning for the tool magazine
- o Setting tool compensation
- o Upgrading firmware
3. Status bar

1.3.3 Main Interface of NcStudio Software

The main interface of **NcStudio** software is as follows:
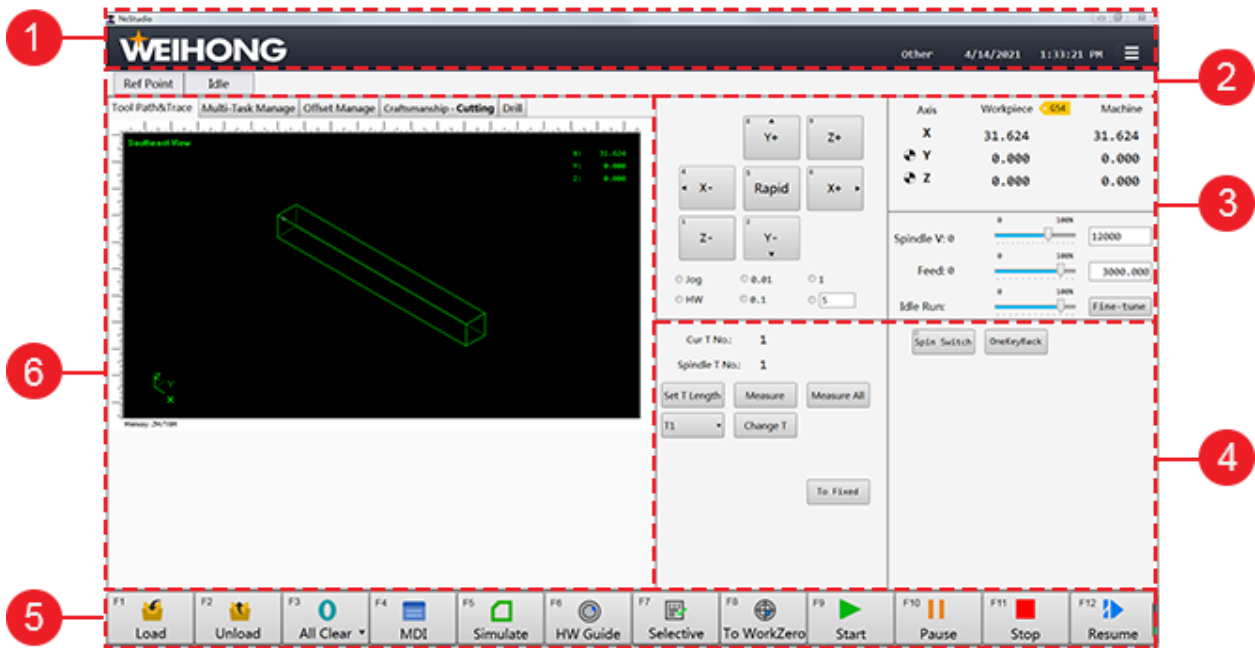
1. Title bar
2. Status bar
3. Machine control bar
4. Function control bar
5. Operation control bar
6. Function window

1.3.3.1 Title Bar

In this bar, you can check the following:

- Weihong logo
- Status of the control system
- Current date and time

- Hamburger menu

  In the hamburger menu, you can achieve the following:
  - o Executing returning to the machine origin or set datum.
  - o Setting tool parameter.
  - o Setting parameters for the tool magazine.
  - o Setting parameters for loading and unloading.
  - o Setting printer.
  - o Setting common parameters.
  - o Setting NcTune.
  - o Checking help information.

1.3.3.2   Status bar

In this bar, you can check the current operation mode, status, and tips or warnings.

You can double click the blank area to check different types of logs for troubleshooting, including information, warning, alarms. See Check System Logs for details.

1.3.3.3   Machine Control Bar

In this area, you can control the movement of the machine tool.

This bar is as follows:
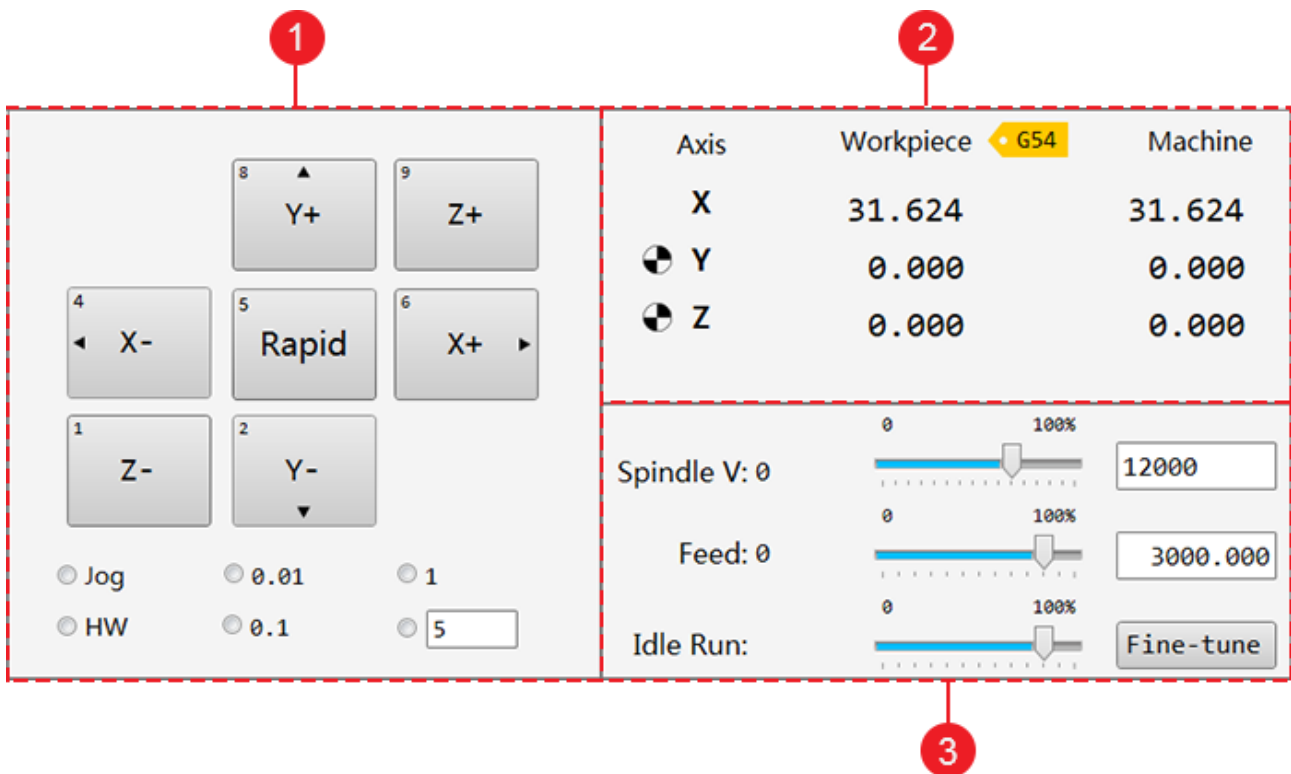
1. Manual control bar
   o Buttons for axis direction: move each axis towards positive or negative direction.
   o Mode buttons
      ▪ Jog
        Click an axis direction button. The machine tool keeps running at jog speed until you release the button.
        Click **Rapid** and an axis direction button. The machine tool keeps running at rapid jog speed until you release the button.
      ▪ HW: select the axis direction and handwheel override gear in the handwheel, and rotate the control disk. The machine tool is controlled by the handwheel and the selected axis moves at the selected handwheel speed.
      ▪ Step: click an axis direction button and release it. The machine tool moves the selected step size.
        You can select the fixed step size among 0.01mm, 0.1 mm, and 1 mm or a customized step size (default: 5mm).
        Note: Please do not set the step size to a too large value to avoid damage due to misoperation, and do not click an axis direction button too frequently because the system needs a certain time for response.

2. Coordinate display bar: check the workpiece coordinate, machine coordinate of each axis and the current workpiece coordinate system.
3. Speed control bar: adjust the spindle speed, feedrate and G00 speed, and check the current spindle speed and feedrate.
   The relationship between the override and speed is as follows:
   Current feedrate = Current feedrate override * Setting speed
   Current G00 speed = Current feedrate override * G00 setting speed

1.3.3.4   Function Control Bar

This bar is as follows:
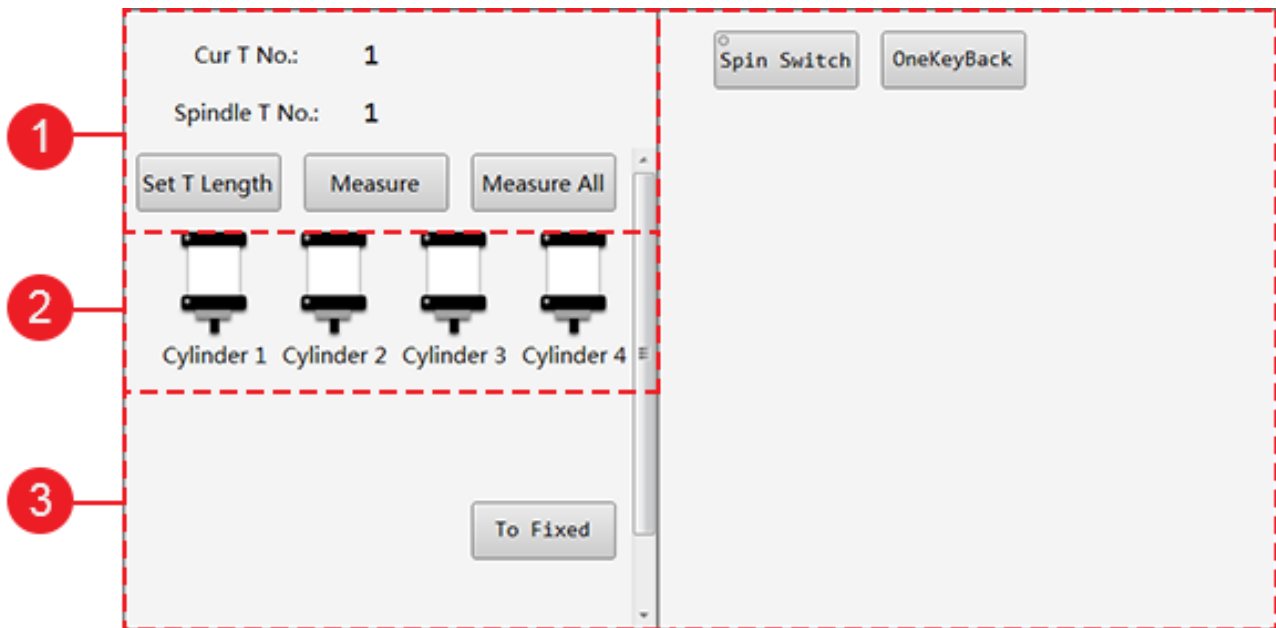
1. Operations for tool calibration
   o Check the current tool number and tool number on the spindle.
   o Do tool calibration:
     ▪ **Set T Length**: directly set tool offset of Z-axis.
     ▪ **Measure**: do tool calibration.
     ▪ **Measure All**: select several tools and do tool calibration.
2. Cylinder control bar: <u>turn on / off the cylinders</u>. At most four cylinders are supported.
   Note: If no cylinder is selected in the configuration bar in **NcConfig** software, this bar will be hidden.

3. Customized function bar: show ports and functions configured by **NcConfig** software. See <u>Set Machine Parts</u> for details.

1.3.3.5   Operation Control Bar

Operations in this bar are as follows:

- : load a program file.

- : unload a program file.

- : do clearing for a single axis or all axes.

- **MDI**: input MDI, including rapid movement and simple machining.

- **Simulate**: execute simulation. You can check the simulation track in **Tool Path&Trace** window to see if the toolpath exceeds the workbench stroke.

- **HW Guide**: control the machine tool with the handwheel.

- **Selective**: execute an optional skip of the program file, by customizing the start line and end line of the program block, so that you can freely select the machining range.

- **To WorkZero**: execute returning to the workpiece origin.

- **Start**: start machining.

- **Pause**: pause machining.

- **Stop**: stop machining.

- **Resume**: resume machining from the exact interrupted position when power interruption or an E-stop alarm occurs.

1.3.3.6   Function Windows

Function windows include the following:

- **Tool Path&Trace**: show the content of the loaded program file and the track during machining.
- **Multi-task Manage**: manage and edit task files for multi-task machining.
- **Offset Manage**: manage the workpiece offsets and public offsets.
- **Craftsmanship**: show the current technics, and set related technic parameters.
- **Gang Drilling**: add and orange the drilling in the drilling packages.

2   Settings in NcConfig Software

2.1   Overview

With **NcConfig** software, you can configure machine parts, set port mapping and port protection, and generate an installation package for multi-configuration, so as to improve the efficiency of project developing.

To open the main interface of **NcConfig** software, do the following:

1. Under the installation path *C:\Program Files\Weihong\NcStudio\NcConfig\Bin*, find and double click **NcConfig.exe**.
2. **Optional:** If **Projects** / **Configuration** page does not show in the left, in the menu bar, click **View → Projects** / **Configuration**.

2.2   Configure Devices

You can freely configure Lambda terminal board and the extended terminal board.

To configure devices, do the following in the main interface of **NcConfig** software:

1. In **Configuration** page in the left, click **NcStudio → Hardware Connection and Configuration**.
2. Click **Topology Configuration**, and right click to choose **Edit Topology**. **Topology Configuration** dialog box pops up:
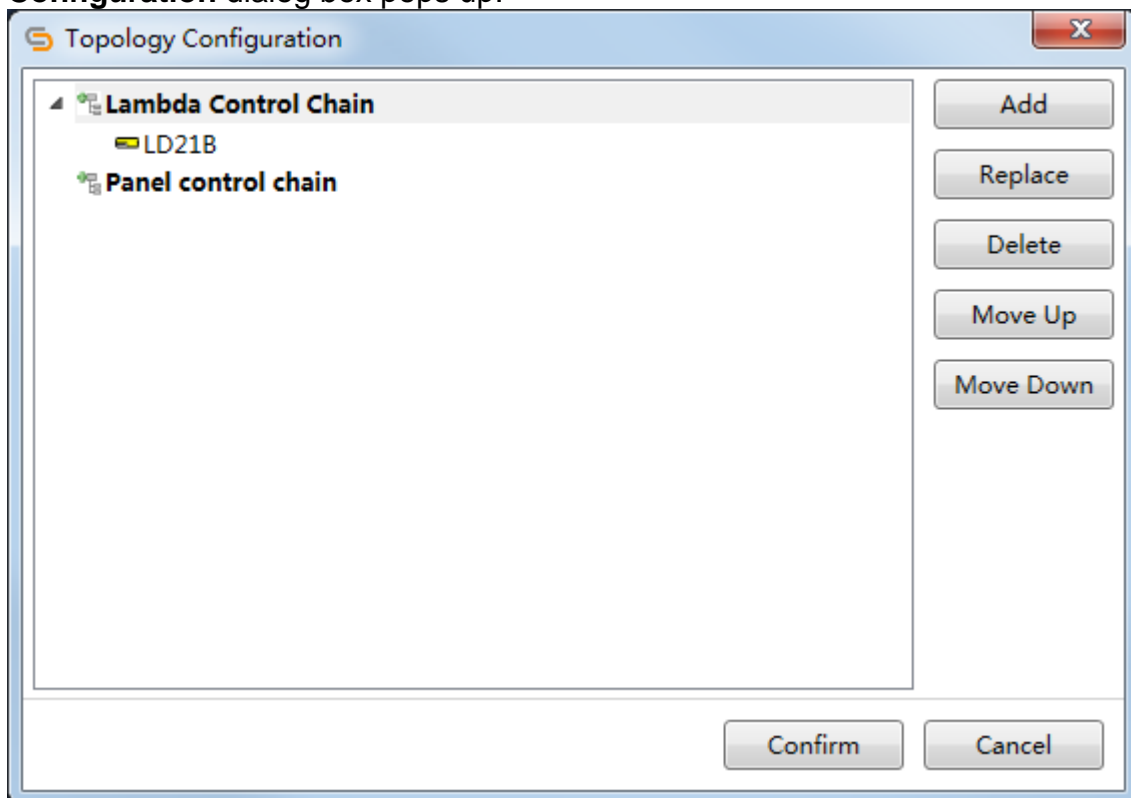
3. Configure terminal boards based on your needs:
   - To add the target terminal board, click **Add**.
   - To replace the target terminal board, click **Replace**.
   - To Delete the target terminal board, click **Delete**.
   - To change the order of terminal boards, click **Move Up** / **Move Down**.
4. Click **Confirm**.

2.3   Configure Machine Parts

It is used to set machine parts by **NcConfig** software, so that you can customize related functions in the customized bar of **NcStudio** software.

Machine parts include M codes, panels, double stations, tool magazine, saw, gang drill, material loading and unloading, handwheel and press wheel.

To configure machine parts, do the following:

1. In **Configuration** page in the left, click **NcStudio**, and double click **Machine Parts**. **Components** page shows:
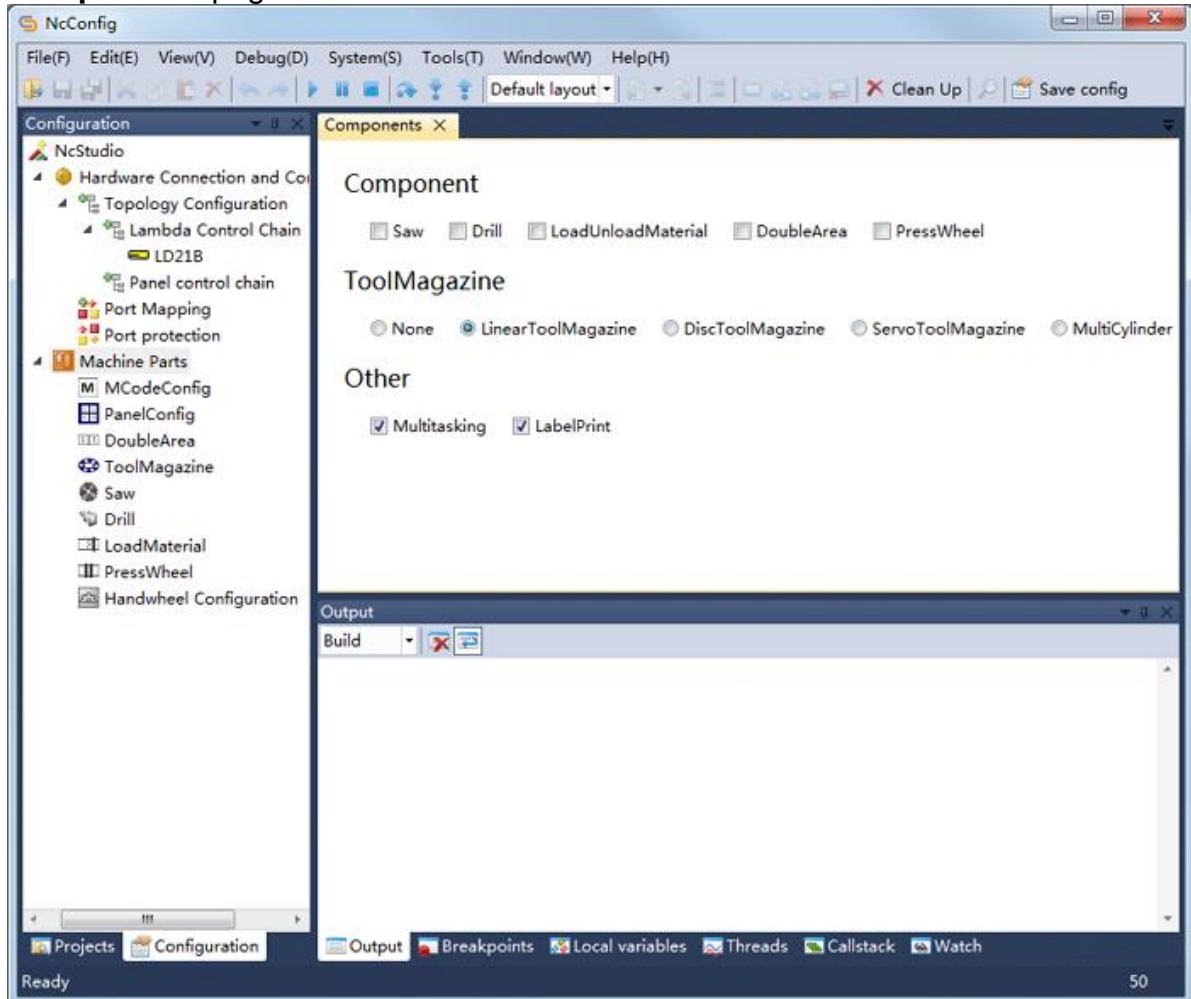
2. Check the target machine parts.
   After the target machine parts are checked, you can customize the following functions in the related software:
   NcTune software:
   - o Doing commissioning for the tool magazine
   - o Setting the gang drill
   - o Setting tool compensation
   - o Setting parameters for loading and unloading
   NcStudio software:
   - o Setting tool parameter
   - o Setting parameters for the tool magazine
   - o Setting parameters for loading and unloading
   - o Setting printer
3. Do the following based on your needs:
   - o Configuring M codes
   - o Configuring panels
   - o Configuring double stations

○ Configuring the tool magazine.
○ Configuring the saw
○ Configuring the gang drill
○ Configuring loading and unloading
○ Configuring handwheel
○ Configuring press wheel

2.3.1 Configure M Codes

It is used to configure M codes, so as to control the movement of the machine tool and the port status.

Note: Please configure M codes based on the selected machine parts, instead of configuring them freely.

M codes include the following:

- Function: these M codes are used to control the movement of the machine tool:

Code	Meaning	Code	Meaning
MCode 13	Drill on	MCode 15	Drill off
MCode 100	Load	MCode 200	Unload
MCode 300	Load & unload	MCode 753	Press wheel on
MCode 754	Press wheel off	MCode 23	Saw on
MCode 25	Saw off	MCode 26	Saw blade retracted
MCode 27	Saw blade swings in X direction	MCode 28	Saw blade swings in Y direction

- Valve: these M codes are used to control the port status:

Code	Meaning	Code	Meaning
MCode 701	Vacuum adsorption on	MCode 702	Vacuum adsorption off
MCode 703	Front pos cylinder on	MCode 704	Front pos cylinder off
MCode 707	Left pos cylinder on	MCode 708	Left pos cylinder off
MCode 709	Right pos cylinder on	MCode 710	Right pos cylinder off
MCode 711	Back push cylinder on	MCode 712	Back push cylinder off
MCode 713	Right push cylinder on	MCode 714	Right push cylinder off
MCode 715	Left push cylinder on	MCode 716	Left push cylinder off
MCode 719	Unload cylinder on	MCode 720	Unload cylinder off
MCode 721	Load cylinder on	MCode 722	Load cylinder off
MCode 723	Load blow on	MCode 724	Load blow off
MCode 725	Vacuum destroy on	MCode 726	Vacuum destroy off

To configure M codes, do the following:

1. In **Configuration** page in the left, double click **MCodeConfig**. **MCodeConfig** page shows.
2. To modify the target M code, select the target function, click on the target input box of **MCode** column, and input a value.

2.3.2 Configure Panels

It is used to configure panels, so that buttons of related functions will show in the customized bar of **NcStudio** software.

The configurable function buttons include: spindle switch, one key back, to fixed position, area 1/2 start, area 1/2 pos cylinder, area 1/2 vacuum adsorption, area 1/2 vac on, area 1 dust removal, area 1 pos cylinder, dust removal cylinder, drill start, drill cylinder, drill reset, load material, unload material, load & unload material, unload measure start, load cylinder, unload cylinder, vacuum adsorb, vacuum destroy, load blow, load adsorb, front/right/left pos cylinder, back/left/right push cylinder, press switch, saw T31/T32, saw ON, saw cylinder, manual unclamp tool, tool magazine cylinder, and dust cover.

Before configuring panels, make sure related machine parts have been checked in **Components** page.

To configure panels, do the following:

1. In **Configuration** page in the left, double click **PanelConfig**. **PanelConfig** page shows.
2. To configure a button for the target function in the customized bar of **NcStudio** software, click the target dropdown box, and select a target function.

2.3.3 Configure Double Stations

It is used to use two stations for machining by configuring related ports, in which the system can directly start machining on station 2 after machining on station 1 finishes, so as to save time of loading and unloading and improve machining efficiency.

To configure double stations, do the following:

1. In **Configuration** page in the left, double click **DoubleArea**. **DoubleArea** page shows.
2. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.

2.3.4 Configure the Tool Magazine

It is used to configure the target type of tool magazine and multi-cylinder.

The supported tool magazines include the following:

- Linear tool magazine
- Disk tool magazine
- Servo tool magazine

To configure the tool magazine, do the following:

1. In **Configuration** page in the left, double click **Machine Parts**. **Components** page shows.
2. In **ToolMagazine** area, select the target magazine type.

3. In **Configuration** page in the left, double click **ToolMagazine**. **ToolMagazine** page shows:

   o If the type of tool magazine is linear tool magazine, set parameter **Tool magazine capacity**.
     Tool magazine capacity: the total number of tools in tool magazine.

   o If the type of tool magazine is disk tool magazine, set the following parameters:
     ▪ **Tool magazine capacity**: the total number of tools in the tool magazine.
     ▪ **Count tool overtime(ms)**: the maximum time to wait for the tool counting signal.

   o If the type of tool magazine is servo tool magazine, set the following parameters:
     ▪ **Tool magazine capacity**: the total number of tools in the tool magazine.
     ▪ **Servo axis index**: the axis number of the servo axis.

   o If the type of tool magazine is multi-cylinder, select a cylinder type and set parameter **Cylinder count**:
     ▪ **Multi-cylinder single-fre**: one inverter controls the start and stop of several spindles.
     ▪ **Multi-cylinder multi-fre**: one Z-axis equips with two or four cylinders, and several inverters control the start and stop of several spindles.
     ▪ **Four-cylinder double-fre**: one Z-axis equips with four cylinders, and two inverters control the start and stop of several spindles.

4. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.

2.3.5 Configure the Saw

To configure the saw, do the following:

1. In **Configuration** page in the left, double click **Saw**. **Saw** page shows.
2. Set parameters **Saw axis index**.
3. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.

2.3.6 Configure the Gang Drill

To configure the gang drill, do the following:

1. In **Configuration** page in the left, double click **Drill**. **Drill** page shows.
2. Set parameter **Drill start number**.
3. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.

2.3.7 Configure Loading and Unloading

It is used to make sure that the physical address is the same with the logic address, so as to control the movement of the front positioning cylinder.

To configure loading and unloading, do the following:

1. In **Configuration** page in the left, double click **LoadMaterial**. **LoadMaterial** page shows.
2. Select a type for loading and unloading:
   o **Standard load & unload material**: load and unload a group of boards.
   o **Multi-plate load & unload material**: load and unload two or four groups of boards.

3. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.

Example

There is only one front positioning cylinder for three types of boards, which is connected to Y10 on Lambda terminal board. Thus, you need to set the logic address of front positioning cylinder for board 1, board 2 and board 3 to Y10.

2.3.8 Configure the Handwheel

After configuring the handwheel, you can control the movement of the machine tool by the handwheel or enable handwheel guide.

To configure the handwheel, do the following:

1. In **Configuration** page in the left, double click **Handwheel configuration**. **Handwheel configuration** page shows.
2. Change **Turn on** switch to **ON** status.

2.3.9 Configure the Press Wheel

After configuring the press wheel, the phenomenon that the board deviates during machining can be avoided.

To configure the press wheel, do the following:

1. In **Configuration** page in the left, double click **PressWheel**. **PressWheel** page shows.
2. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.

2.4 Set Port Mapping

It is used to make sure that the physical address is the same with the logic address.

To set port mapping, do the following:

1. In **Configuration** page in the left, double click **Port Mapping**. **Port Mapping** page shows:
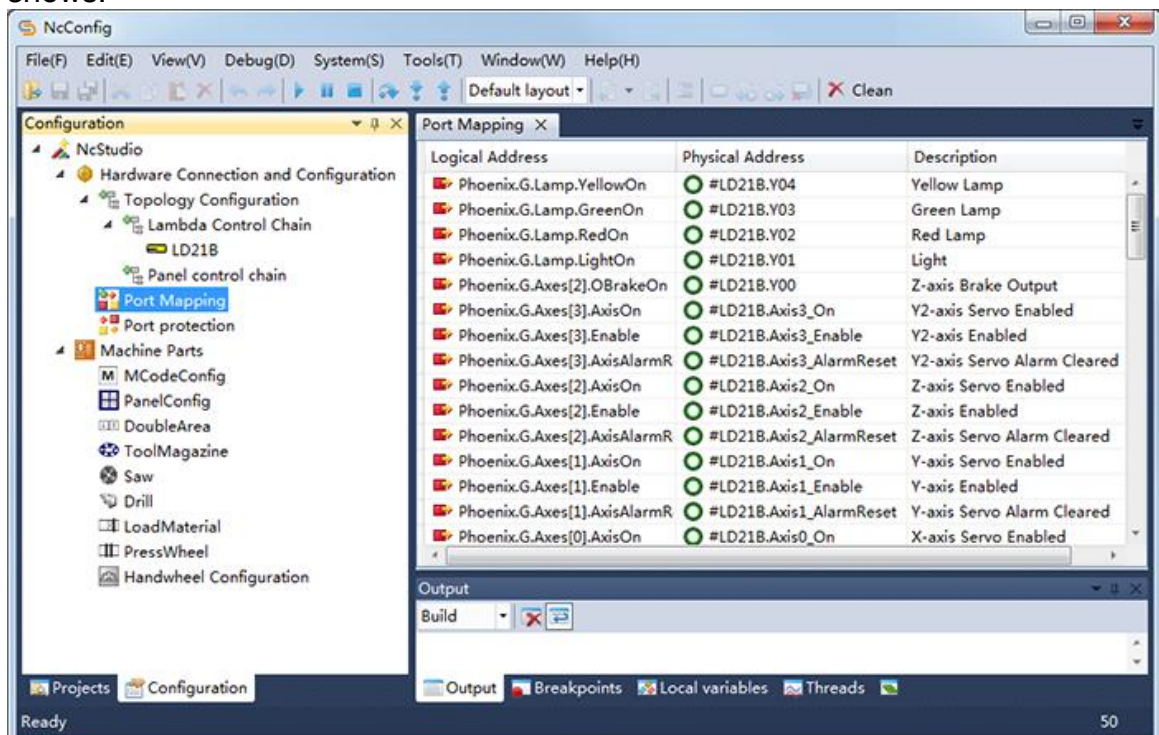
2. Double click the target port in **PhysicalAddress** column, and select the target Lambda terminal board and port.
3. To save the modification, do one of the following:

   o In the menu bar, click ![save icon].
   o Close the **Port Mapping** page, and click **Yes** in the popup.

   If the modification is not saved, sign * shows in the page name.

2.5 Set Port Protection

The type of port protection includes the following:

- **Open**: the protected port automatically has a signal when the software starts.
- **Close**: the protected port automatically closes its signal when the software closes.
- **Follow**: the status of the following port is the same with that of the protected port.
- **RFollow** the status of the following port is different from that of the protected port.

To set port protection, do the following:

1. In **Configuration** page in the left, click **NcStudio → Hardware Connection and Configuration**, and double click **Port protection**. **Port Protection** page shows.
2. Change the ON/OFF switch of the target port in **Enable** column to **ON** status.
3. Double click the target port of **Protection Type** column, and select a protection type:
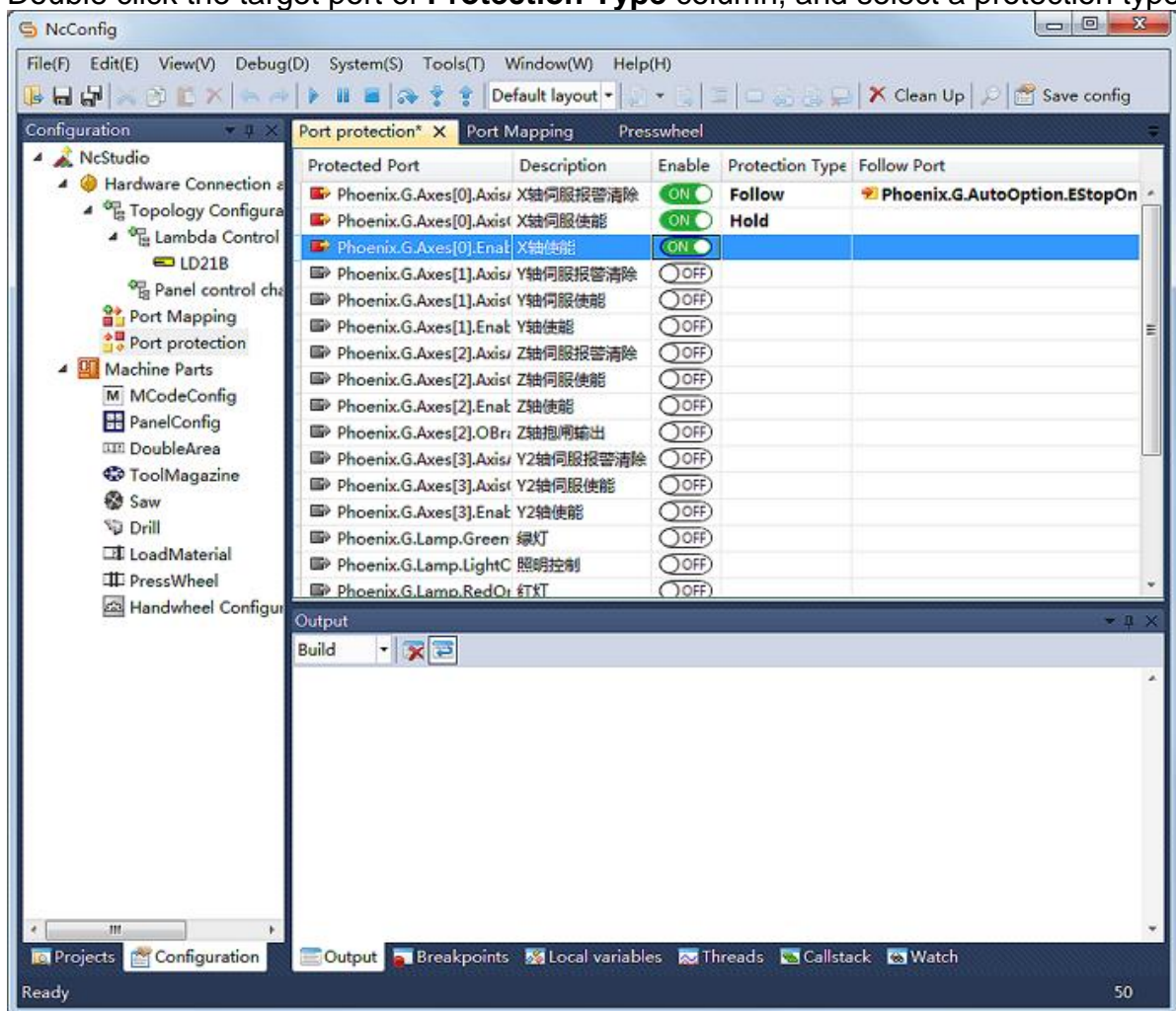
4. To save the modification, do one of the following:

    o In the menu bar, click 📁.

    o Close **Port Protection** page, and click **Yes** in the popup.

If the modification is not saved, sign * shows in the page name.

2.6 Generate an Installation Package for Multi-configuration

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

This operation will pack up all folders whose name begins with *Default_Config*.

To generate an installation package, do the following:

1. In the menu bar, click **System → Multi-configuration packaging**. **Packup Tool** dialog box pops up:

2. Modify the name and storage path of the installation package.

3. **Optional:** Click **Advanced**, and set the following advanced settings:

- o Language: at present, the system supports Chinese and English.
- o Parameter migration:
  - **Keep All Local Parameters**: use all parameters in the local.
  - **Keep Only Machine-Specific Parameters**: use parameters about machine characteristics in the local.
  - **Fresh Installation**: fresh Installation
- o Auto boot: whether to automatically start the software once the machine boots.
- o Others
  - **Support language selection**: whether to support switching the current language.
  - **Create a desktop shortcut**: whether to create a shortcut on the desktop.
  - **Start the software after installation**: whether to automatically start the software after installation.

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

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

3   Settings in NcTune Software

3.1   Overview

With **NcTune** software, you can do basic commissioning, applied commissioning, commissioning for the machine tool, and machining commissioning for WEIHONG **NcStudio CNC System for Cutting Machines**.

Each page in **NcTune** software can be dragged into an independent page for checking information on several pages at the same time. You can double click the title bar of the target page for restoring.

To enter the main interface of **NcTune** software, do one of the following:

- In the upper right corner of the main interface of **NcStudio** software, click the hamburger menu ▤ → **NcTune**.
- Close **NcStudio** software, find and double click **NcConfig.exe** in the installation path *C:\Program Files\Weihong\NcStudio\NcConfig\Bin*.

3.2   Basic Commissioning

3.2.1   Overview

Through this part, you can quickly know the commissioning process of WEIHONG **NcStudio CNC System for Cutting Machines**.

The basic commissioning process is as follows:
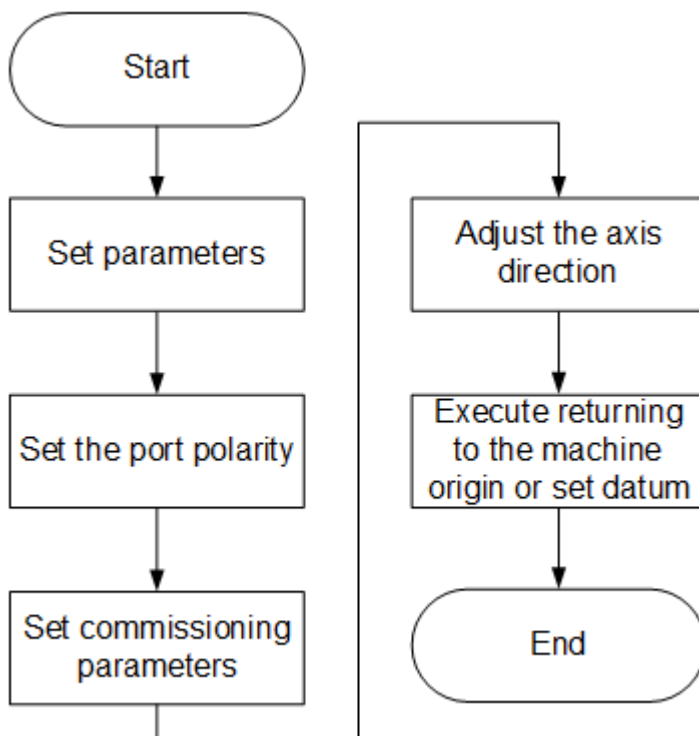
3.2.2   Set Parameters

It is used to set parameters according to the machining demands, to make sure the machining precision and improve machining efficiency.

To set parameters, do the following:

1. To open **Parameter** page, do one of the following:
   o In **Welcome** page, click **Parameter**.
   o In the menu bar, click **View → Parameter**.
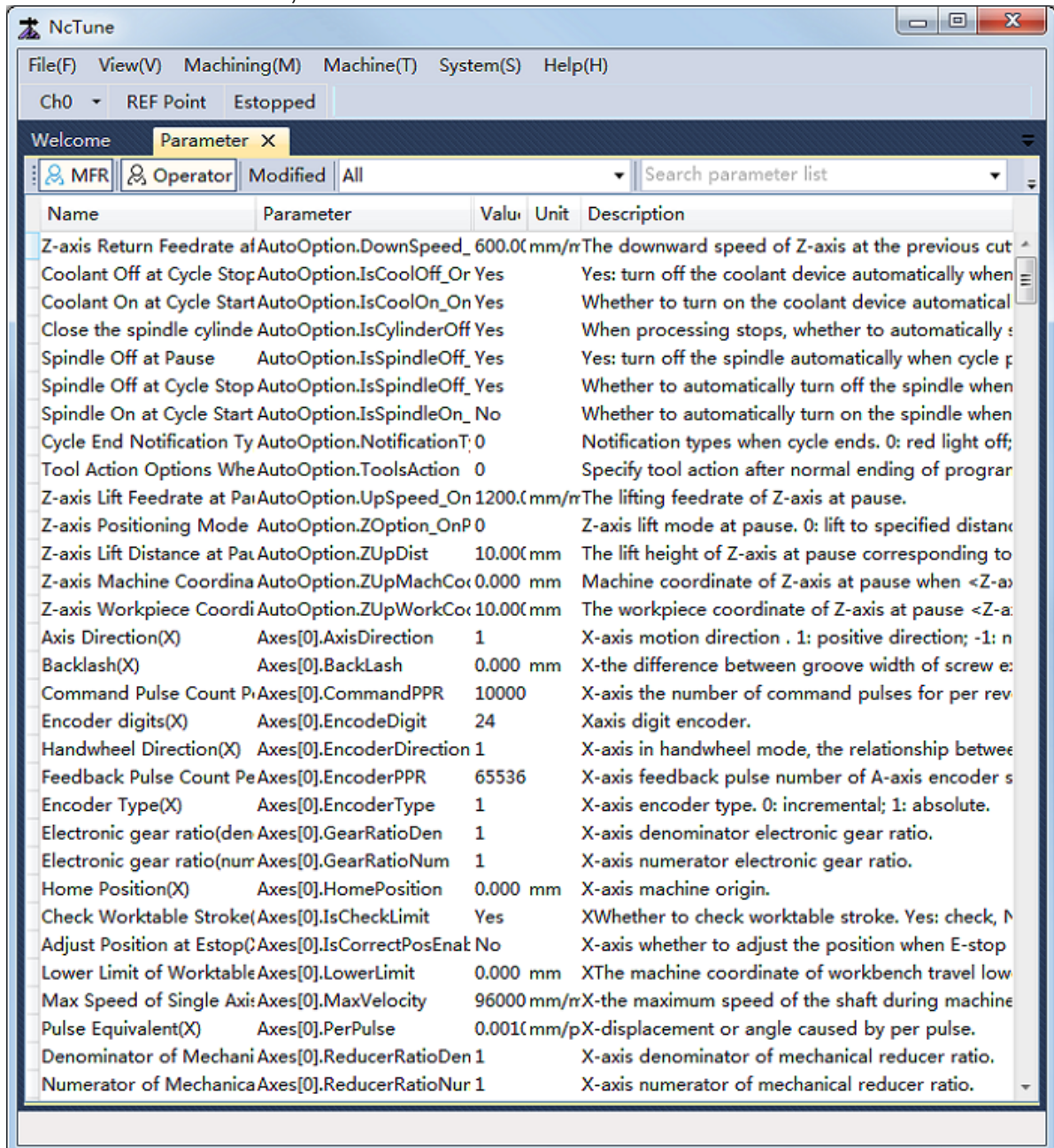
2. To modify the value of the target parameter, double click the target parameter in **Value** column, and input a value.
3. **Optional:** To set the target parameter as a frequently used parameter, select the target parameter and click **Set Common** in the upper right corner.
   After setting it as a frequently used parameter, to check common parameters, in the upper right corner of the main interface of **NcStudio** software, click the hamburger menu → **Common Parameters**.

3.2.3 Set the Port Polarity

It is used to check if the polarity of each port is correct, and modify the polarity if not.

The polarities of input/output ports in the software are specified in terms of the switch type:

- Polarities of normally open ports: NO.
- Polarities of normally closed ports: NC.

The status of each port includes the following:

- 🔴: the port has no signal.
- 🟢: the port has a signal.
- 🔲: the port is under test with no signal.
- 🔲: the port is under test with a signal.

To set the port polarity, do the following:

1. To open **Port** page, do one of the following:
   o In **Welcome** page, click **I/O Port**.
   o In the menu bar, click **View → I/O Port**.
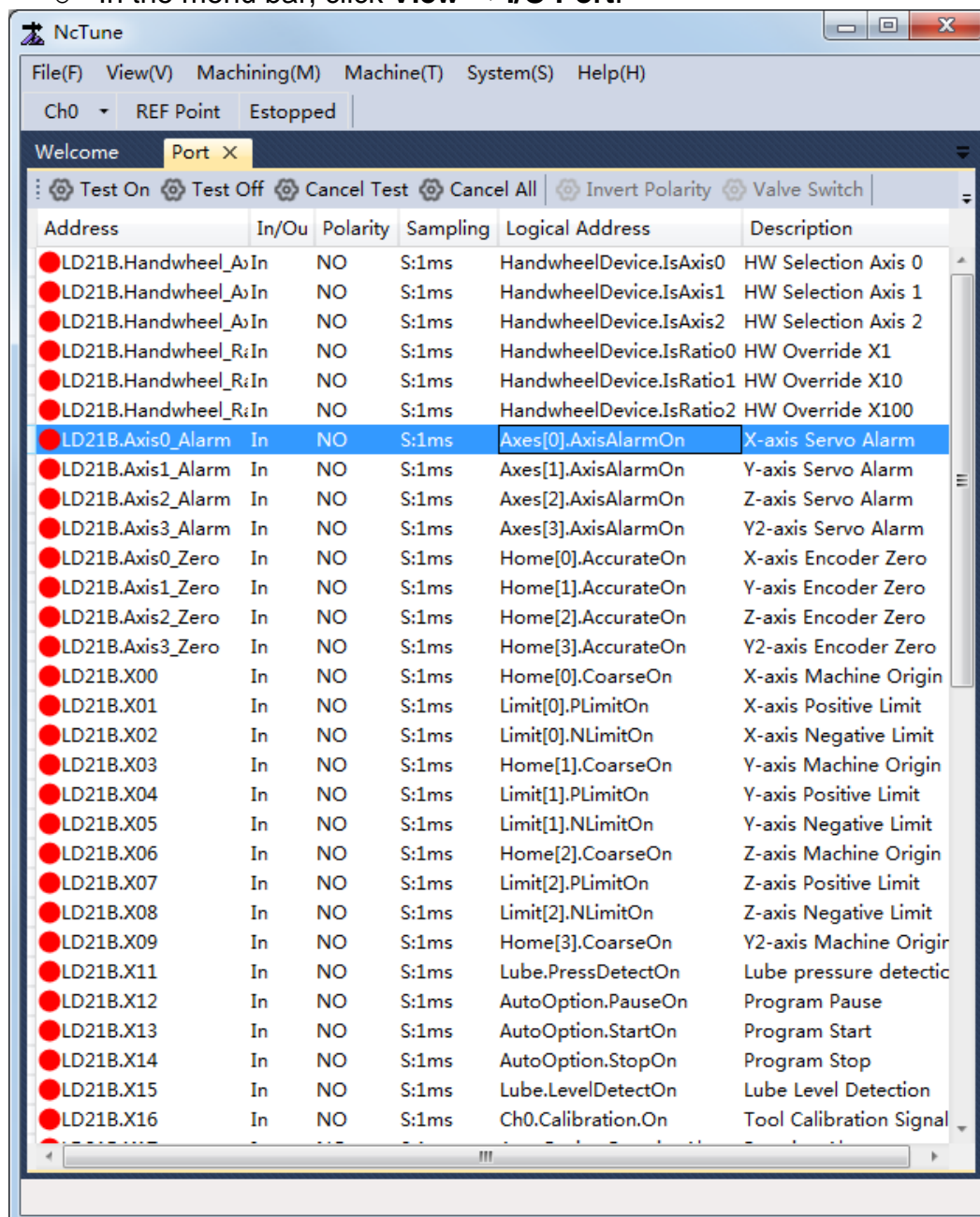
2. Select the target port, right click and select **Invert Polarity**.

In this interface, you can also do the following:

*   To simulate the hardware signal, press **Test On** to conduct a simulation test or **Test Off** to close a simulation test.
*   To cancel a simulation test for the selected port, click **Cancel Test**.
*   To cancel a simulation test for all ports, click **Cancel All**.
*   To open or close the target port without affecting its use in the actual use, click **Valve Switch**.
*   To set the sampling interval and enable filter time, click **Sampling**, slide bar **Sampling Interval** to set the sampling interval, and check **Filter**. The system rules out interference signals whose occurrence time is shorter than the set interval.
*   To show all ports, click **Show All**.

3.2.4 Set Commissioning Parameters

It is used to set commissioning parameters to make the system, the servo drive and the machine tool can match with each other.

This operation differs in the type of control system:

*   Set commissioning parameters for the non-bus type control system: including setting parameters **Pulse Equivalent**, **Subdivision**, **Pitch**, **Mechanical Deceleration Ratio**, **Electronic Gear Ratio**.
*   Set commissioning parameters for the bus type control system: including setting parameters **Electronic Gear Ratio**, **Encoder Resolution**, **Pitch**, and **Mechanical Deceleration Ratio**.

3.2.4.1 Set Commissioning Parameters for the Non-bus Type Control System

To set commissioning parameters for the non-bus type control system, do one of the following based on the motor type:

*   If a stepping motor is used, do the following:
    a. Set system parameters **Pulse Equivalent**, **Pitch**, and **Mechanical Deceleration Ratio**.
    b. Set drive parameter **Subdivision**.
    c. Verify if the parameter settings are correct according to the following formula:
        Pulse Equivalent = Pitch / (360 / Stepping Angle * Subdivision * Mechanical Deceleration Ratio)
        If the parameter settings are incorrect, repeat step 1~3.
*   If a servo motor is used, do the following:
    a. Set system parameters **Pulse Equivalent**, **Encoder Revolution**, **Pitch**, **Mechanical Deceleration Ratio** and **Electronic Gear Ratio**.
    b. Verify if the parameter settings are correct according to the following formula:
        Electronic Gear Ratio = (Encoder Revolution * Pulse Equivalent) / Pitch * Mechanical Deceleration Ratio
        If the parameter settings are incorrect, repeat step 1~2.

Related Parameters

*   **Pitch**: the axial distance between the corresponding points of two adjacent teeth on the threads.
*   **Stepping angle**: the angle at which the motor rotates every time the control system sends a stepping pulse signal.
*   **Subdivision**: the percentage of the actual step angle to the fixed step angle.

- **Mechanical Deceleration Ratio**: the ratio of reducer input speed to output speed. It equals to the ratio of the teeth number of driven wheel to that of driving wheel. When applied in CNC machines, it specifies the ratio of motor speed to screw speed.
- **Encoder Resolution**: the pulse needed when the servo motor axis rotates one circle.
- **Pulse Equivalent**: 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. Smaller value gets higher machining precision and surface quality; larger value gets faster maximum feedrate.
  It can be calculated in terms of the actual electronic gear ratio of the servo drive.
- **Electronic Gear Ratio**: the ratio that the servo enlarges or shrinks the received pulse frequency. If its value is greater than 1, the servo enlarges the received pulse frequency; if it is less than 1, the servo shrinks the received pulse frequency.
  It can be calculated in term of the value of system parameter **Pulse Equivalent**.

3.2.4.2 Set Commissioning Parameters for the Bus Type Control System

When the control system is the bus type control system, the motor type is servo motor.

To set commissioning parameters for the bus type control system, do the following:

1. Set system parameters **Encoder Revolution**, **Pitch** and **Mechanical Deceleration Ratio**.
2. Set drive parameter **Electronic Gear Ratio**:
3. Verify if the parameter settings are correct according to the following formula:
   Electronic Gear Ratio = (Encoder Revolution * Pulse Equivalent) / Pitch * Mechanical Deceleration Ratio
   If the parameter settings are incorrect, repeat step 1~3.

Related Parameters

- **Pitch**: the axial distance between the corresponding points of two adjacent teeth on the threads.
- **Stepping angle**: the angle at which the motor rotates every time the control system sends a stepping pulse signal.
- **Subdivision**: the percentage of the actual step angle to the fixed step angle.
- **Mechanical Deceleration Ratio**: the ratio of reducer input speed to output speed. It equals to the ratio of the teeth number of driven wheel to that of driving wheel. When applied in CNC machines, it specifies the ratio of motor speed to screw speed.
- **Electronic Gear Ratio**: the ratio that the servo enlarges or shrinks the received pulse frequency. If its value is greater than 1, the servo enlarges the received pulse frequency; if it is less than 1, the servo shrinks the received pulse frequency. It should be set the same with the electronic gear ratio set in the servo drive. It is set to 1:1 by default.
- **Encoder Resolution**: the pulse needed when the servo motor axis rotates one circle.
- **Encoder Type**: it should be set the same with the actual encoder type.
  - 0: incremental encoder
  - 1: absolute encoder
- **Encode Digit**: it should be set the same with the actual condition.

3.2.5 Adjust the Axis Direction

It is used to check if the positive direction of each axis is the same with the direction specified 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. Judge the positive direction of X-axis according to **Right Hand Rule**.
2. In the control panel, click **X+** to move X-axis and observe if the moving direction of X-axis is the same with the judged direction.
   If not, modify the value of parameter **Axis Direction(X)** to the opposite value.

3.2.6 Execute Returning to the Machine Origin or Set Datum

Before operating the machine tool, please execute returning to the machine origin or set datum to adjust coordinates.

This operation differs in the encoder type:

- Incremental encoder: execute returning to the machine origin
- Absolute encoder: set datum

The encoder type is determined by parameter **Encoder Type**.

3.2.6.1   Execute Returning to the Machine Origin

The origin of the machine coordinate system (inherent coordinate system of a machine tool), also called mechanical origin or home, is a fixed point assigned by designing, manufacturing and debugging before the machine tool leaves the factory.

This operation is used to synchronize the machine coordinate system of the system and the machine tool.

Before executing returning to the machine origin, make sure all servo alarms have been removed.

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

1. In the menu bar, click **Machine** → **Homing**. **Homing** dialog box pops up:
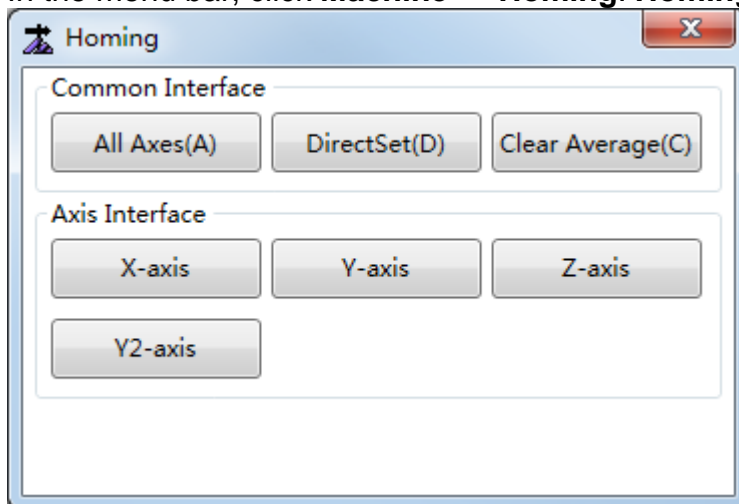
2. **Optional:** If the switch position of the machine origin changes, click **Clear Average**.

3. Do one of the following:
    o To return all axes to the machine origin, click **All Axis**. Z-axis returns to the machine origin firstly, then X-axis, and Y-axis return in sequence.
    o To return a single axis to the machine origin, click the button for a single axis, the target axis returns to the machine origin.
    o If you make sure that the current position is the same with the machine coordinate, the machine tool has never been turned off and E-stop has never occurred, to set the current position as the machine origin, click **Direct Set**.

After executing returning to the machine origin, sign ⊕ appears in front of each axis in the axis coordinate display bar.

3.2.6.2　　　*Set Datum*

With an absolute encoder, you can set the current position as the machine origin through setting datum when you use the system at the first time. Compared to returning to the machine origin with an incremental encoder, it owns the following advantages: No need to set returning orders for all axes.

No need to set datum again after restarting the system, and recovering from power interruption and E-stop because the system will automatically read the datum information.

To set datum, do the following:

1. In the menu bar, click **Machine → Datum Setting**.
2. Do one of the following:
    o To set datum for all axes, click **All Axis Datum Setting**.
    o To set datum for a single axis, click **Datum Setting(X)** / **Datum Setting(Y)** / **Datum Setting(Z)**.

After setting datum, sign ⊕ appears in front of each axis in the axis coordinate display bar.

3.3　Applied Commissioning

3.3.1　Set Drive Parameters

It is used to set basic drive parameters to drive the movement of the machine tool during commissioning.

To set drive parameters, do the following:

1. To open **Driver Parameter** page, do one of the following:
    o In **Welcome** page, click **Driver Parameter**.
    o In the menu bar, click **View → Driver Parameter**.
2. To get the latest drive parameters, click **Refresh**.
3. To check the drive parameters of the target axis, click the dropdown box of the related box, and select the target axis.
4. To modify the value of the target drive parameter, double click the target drive parameter in **Value** column, and input a value.
5. **Optional:** To export drive parameters for later use, click **Export**.

3.3.2　Manage Offsets

Offsets include the following:

- **Workpiece offset**: the distance between the workpiece origin and the machine origin.
- **Public offset**: the distance between X-axis / Y-axis / Z-axis and the machine origin for all workpiece coordinate systems.
- **Tool offset**: the distance between the tool and standard tool.

The relationship between workpiece coordinate and offsets is as follows:

Workpiece coordinate = Machine coordinate - Workpiece offset - Public offset - Tool offset
To manage offsets, do the following:

1. To open **Offset** page, do one of the following:
   - In **Welcome** page, click **Offset**.
   - In the menu bar, click **View → Offset**.
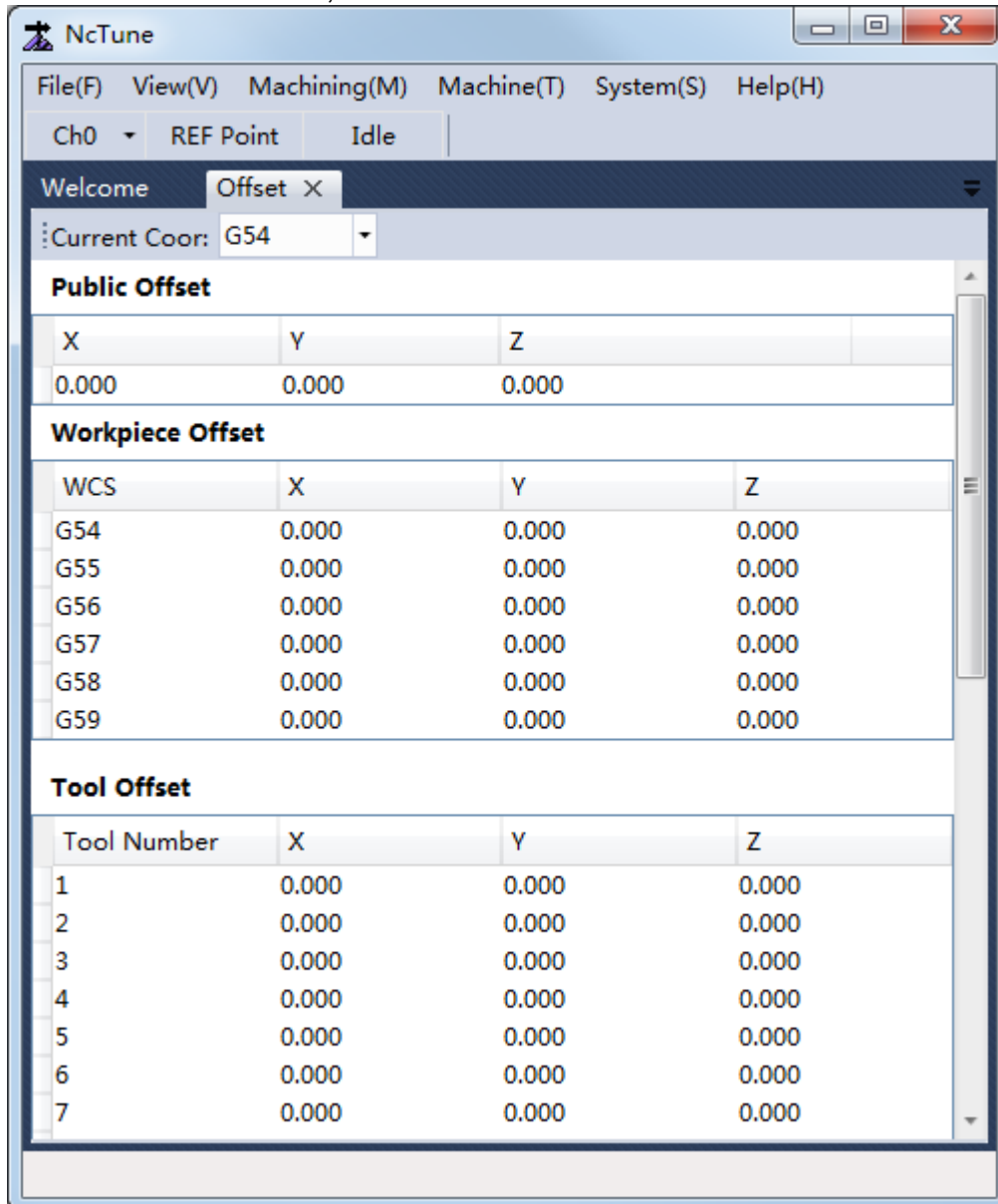
2. In the dropdown box of **Current Coor**, select the target workpiece coordinate system.
3. To set the public offset, in **Public Offset** area, click the input box of the target axis, and input a value.
4. To set the workpiece offset, in **Workpiece Offset**, click the input box of the target axis, and input a value.
5. To set the tool offset, in **Tool Offset**, click the input box of the target axis, and input a value.

3.3.3 Check the Control Panel

With the control panel, you can check the coordinates of each axis, control the status and movement of the machine tool, adjust spindle / feedrate / G00 speed, check the current spindle and feed speed, use function buttons, and view the content of the load program file.

Function buttons include **Load**, **Unload**, **To Origin**, **To Fixed**, **Simu**, **Start**, **Pause**, **Stop**, **Resume**, **Selective**, and **HWGuide**.

To open **Control Panel** page, do one of the following:

- In **Welcome** page, click **Control Panel**.
- In the menu bar, click **View → Control Panel**.
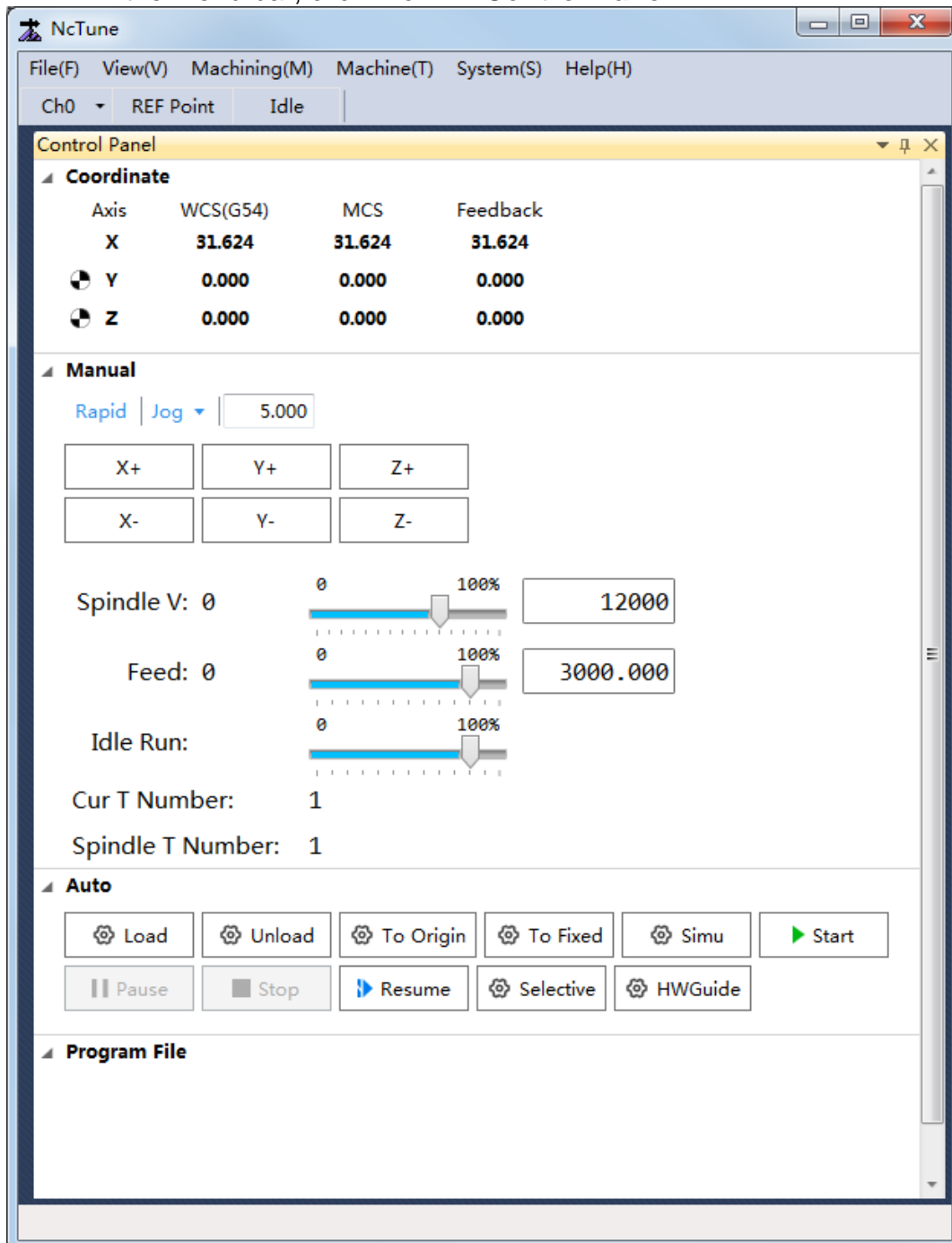
3.3.4 Check Machining Statistics

It is used to check the whole machining results.

The machining statistics include the following:

- Workpiece count
- Current line
- Machining length
- Upper limit / lower limit of machining range
- Difference of machining range
- Cut time / elapsed time / start time / end time
- G00~G03 Tool travel length
- In machining
- Upper limit / lower limit of movement range
- Difference of movement range
- Name
- Finished
- Process time
- Remaining time

To open **Statistics** page, do one of the following:

- In **Welcome** page, click **Statistics**.
- In the menu bar, click **View → Statistics**.

NcTune			
File(F) View(V) Machining(M) Machine(T) System(S) Help(H)			
Ch0 ▼ REF Point Idle			

Statistics	▼ ⊓ ×
▲ Statistics	
Enable Statistics	False
Part Count	0
Current Line	0
Machining Length	0
Machining Range Upper Limit	System.Double[]; Count: 6
Machining Range Lower Limit	System.Double[]; Count: 6
Machining Range Difference	System.Double[]; Count: 6
Cut Time	00:00:00
Elapsed Time	00:00:00
Start Time	1/1/0001 12:00:00 AM
End Time	1/1/0001 12:00:00 AM
G00 Tool Travel Length	0
G01 Tool Travel Length	0
G02 Tool Travel Length	0
G03 Tool Travel Length	0
In Machining	False
Movement Range Upper Limit	System.Double[]; Count: 6
Movement Range Lower Limit	System.Double[]; Count: 6
Movement Range Difference	System.Double[]; Count: 6
Name	AutoMain
Finished	0
Process Time	00:00:00
Remaining Time	00:00:00

3.3.5 Set the Gang Drill

It is used to add and arrange drills and set related parameters.

Before setting the gang drill, make sure gang drill has been enabled in **NcConfig** software.

Operations about drill include the following:

- **Select**: select the target drill.

- **Spindle**: add a spindle.

- **Vertical drill**: add a vertical drill to cut vertical holes.

- **Horizontal drill (up and down)**: cut side openings in front of or behind the actual position.

- **Horizontal drill (right and left)**: cut side openings in the right/left of the actual position.

To set the gang drill, do the following:

1. To open **Drill** page, do one of the following:
   o In **Welcome** page, click **Drill**.
   o In the menu bar, click **View → Drill**.
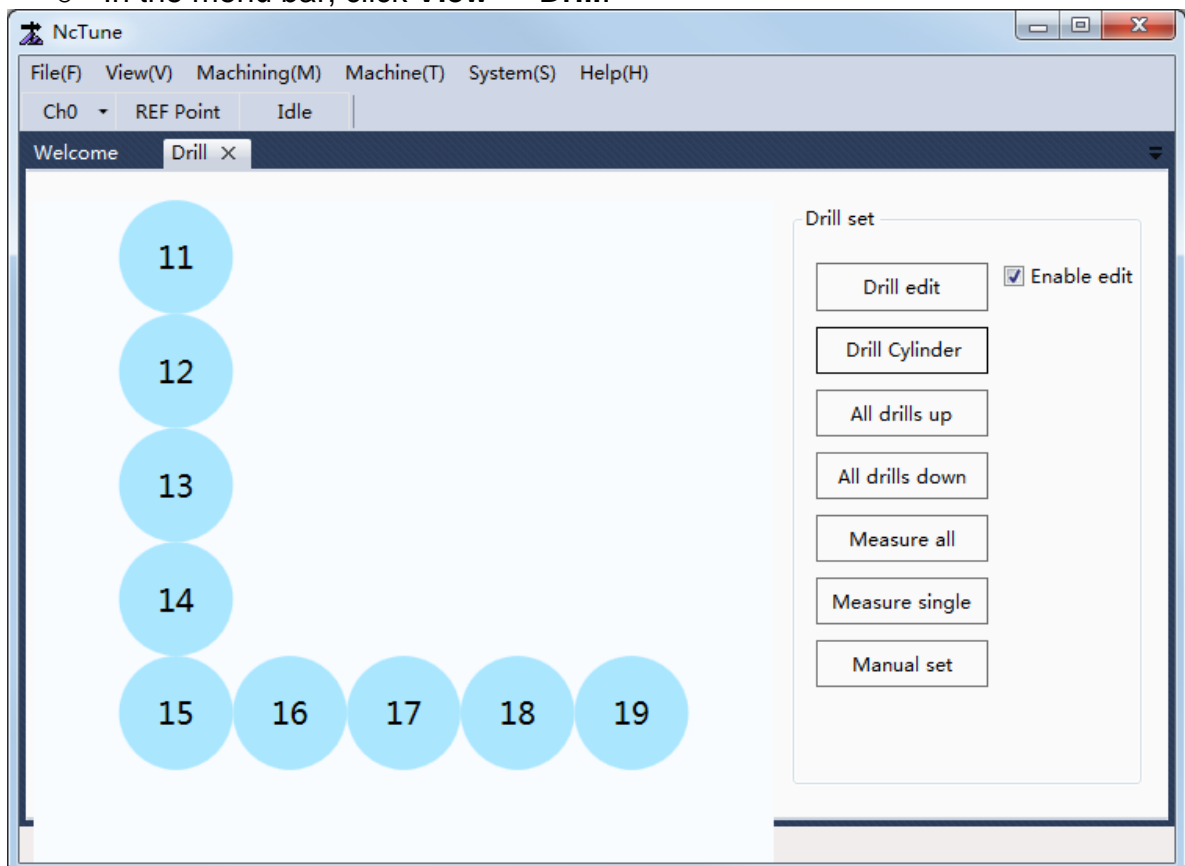
2. In the upper right corner, check **Enable edit**.

3. Click **Drill edit**. **Drill edit** dialog box pops up:

Index	Type	X	Y	Z	X Length	Y Length	X External length	Y External length
11	Vertical	0.000	192.000	0.000	16.000	16.000	0.000	0.000
12	Vertical	0.000	160.000	0.000	16.000	16.000	0.000	0.000
13	Vertical	0.000	128.000	0.000	16.000	16.000	0.000	0.000
14	Vertical	0.000	96.000	0.000	16.000	16.000	0.000	0.000
15	Vertical	0.000	64.000	0.000	16.000	16.000	0.000	0.000
16	Vertical	32.000	64.000	0.000	16.000	16.000	0.000	0.000
17	Vertical	64.000	64.000	0.000	16.000	16.000	0.000	0.000
18	Vertical	96.000	64.000	0.000	16.000	16.000	0.000	0.000
19	Vertical	128.000	64.000	0.000	16.000	16.000	0.000	0.000

4. In the upper area, select a drill type and add a drill.
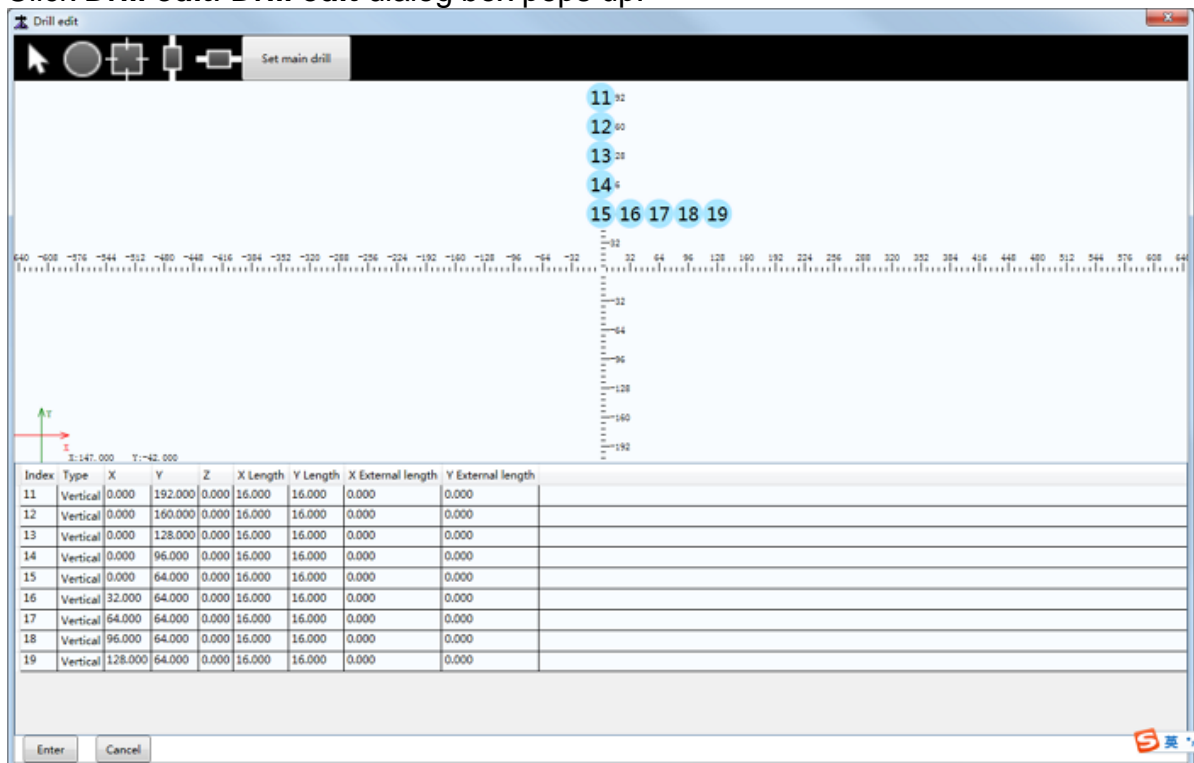5. **Optional:** To move the target drill, press the mouse wheel and drag the target drill to the target position and release the mouse wheel.
6. Set the following parameters for the target drill:
   o **X**, **Y** and **Z**: The offset of drill or spindle in X-axis / Y-axis / Z-axis direction.
   o **X / Y length**: The size of each drill, except horizontal drills.
   o **X / Y extended length**: The size of each horizontal drill.
   o **Port**: The logic address of the drilling cylinder.
7. In **Drill** page, click **Drill Cylinder**.
8. **Optional:** To close all drills, click **All drills up**.
   To open all drills, click **All drills down**.
9. **Optional:** To measure tool offsets of all drills in Z-axis direction, click **Measure all**.
   To measure the tool offset of the target drill, click **Measure single**.

3.3.6 Check Logs

It is used to check different types of logs that record important operations and system events after this startup and historical logs for troubleshooting.

To check logs, do the following:

1. To open **Log** page, do one of the following:
   - In **Welcome** page, click **Log**.
   - In the menu bar, click **View → Log**.
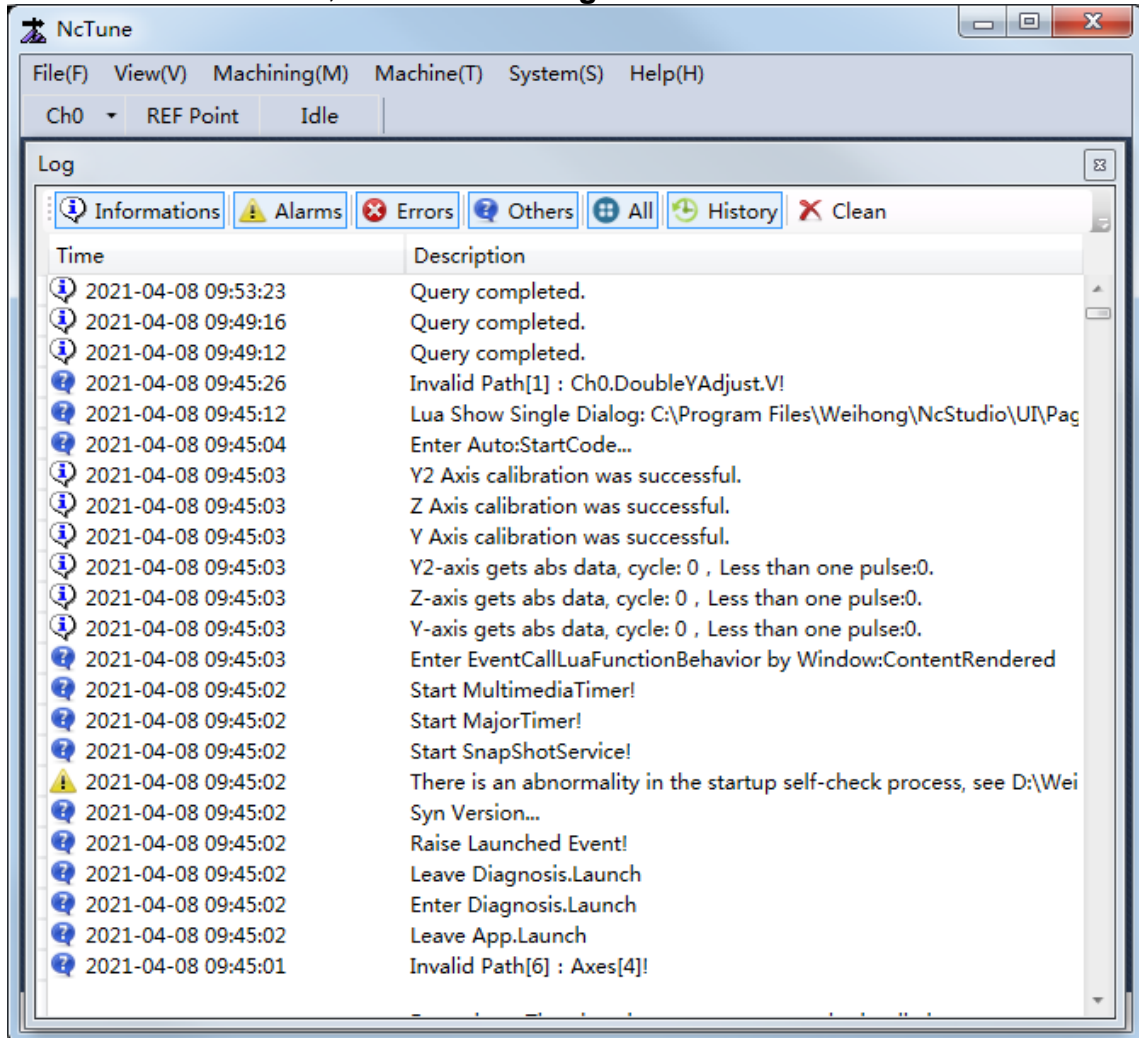
2. Check different types of logs:
   - Click **Information**. Logs with icon 🛈 about running status show.
   - Click **Alarms**. Warnings with icon ⚠ show.
   - Click **Errors**. Errors with icon ❌ show.
   - Click **Others**. Logs with 🛈 about operation steps and internal codes show.
   - Click **All**. The corresponding logs that generate after this startup show.
   - Click **History**. Logs since the installation of the software show.
3. **Optional:** To clear logs, click **Clear**.
   Note: Please regularly clear logs. Otherwise, too many log files will slow down the system.

3.3.7 Do Commissioning for the Tool Magazine

Before doing commissioning for the tool magazine, make sure multi-cylinder configuration is not enabled.

To do commissioning for the tool magazine, do the following:

1. To open **Magazine Setting**, do one of the following:
   - In **Welcome** page, click **Magazine Setting**.
   - In the menu bar, click **View** → **Magazine Setting.
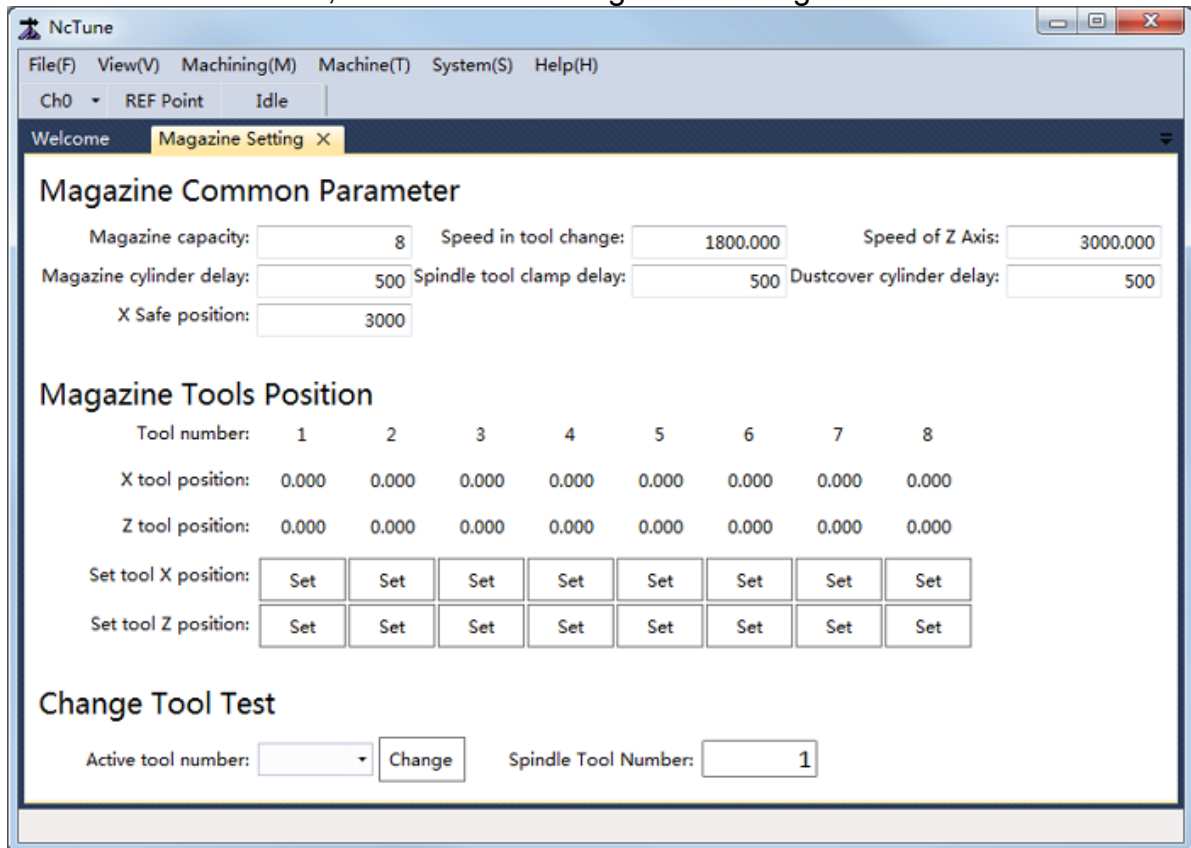
2. In **Magazine Common Parameter** area, set the following parameters:
   - **Magazine capacity**: the total number of tools in the tool magazine.
   - **Speed in tool change**: The spindle speed during tool change.
   - **Speed of Z-axis**: the speed for Z-axis moving to the upper / lower position during tool change.
   - **Magazine cylinder delay**: the delay time to turn on the magazine cylinder.
   - **Spindle tool clamp delay**: the delay time to clamp / unclamp tool.
   - **Dustcover cylinder delay**: the delay time for the dustcover up and down.
   - **X safe position**: it is used to set the X-axis protection position by judging if the current X-axis position is less than the X-axis position of the tool magazine when the tool magazine is pushed out.
3. To set the current position of tool magazine, in **Magazine Tools Position** area, click **Set**.
4. **Optional:** To replace the current tool, in **Change Tool Test** area, click the dropdown box of **Active tool number**, select the target tool number, and click **Change**.

3.3.8 Set Tool Compensation

Tool compensation includes tool length compensation and tool radius compensation.

In CNC machining, the CNC system actually controls the tool center or the related point of the tool rest whose motion track is controlled directly to realize profile processing for the parts. The cutting part actually involved is the tool nose or the cutting edge which has dimensional variation with the tool center or the related point of the tool rest, so the CNC

system has to compute the corresponding coordinates of the tool center or the related point of the tool rest according to the actual coordinate position of the tool rest or the cutting edge (namely the actual coordinate position of the part profile), which is called tool compensation.

Before setting tool compensation, set the following parameters to **Yes**:

- **Enable Tool Length Compensation**
- **Enable Tool Radius Compensation**

To set tool compensation, do the following:

1. To open **Tool Compensation** page, do one of the following:
   - In **Welcome** page, click **Tool Compensation**.
   - In the menu bar, click **View → Tool Compensation**.
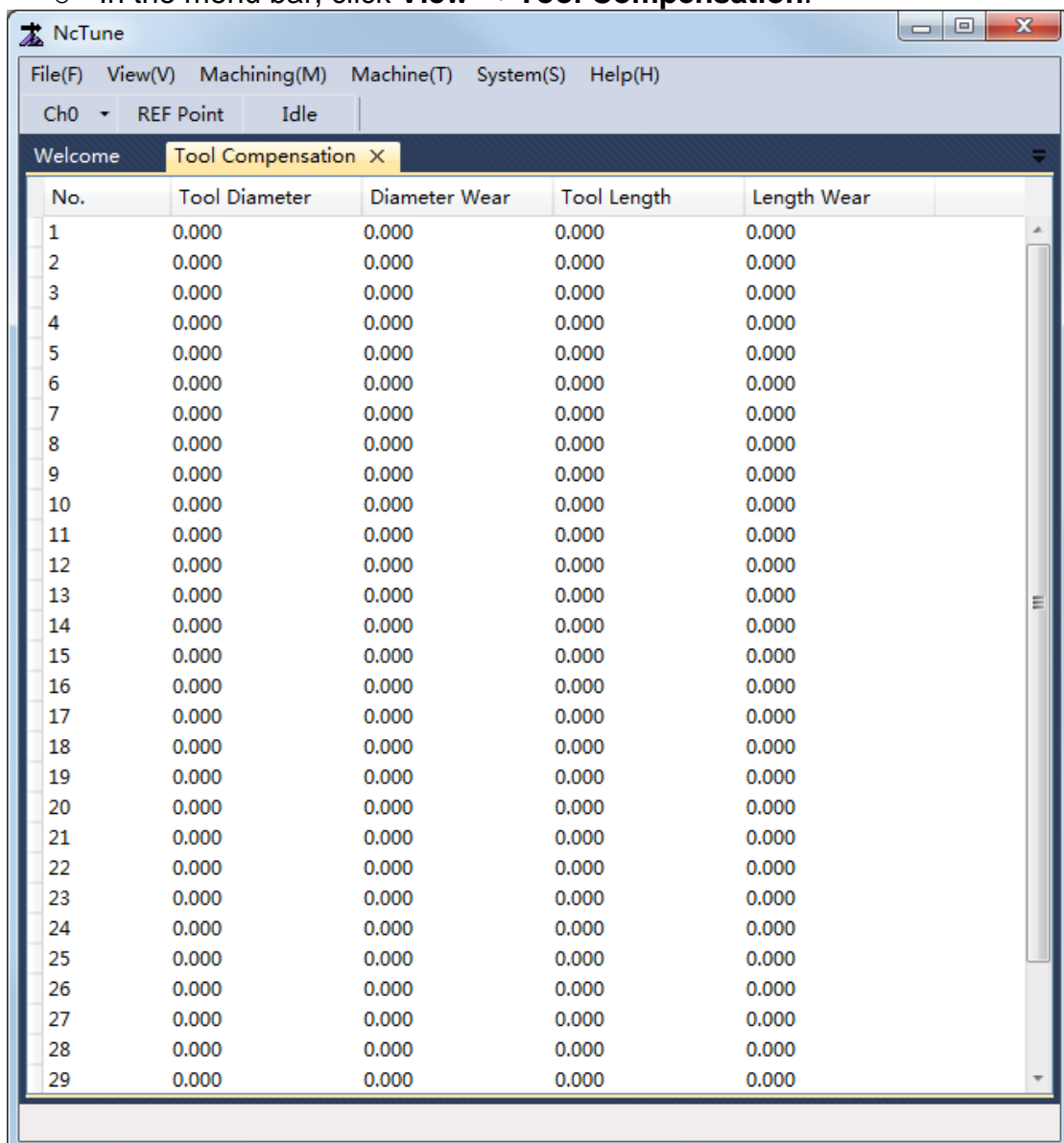
2. Set the following parameters for the target tool:
   - **Tool Diameter**: the diameter of the current tool.
   - **Diameter Wear**: the diameter wear of the current tool.
   - **Tool Length**: the length of the current tool.
   - **Length Wear**: the length wear of the current tool.

3.3.9 Turn on the Spindle in Advance

It is used to turn on the spindle of the target cylinder before switching to the target cylinder, so as to reduce the spindle preparation time and improve machining efficiency. It is controlled by the parameter **Spindle before Hand Time**.

At present, it is only applicable to the mechanical structure of multi-cylinder and multi-inverter.

To modify the value of parameter **Spindle before Hand Time**, see <u>Setting Parameters</u> to find the parameter, and modify its value based on your needs.

Example

If the parameter **Spindle before Hand Time** is set to **1000** and changing from T1 to T3 is required. Then:

1. Before executing T3 command, the system turns on the spindle corresponding to T3 1000ms in advance.
2. When executing the T3 command, the system does the following:
   - Turn off the spindle corresponding to T1.
   - Pull in the cylinder corresponding to T1.
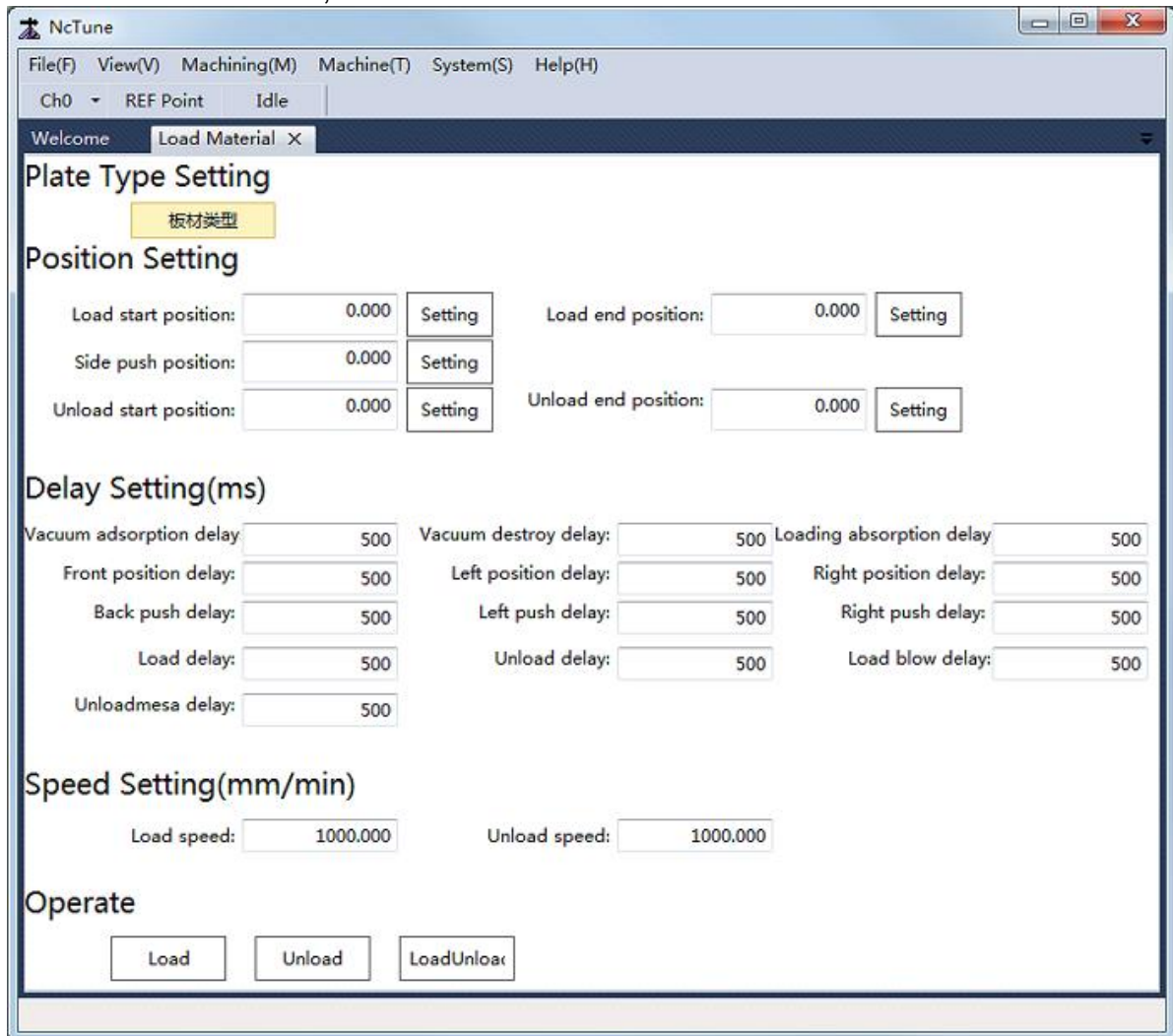   - Push out the cylinder corresponding to T3.

3.3.10 Set Parameters for Loading and Unloading

It is used to set parameters for loading and unloading, so as to improve machining accuracy and efficiency.

Before setting parameters for loading and unloading, <u>make sure loading and unloading function is configured in **NcConfig** software</u>.

To set parameters for loading and unloading, do the following:

1. To open **Load Material** page, do one of the following:
   o In **Welcome** page, click **Load Material**.
   o In the menu bar, click **View → Load Material**.

2. Set the following parameters according to your needs:
   o **Load start / end position**: the start / end position for loading material.
   o **Side push position**: the fixed position for side pushing.
   o **Unload start / end position**: the start / end position for unloading material.
   o **Vacuum adsorption / destroy delay**: the delay time for vacuum adsorption / destroy.
   o **Loading adsorption delay**: the delay time for loading adsorption.
   o **Front / Left / Right / Back position delay**: the delay time to turn on front / left / right / back positioning cylinder.
   o **Back / Left / Right / push delay**: the delay time to turn on back / left / right pushing cylinder.
   o **Load / unload delay**: the delay time to turn on loading / unloading cylinder.
   o **Load / unload speed**: the speed for loading and unloading.
3. In **Operate** area, select **Load** to load material / **Unload** to unload material / **Load and Unload** to load and unload material.

3.4   Commissioning for the Machine Tool

3.4.1 Adjust the Workpiece Origin

It is used to adjust the workpiece origin.

To adjust the workpiece origin, do the following:

1. In the menu bar, click **Machine → Datum Setting**.
2. Do one of the following:
    o To adjust the workpiece origin for all axes, click **All Axis Adjustment**.
    o To adjust the workpiece origin for a single axis, click **Adjustment(_X)(X)** / **Datum Setting(_Y)(Y)** / **Adjustment(_Z)(Z)**.

3.4.2 Execute Tool Calibration

It is used to measure the selected tool, so as to make sure it works normally during machining.

Types of tool calibration includes the following:

- **Multi-tool measure**: it is used to measure the tool on a certain fixed position of the machine tool to reconfirm tool offset.
- **First calibration / after tool change**: the first calibration is used to define the workpiece coordinate system. After tool change, please do tool calibration again to define the tool length offset.
- Mobile calibration it is used to do tool calibration in the current position.

This operation mainly focuses on the last two types of tool calibration.

To execute tool calibration, do the following:

1. To open **Calibration** page, do one of the following:
    o In **Welcome** page, click **Calibration**.
    o In the menu bar, click **Machine → Calibration**.
2. Click the dropdown box of **Calibration Mode** and select the target calibration type.
3. In **Basic Setting** area, set the following parameters:
    o **Positioning Times**: set the fine positioning times during tool calibration to improve calibration precision.
    o **Positioning Speed**: set the speed during fine positioning of tool calibration.
4. In **Fixed Tool Sensor** area, set the following parameters, and click **Set**:
    o **Fixed Point in X**: the coordinate position of the fixed tool sensor in X-axis / Y-axis direction.
    o **Start Point in Z**: the coordinate position to start tool calibration in Z-axis direction.
    o **Z Lowest Pos**: the lowest coordinate of Z-axis during tool calibration.

5. To start tool calibration, do one of the following:
   o **Multi-tool measure**
     ▪ To set the current machine coordinate of Z-axis to Z-axis tool offset of the current tool, click **Manual Set**.
     ▪ Click ****Auto Measure ****. The system automatically starts tool calibration and saves offset to the corresponding tool offset after measurement.
   o **First calibration / after tool change**
     ▪ If it is the first time to start tool calibration, click **First-time**. The system automatically starts the first-time tool calibration and saves offset to the corresponding tool offset after measurement.
     ▪ After tool change, click **After Change**. The system automatically starts the first-time tool calibration and saves offset to the corresponding tool offset after measurement.

The system starts tool calibration:

- **Multi-tool measure**
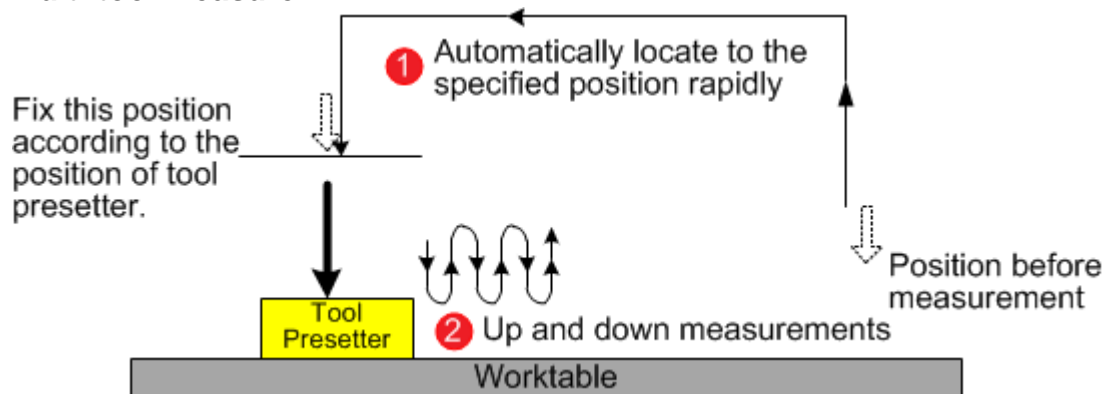
- **First calibration / after tool change**
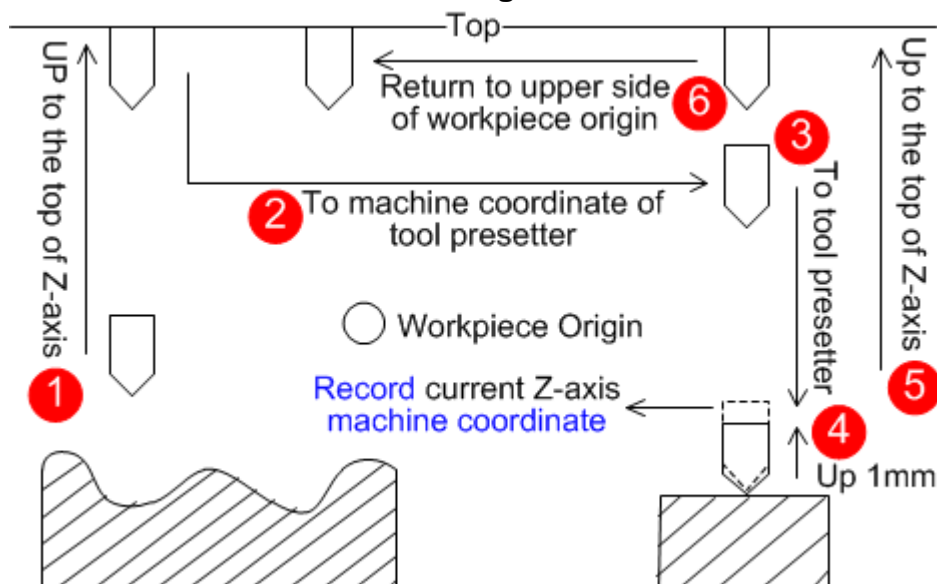
After tool calibration, to verify if the corresponding tool offset is set successfully, check parameter **Tool Offset** in **Parameter** page.

3.4.3 Set Screw Error Compensation

It is used to compensate the screw error and improve the machining precision when errors exist in the machine tool and the desired precision cannot be reached.

To set screw error compensation, see Set Parameters to find parameter **Lead Screw Compensation Mode**, and set it to a value except **0**.

- 1: backlash compensation
- 2: backlash compensation and unidirectional compensation: this operation is the same with the bidirectional compensation.
- Bidirectional compensation

3.4.3.1   Set the Backlash Compensation

To set backlash, see Set Parameters to find and set parameter **Backlash**.

3.4.3.2   Set the Bidirectional Compensation

To set bidirectional compensation, do the following:

1. To open **Screw Error Compensation** page, do one of the following:
   o In **Welcome** page, click **Screw Error**.
   o In the menu bar, click **Machine → Screw Error**.
2. To generate the curve for screw error compensation, do one of the following:
   o To import the target file of screw error compensation, click **Import**, and select the target file.
   o To add error value, click **Add**.
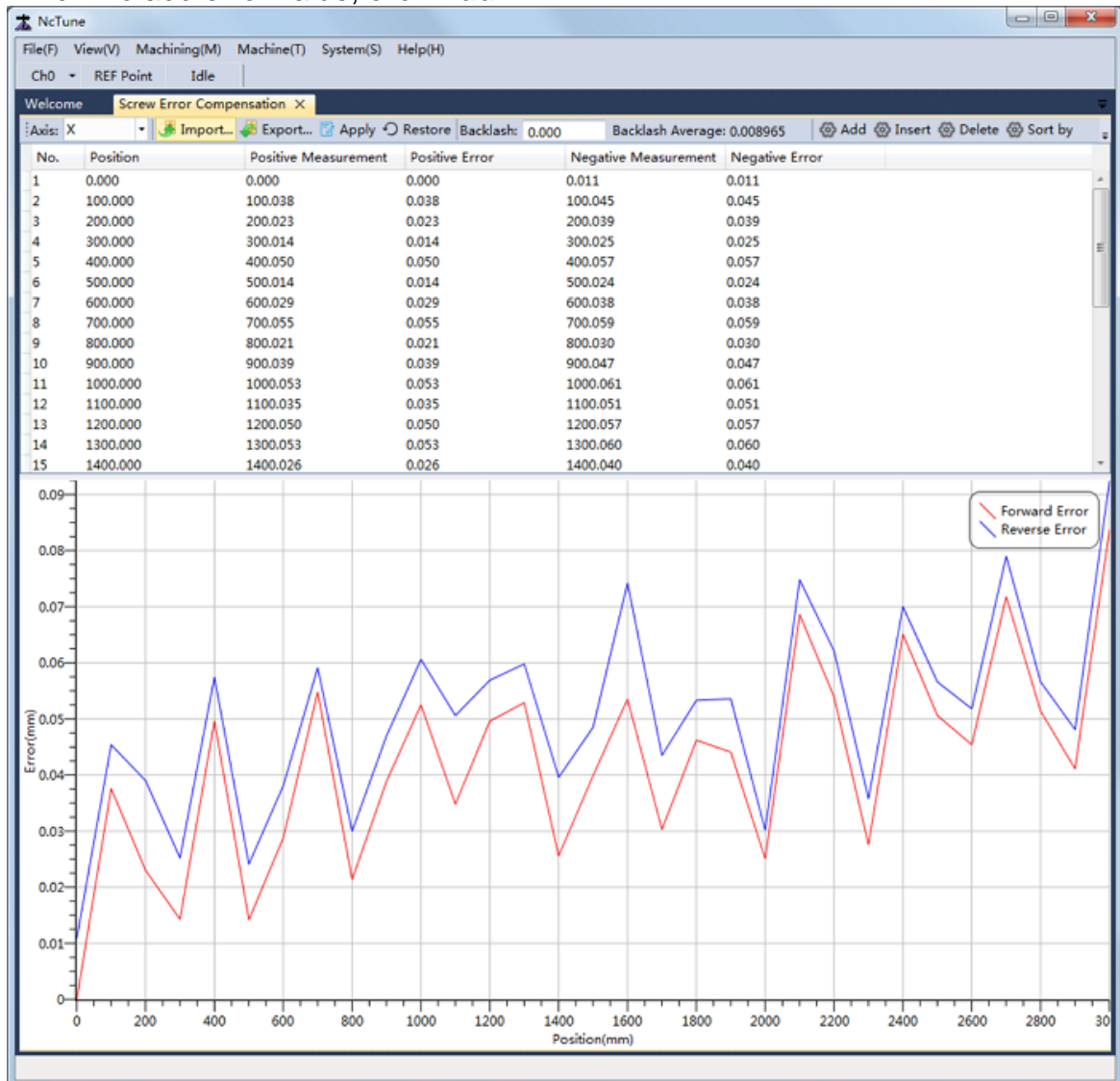
Red curve: positive error; blue curve: negative error.

3. Click **Apply**. The system automatically saves the compensation data to the corresponding configuration file.

To make compensation data effective, restart **NcTune**. During machining, the system automatically does screw error compensation based on the compensation file.

3.4.4 Execute MDI

It is used to input and execute simple commands, so as to make machine tools move quickly or execute simple operations.

To execute MDI, do the following:

1. To open **MDI** page, in the menu bar, click **Machine → MDI**:
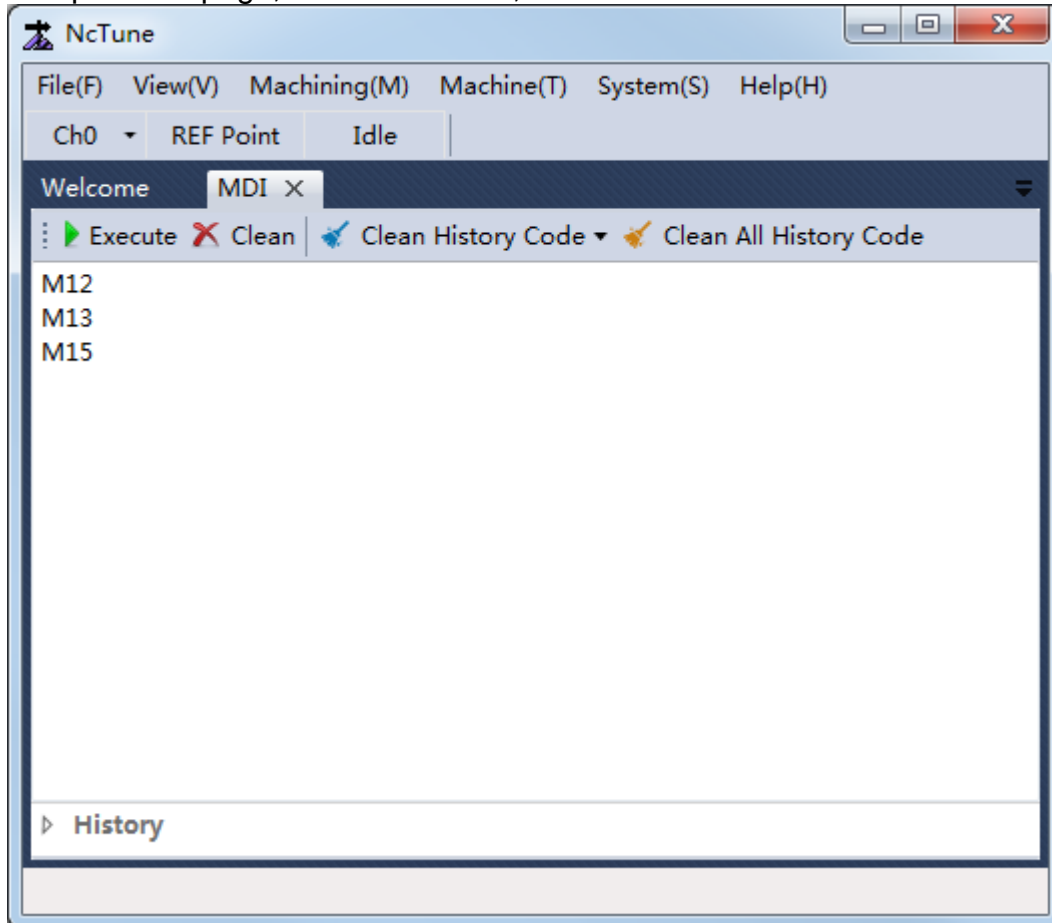
2. Enter the target commands.
3. Click **Execute**. The system starts to execute the input commands.
4. **Optional:** To clear the input commands, click **Clear**.
5. **Optional:** To clear the target history command, click the dropdown box of **Clear History Code**, and the target history command, and click **Clear History Code**. To clear all history commands, click **Clear All History Code**.

3.4.5 Execute Centering

It is used to get the center point of a regular workpiece and set it as the workpiece origin.
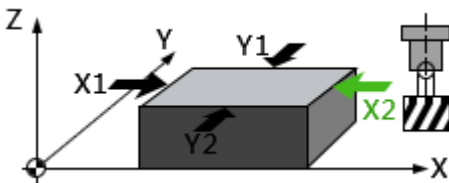
To execute centering, do the following:

1. To open **Centering** dialog box, in the menu bar, click **Machine → Centering**.
2. To select a centering type, in the dropdown box of **Centering Type**, select **Line** / **Circle**.

3. In the dropdown box of **WCS**, select the target workpiece coordinate system (G54~G59).
4. **Optional:** To enable the edge finder for accurate centering, click **Edge Finder**.
5. Do one of the following according to the centering type:
   - Line centering
   - Circle centering

3.4.5.1 Execute Line Centering

It is used to find the midpoint of a workpiece between any two points in X-axis / Y-axis direction. The shape of workpiece is rectangle, square, or regular polygon.

The schematic diagram of line centering is as follows:

Among them:

- X1 and X2: the edge of workpiece in X-axis direction.
- Y1 and Y2: the edge of workpiece in Y-axis direction.

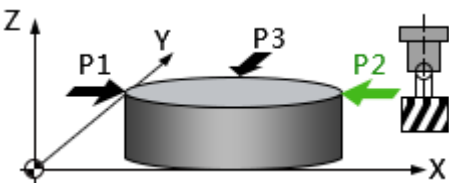To execute line centering, do the following:

1. Move the tool to X1-axis / Y1-axis position, and click **Record X** / **Record Y**. The system records the machine coordinate of the current point.
2. Move the tool to X2-axis / Y2-axis position, and click **Center X** / **Center Y**. The system automatically calculates the midpoint coordinate based on the coordinate of the current point and records in the last step, and sets it as the workpiece origin.

Note: During centering, when one axis is being centered, the other should keep still.

3.4.5.2 Execute Circle Centering

It is used to automatically calculate the center point of a circular blank via three points on the circumference. The center point will be set as the workpiece origin.

The schematic diagram of circle centering is as follows:

Among them: P1, P2, and P3: three points on the circumference.

Note: To make sure the accuracy of the circle center, the three points on the circumference should be scattered as far as possible.

To execute circle centering, do the following:

1. Move the tool to P1 position, and click **Record 1**. The system records the machine coordinate of the current point as the first group.
2. Move the tool to P2 position, and click **Record 2**. The system records the machine coordinate of the current point as the second group.

3. Move the tool to P3 position, and click **Centering**. The system records the machine coordinate of the current point as the third group, and automatically calculates the coordinates of the circle center based on the three groups and sets it as the workpiece origin.

3.4.6 Adjust Y1-axis and Y2-axis

It is used to separately adjust Y1-axis and Y2-axis. You can adjust the feedback coordinates of Y1-axis and Y2-axis to the same according to the actually synchronous deviation.

To adjust Y1-axis and Y2-axis, do the following:

1. In the menu bar, click **Machine → Y1Y2 Jiggle**. **Y1Y2 Jiggle** dialog box pops up:
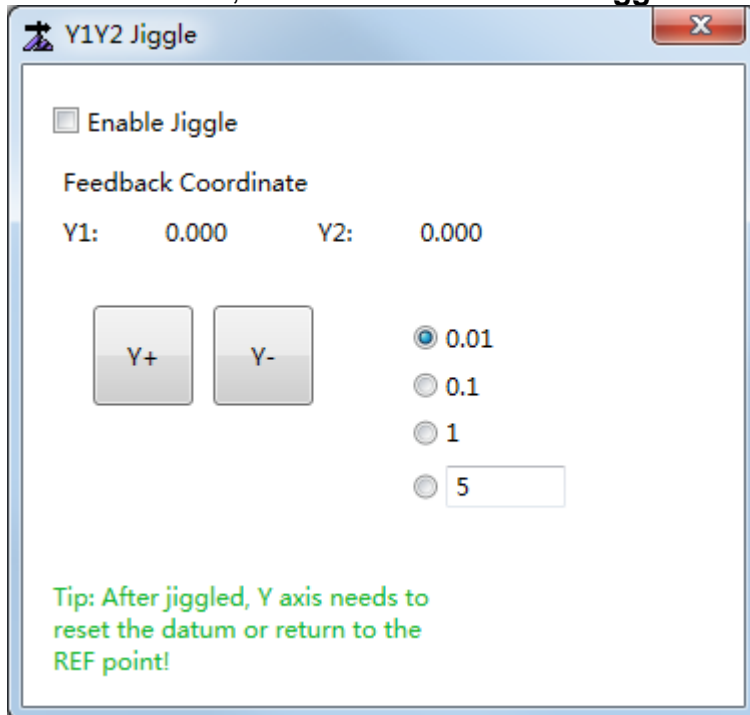
2. Check **Enable Jiggle**.
3. Select a step size (0.01, 0.1 and 1) or customized a step size.
   The customized step size is 5 by default.
4. To adjust the feedback coordinates of Y1-axis or Y2-axis, click **Y1+** / **Y1-** / **Y2+** / **Y2-**.
After Y1-axis and Y2-axis adjustment, execute returning to the machine origin or set datum again.

3.4.7 Execute Data Acquisition

It is used to collect the specified data for each axis, and check sent pulses and feedback pulses.

The specified data include the following:

- Command position in the command / machine coordinate system
- Positional deviation
- Feedback / feedback latch position in the machine coordinate system
- Internal command position in the command coordinate system
- Feedback / command / target speed
- Command torque

To execute data acquisition, do the following:

1. To open **Data Acquisition** page, do one of the following:
   o In **Welcome** page, click **Data Acquisition**.
   o In the menu bar, click **Machine → Data Acquisition**.
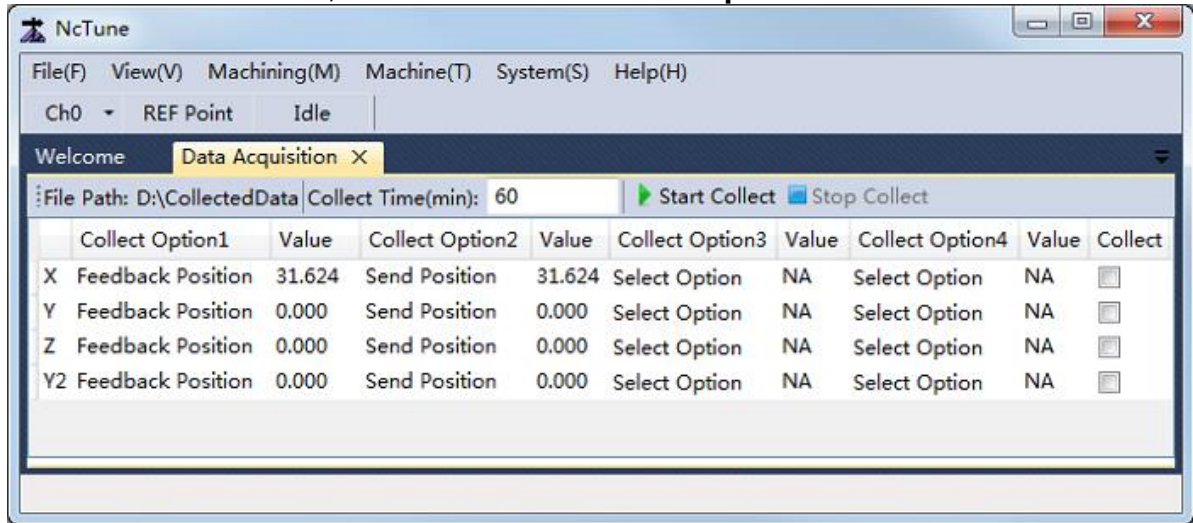
2. Select the target axis and check **Collect**.
3. Double click **Collect Option3** / **Collect Option4**, and select the target data.
4. In the input box of **Collect Time**, input a value and click **Start Collect**.

3.4.8 Execute Learning for the Circularity Test

It is used to run a circle and collect data, to provide reference for circularity test.

To execute learning for the circularity test, do the following:

1. To open **QEC Learning** page, do one of the following:
   o In **Welcome** page, click **QEC Learning**.
   o In the menu bar, click **Machine → QEC Learning**.
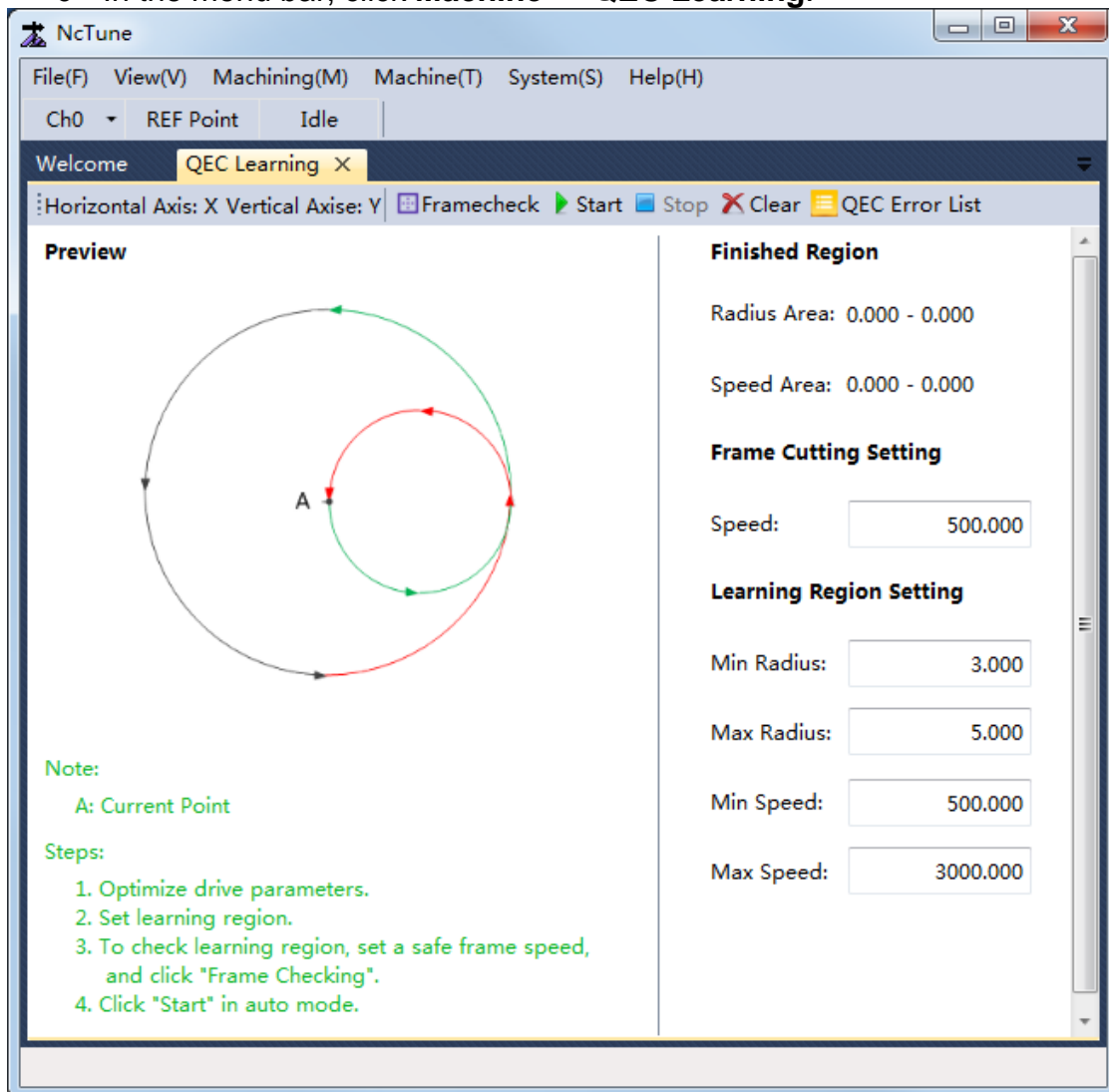
2. In **Frame Cutting Setting** area, set parameter **Speed**, and click **Framecheck**.
3. In **Frame Region Setting** area, set parameters **Min Radius**, **Max Radius**, **Min Speed**, and **Max Speed**.
4. Click **Start**.
5. **Optional:** To check learning data, click **QEC Error List**.

3.4.9 Execute Commissioning for the Circularity Test

It is used to remove distortion phenomenon (usually sharp corner) at the transition caused by one quadrant entering into another.

Before executing commissioning for the circularity test, make sure learning for the circularity test has been executed.

To execute commissioning for the circularity test, do the following:

1. In **QEC Learning** page, click **QEC Error List** and record a group of data.

2. To open **QEC Debugging** page, do one of the following:
    o In **Welcome** page, click **QEC Debugging**.
    o In the menu bar, click **Machine → QEC Debugging**.
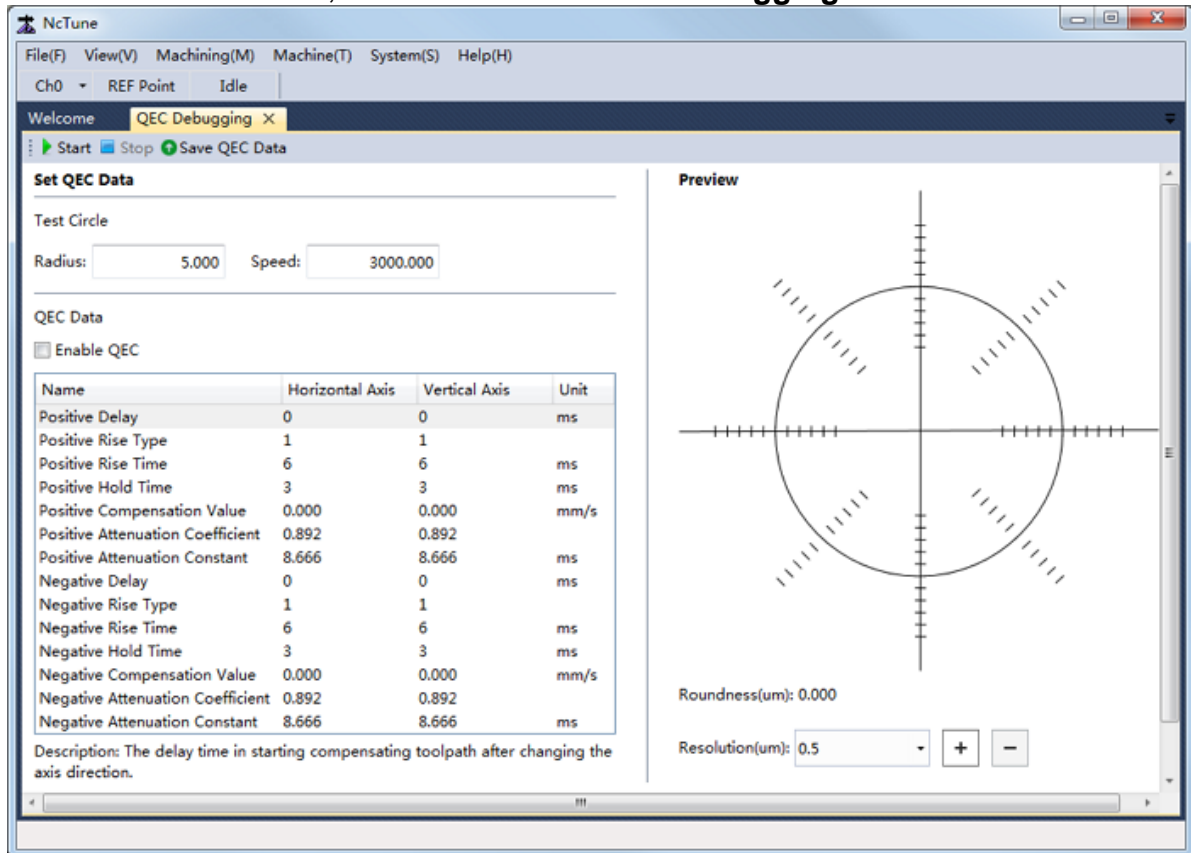
3. Input the recorded data in **Test Circle** area.
4. In **QEC Data** area, check **Enable QEC**.
5. Click **Start**.
6. According to the preview in the right, adjust parameters for the circularity test. When the result is close to a circle and no indent phenomenon occurs, commissioning finishes.
7. **Optional:** To save data for the circularity test, click **Save QEC Data**.

3.5 Machining Commissioning

3.5.1 Modify Password

It is used to modify password that used to enter the main interface of **NcTune** software.

To modify password, do the following:

1. In the menu bar, click **System → Modify Password**. **Modify Password** dialog box pops up:
2. Enter the old password.
3. Enter the new password.
4. Enter the new password again.
5. Click **Modify**.

3.5.2 Upgrade Firmware

After upgrading firmware, software function can be improved and the stability of the system can be enhanced.

Note: Please execute this operation in the help of the technician.

To upgrade firmware, do the following:

1. To open **Update Firmware** page, do one of the following:
   o In **Welcome** page, click **Update Firmware**.
   o In the menu bar, click **System → Update Firmware**.
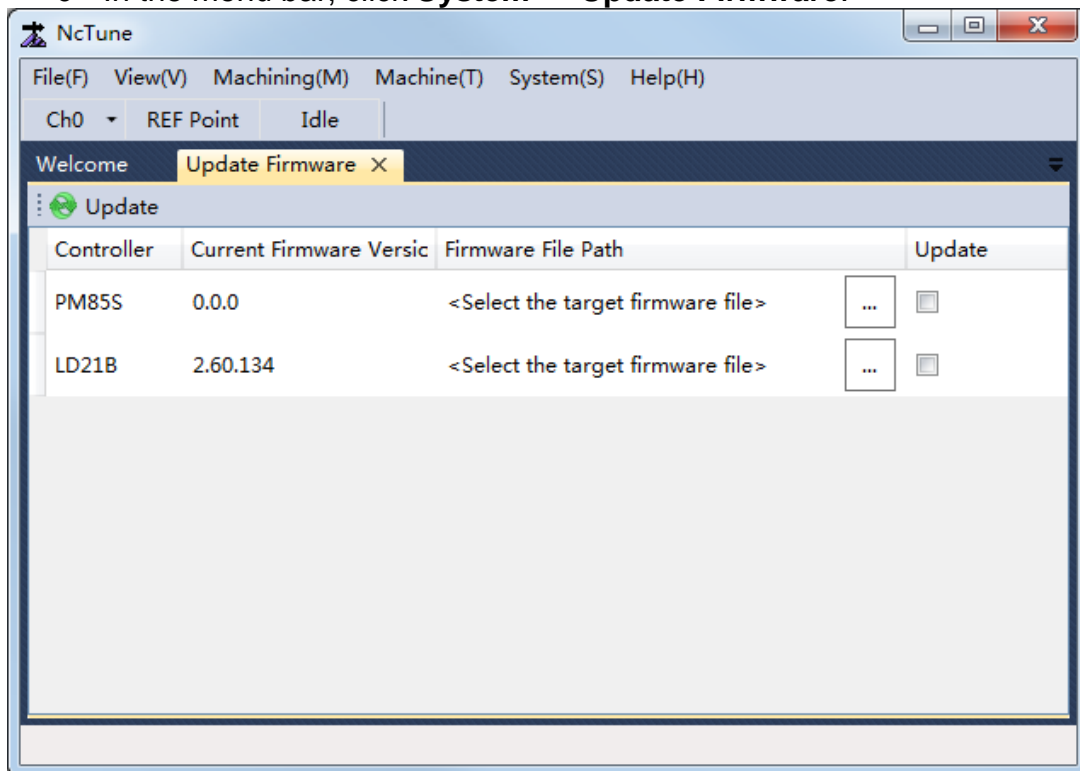
2. Select the target controller and check **Update**.

3. In **Firmware File Path** column, click [...], select the target file under the target file path, and click **Open**.

4. To start upgrade the firmware, in the upper left corner, click Update.

After the system upgrades the firmware, sign shows in front of the corresponding Lambda terminal board or terminal board and the prompt *Succeeded in upgrading the firmware. Please restart the system and hardware* shows.

3.5.3 Load a Program File

It is used to load a program file for machining.

To load a program file, in the menu bar, click **File → Open and Load**, and select the target program file. The system automatically loads the program file.

You can check the track in **Tool Path&Trace** window of **NcStudio** software.

To unload the program file, click **File → Unload**. The track in **Tool Path&Trace** window of **NcStudio** software disappears.

3.5.4 Generate an Installation Package

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

This operation will pack up the current folder whose name begins with *Default_Config*.

To generate an installation package, do the following:

1. In the menu bar, click **File → Pack Up Software**. **Packup Tool** dialog box pops up:

2. Modify the name and storage path of the installation package.

3. **Optional:** Click **Advanced**, and set the following advanced settings:

- o Language: At present, the system supports Chinese and English.
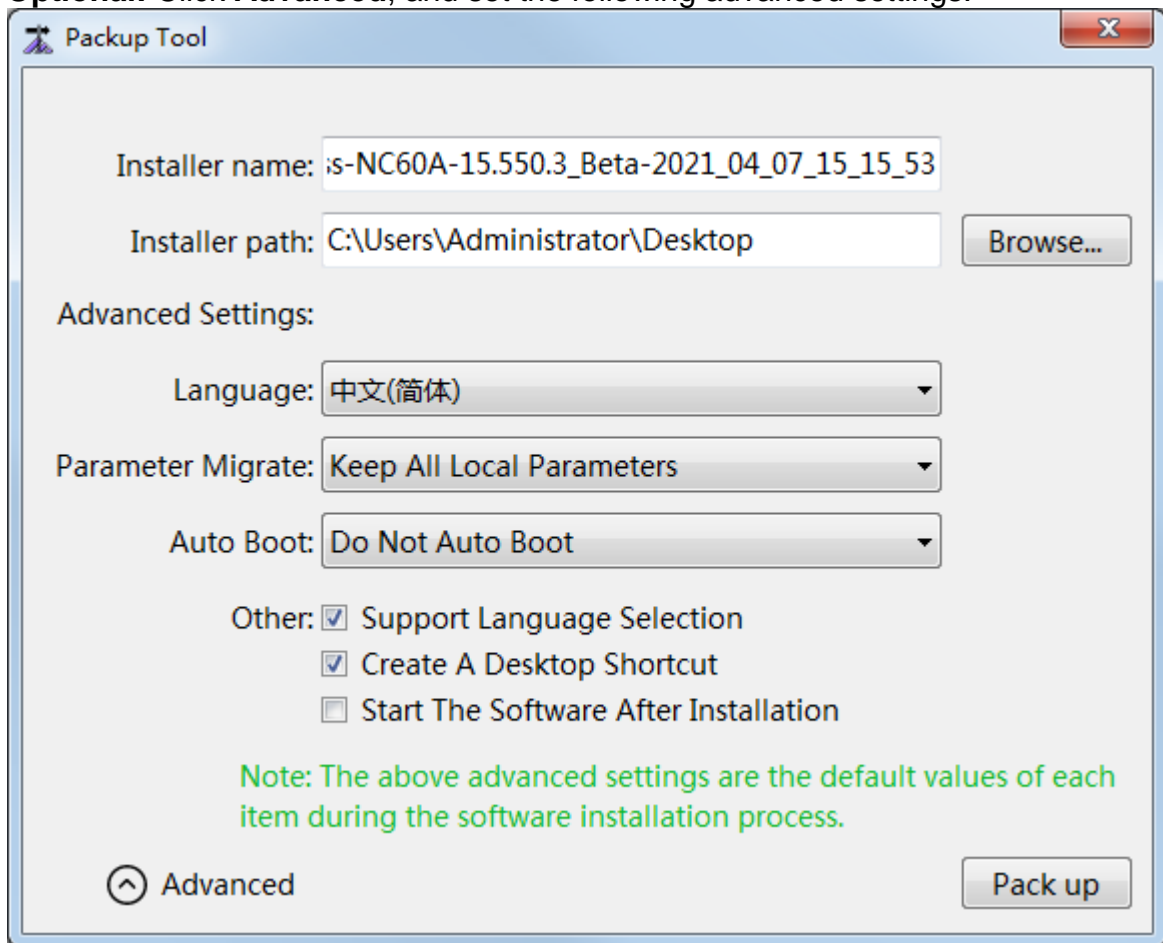- o Parameter migration:
  - **Keep All Local Parameters**: use all parameters in the local.
  - **Keep Only Machine-Specific Parameters**: use parameters about machine characteristics in the local.
  - **Fresh Installation**: fresh Installation
- o Auto boot: whether to automatically start the software once the machine boots.
- o Others
  - **Support language selection**: whether to support switching the current language.
  - **Create a desktop shortcut**: whether to create a shortcut on the desktop.
  - **Start the software after installation**: whether to automatically start the software after installation.
4. Click **Pack up**. The system starts to generate the installation package.

After generating an installation package, find the generated installation package under the set path.

3.5.5 Start Machining

It is used to start automatic machining from the first program file to the last program file in the machining task.

Before starting machining, make sure the following:

- Returning to the machine origin has been executed.
- No alarms like E-stop alarm.

In the menu bar, click **Machining** → **Start**. The system automatically starts machining from the first program file to the last program file.

During automatic machining, you can also do the following:

- To pause machining, in the menu bar, click **Machining** → **Pause**.
- To stop machining, in the menu bar, click **Machining** → **Stop**.
- To resume machining from the exact interrupted position when power interruption or E-stop occurs, in the menu bar, click **Machining** → **Resume**.
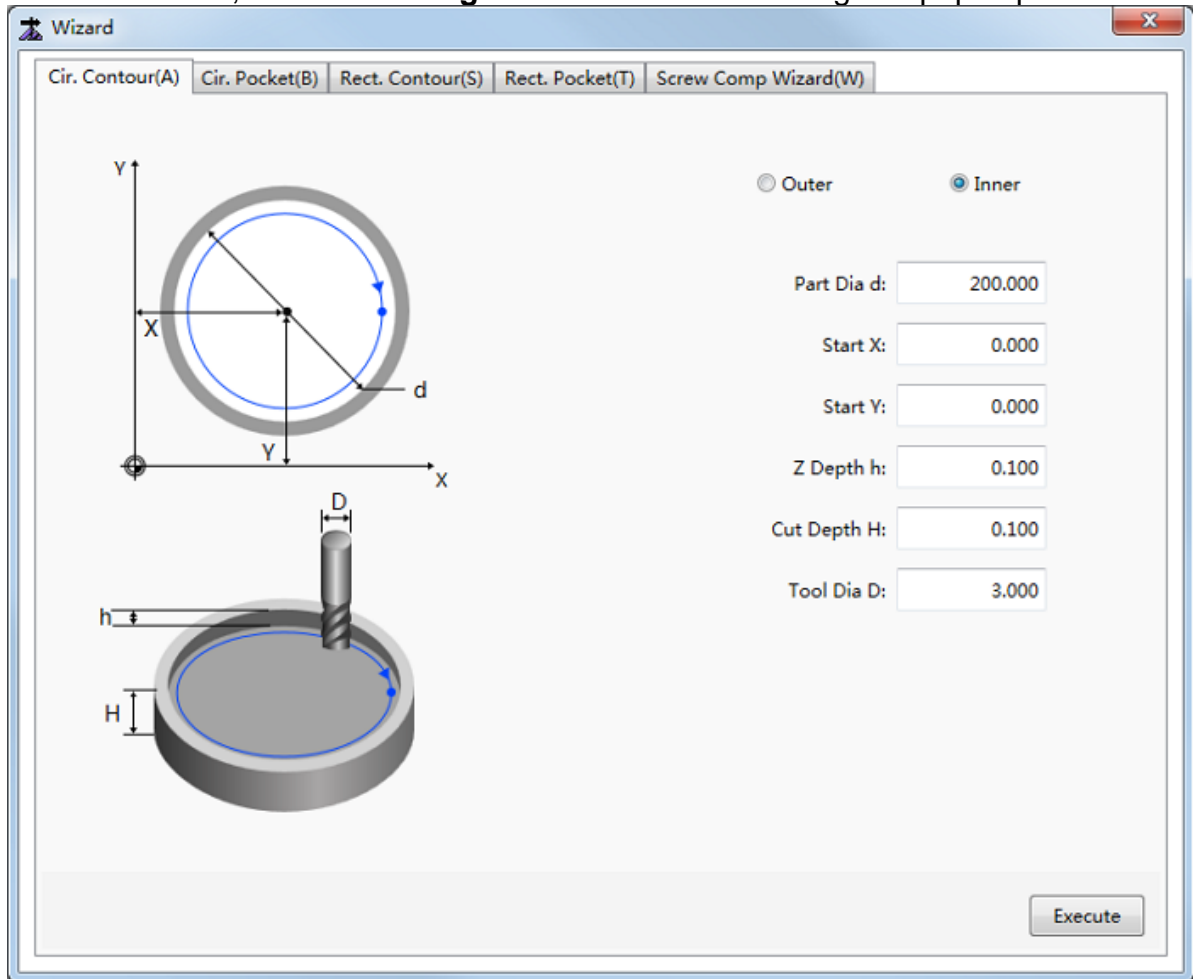
3.5.6 Use Machining Wizard

It is used to quickly generate the machining program with the machining wizard.

With the machining wizard, you only need to set related parameters and save setting, and the system will automatically generate the machining program. Compared to traditional manual programming, this way is simpler and more convenient. Besides, you can also export the machining program generated by the wizard to the local for later use.

To use machining wizard, do the following:

1. In the menu bar, click **Machining** → **Wizard**. **Wizard** dialog box pops up:

2. Select the target wizard, and set related parameters.
    The supported wizard includes the following:
    o   Circle contour
    o   Circle pocket
    o   Rectangle contour
    o   Rectangle pocket
    o   Screw comp wizard

3.5.6.1　　Set Machining Wizard for Circle Contour

It is used to engrave the outer frame or inner frame of the circle.

The schematic diagram is as follows:
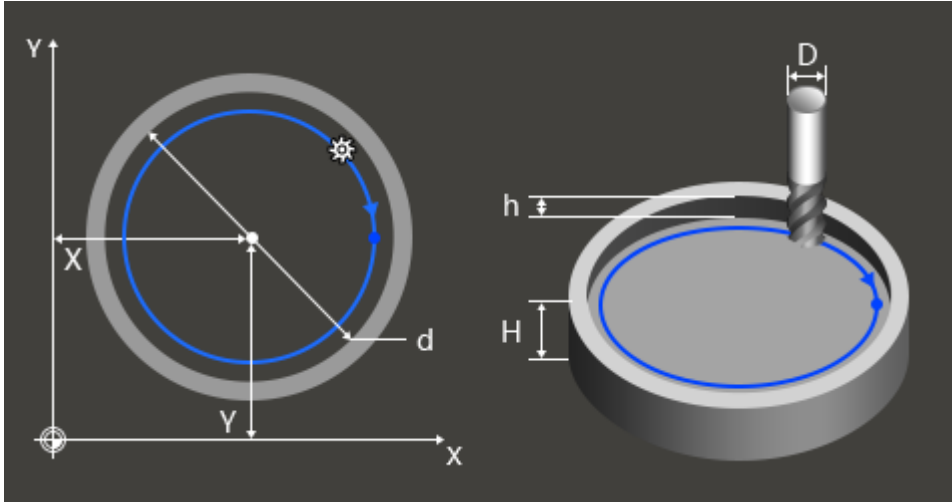
- Outer frame

- Inner frame

To set machining wizard for circle contour, check **Outer** / **Inner**, and set the following parameters:

Part Dia d

- **Start X / Y**
- **Z Depth h**
- **Cut Depth H**
- **Tool Dia D**:

3.5.6.2 Set Machining Wizard for Circle Pocket

It is used to engrave the bottom of the circle.
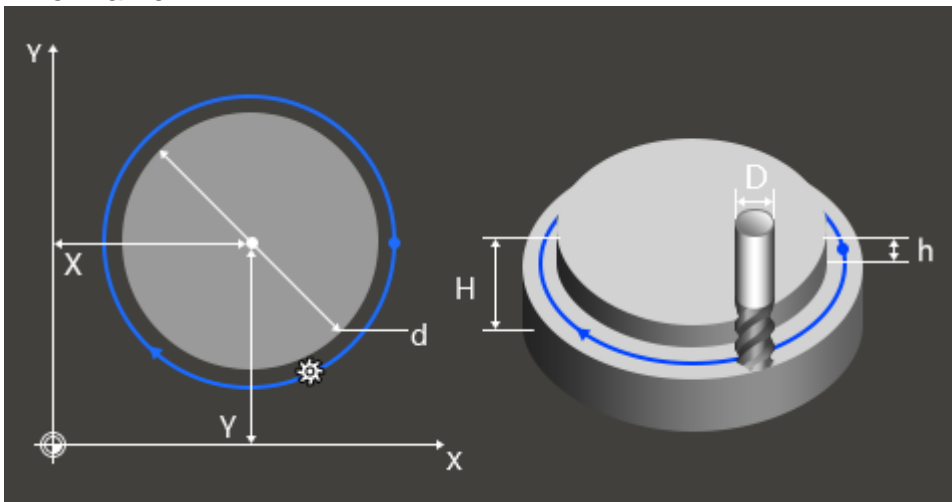
The schematic diagram is as follows:

To set machining wizard for circle pocket, set the following parameters:

- **Part Dia d**
- **Start X / Y**
- **Z Depth h**
- **Cut Depth H**
- **Tool Dia D**:
- **Nose Space T**

3.5.6.3 Set Machining Wizard for Rectangle Contour

It is used to engrave the outer frame or internal frame of the rectangle.

The schematic diagram is as follows:
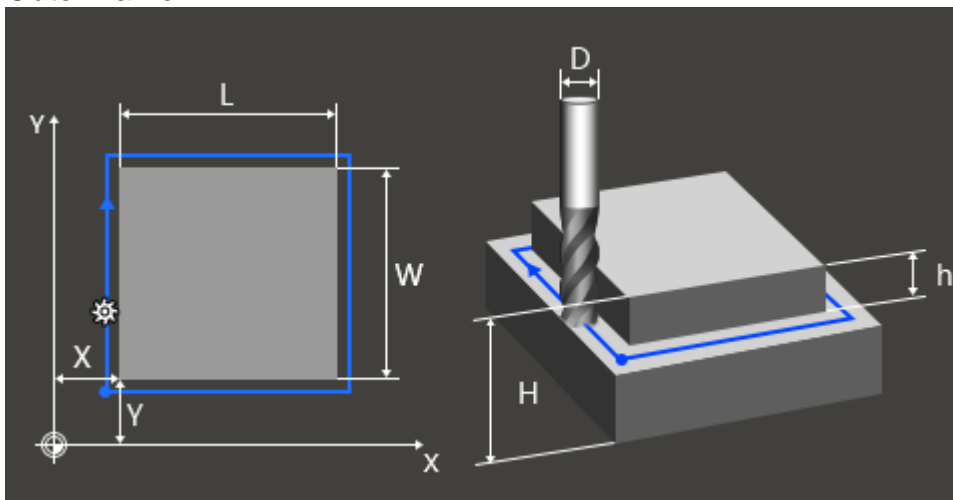
- Outer frame

- Inner frame
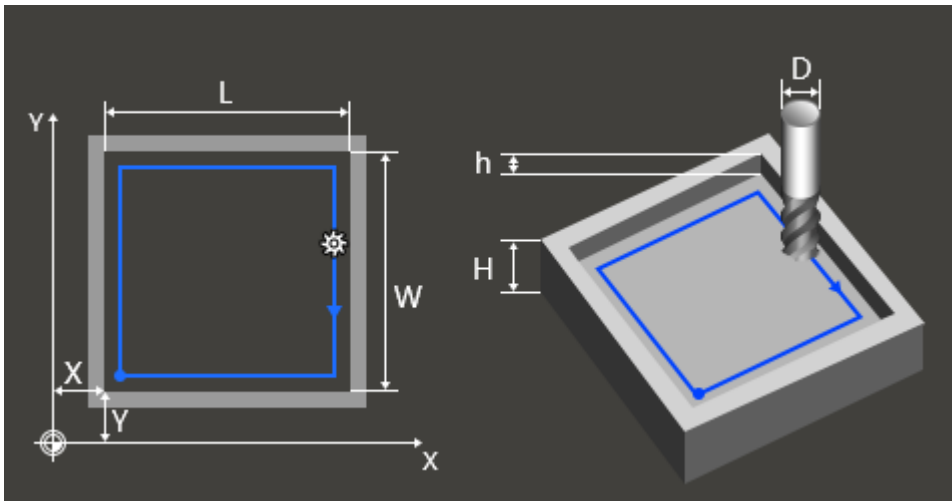
To set machining wizard for rectangle contour, check **Outer** / **Inner**, and set the following parameters:
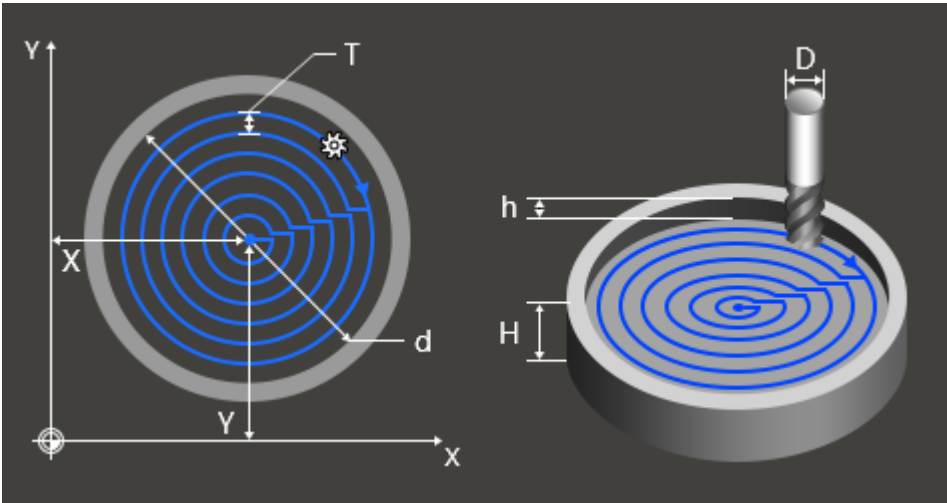
- **Length L**
- **Width W**
- **Start X / Y**
- **Z Depth h**
- **Cut Depth H**
- **Tool Dia D**:

3.5.6.4 Set Machining Wizard for Rectangle Pocket

It is used to engrave the bottom of the rectangle.

The schematic diagram is as follows:

- Horizontal milling
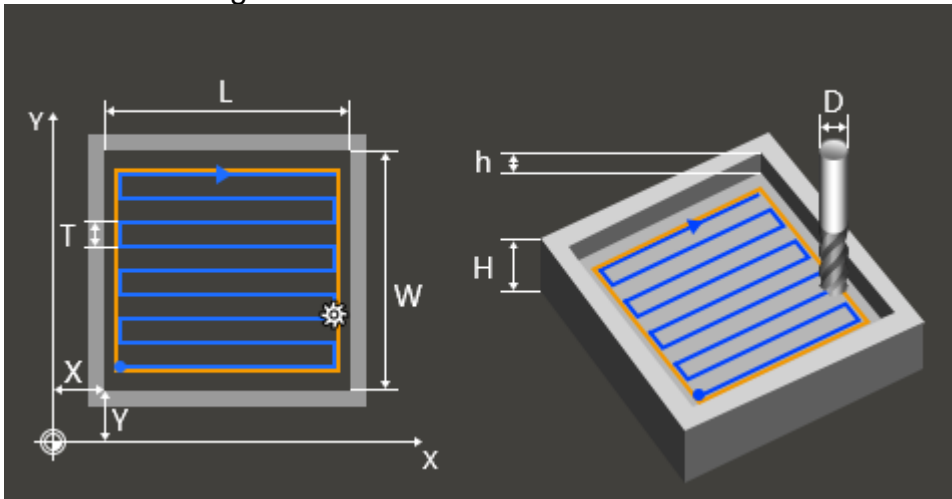
- Vertical milling
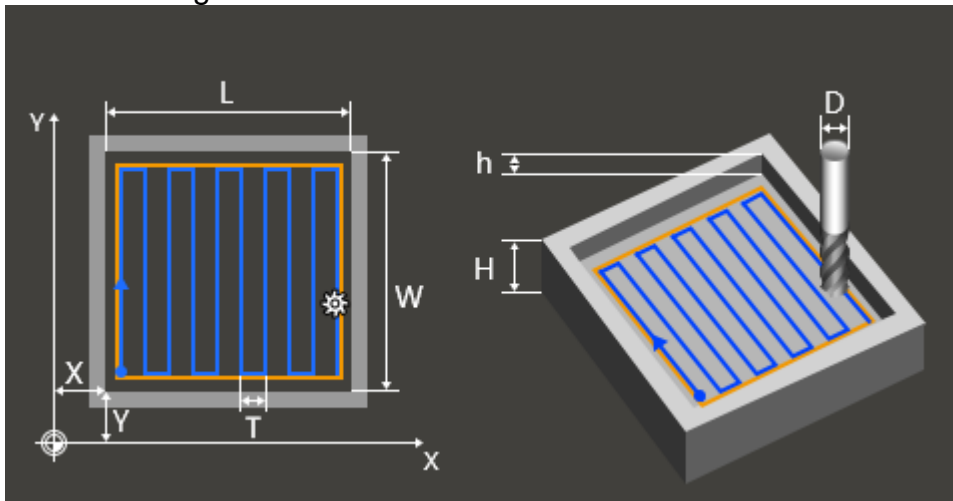
To set machining wizard for rectangle pocket, check **Vertical** / **Horizontal**, and set the following parameters:

- **Length L**
- **Width W**
- **Start X / Y**
- **Z Depth h**
- **Cut Depth H**
- **Tool Dia D**:
- **Nose Space T**

3.5.6.5 Set Screw Compensation Wizard

It is used to use laser interferometer to measure lead screw error.

The schematic diagram is as follows:
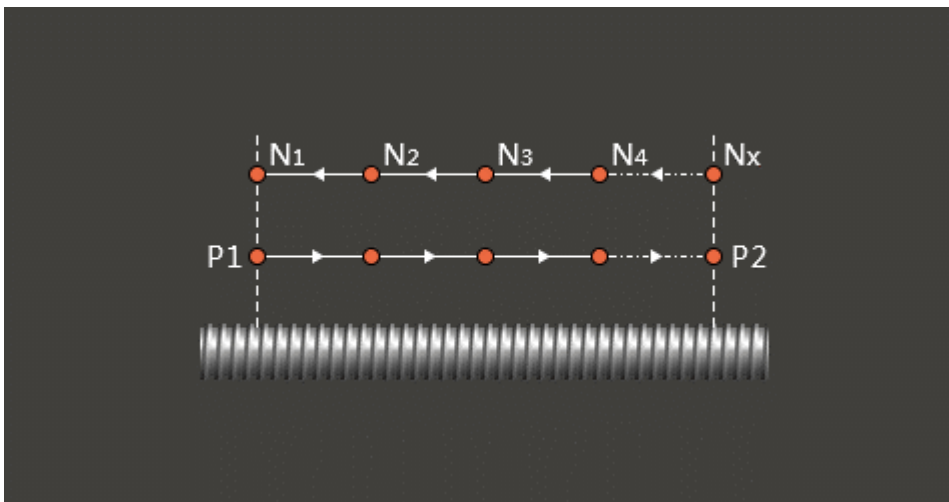
1. Check **X** / **Y** / **Z**, and set the following parameters:
   - **Start P1**
   - **End Position P2**
   - **Measure N1~Nx**
2. Set parameter **Cycle Times**.
3. To conveniently check machining effect, set parameter **Pause**.

4   Settings in NcStudio Software

4.1   Overview

Through this part, you can quickly know the machining operations of **NcStudio** software in WEIHONG **NcStudio CNC System for Cutting Machines**.

4.2   Quick Start

4.2.1   Overview

Through this part, you can quickly know the basic machining process of WEIHONG **NcStudio CNC System for Cutting Machines**.

The machining process is as follows:
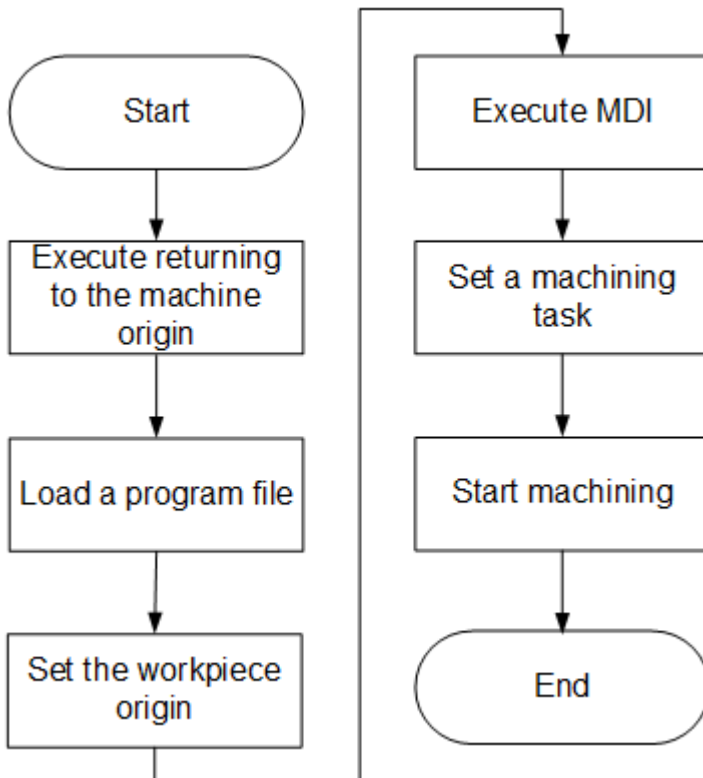
4.2.2   Execute Returning to the Machine Origin

The origin of the machine coordinate system (inherent coordinate system of a machine tool), also called mechanical origin or home, is a fixed point assigned by designing, manufacturing and debugging before the machine tool leaves the factory.

This operation is used to synchronize the machine coordinate system of the system and the machine tool.

Before executing returning to the machine origin, make sure all servo alarms have been removed.

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

1. In the upper right corner of the main interface of **NcStudio** software, click the hamburger menu ▤ → **Homing**. **Homing** dialog box pops up:
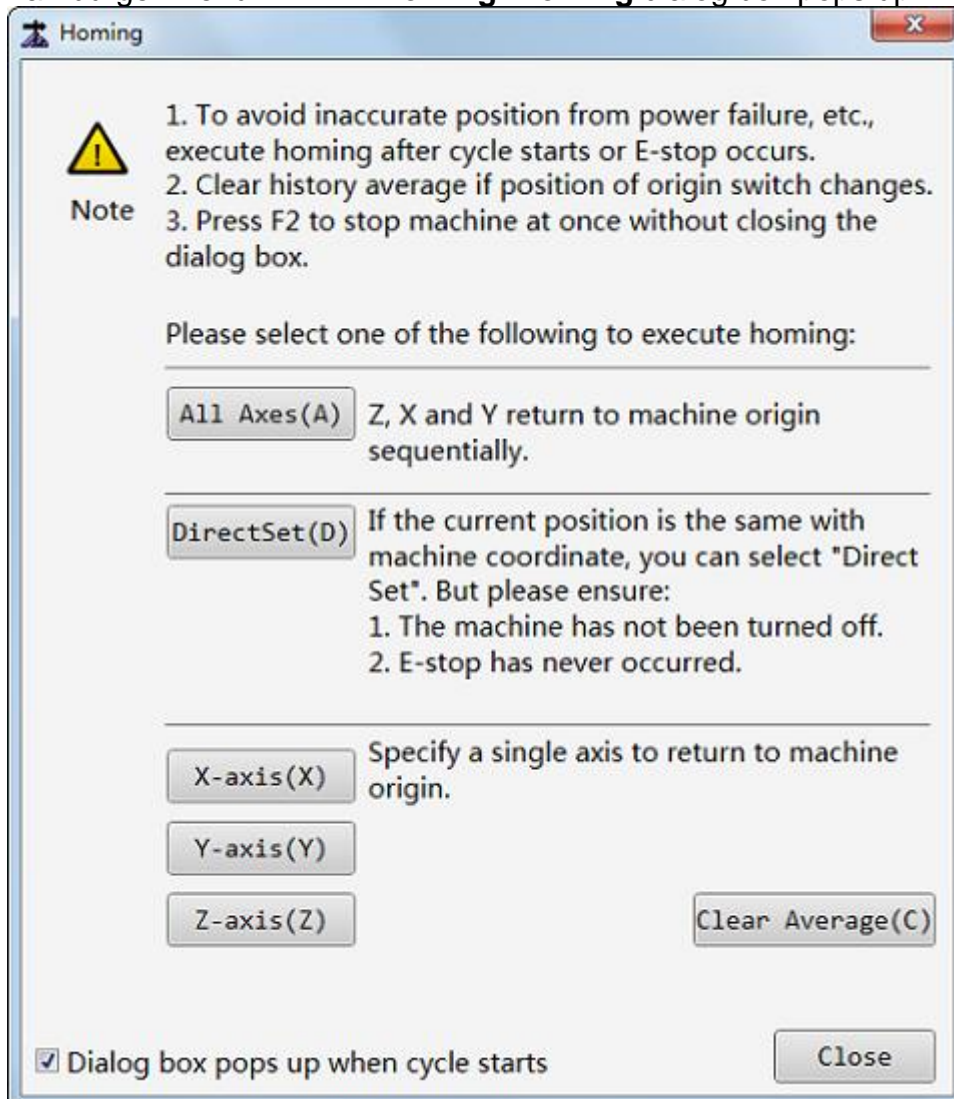
If **Dialog box pops up when cycle starts** in the lower left corner is checked, this dialog box will automatically pop up when **NcStudio** software starts.

2. **Optional:** If the switch position of the machine origin changes, click **Clear Average**, and click **Clear** in **History Average** dialog box.
   Otherwise, a prompt *Compared to history average, distance between coarse positioning signal and fine positioning signal exceeds the allowable range. Failed to execute returning to machine origin* pops up.

3. Do one of the following:
   o To return all axes to the machine origin, click **All Axis**. Z-axis returns to the machine origin firstly, then X-axis, and Y-axis return in sequence.
   o To return a single axis to the machine origin, click the button for a single axis, the target axis returns to the machine origin.
   o If you make sure that the current position is the same with the machine coordinate, the machine tool has never been turned off and E-stop has never occurred, to set the current position as the machine origin, click **Direct Set**.

After executing returning to the machine origin, sign ⊕ appears in front of each axis in the axis coordinate display bar.

4.2.3 Load a Program File

It is used to load a program file for machining.

In the operation button bar, click [F1 Load], and select the target program file. The system automatically loads the program file and shows its track in **Tool Path&Trace** window.
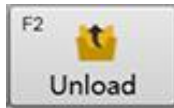
To unload the program file, click [F2 Unload]. Track in **Tool Path&Trace** window disappears.

4.2.4 Set the Workpiece Origin

The zero point of X-axis / Y-axis / Z-axis / A-axis in the program file is the workpiece origin. It is used to define the actual position of the workpiece origin in the board.

It is used to clear the current workpiece coordinate of each axis and set the machine coordinate of each axis to the workpiece offset.

To set the workpiece origin, do the following:

1. Move the machine tool to the position where is the target workpiece origin.
2. In the operation button bar, select a clearing type in the dropdown box of [F3 0 All Clear ▾].
3. Click [F3 0 All Clear ▾]. A prompt *Are you sure to set the current machine coordinate to the workpiece origin* pops up.
4. Click **Yes**.

4.2.5 Execute MDI

It is used to input and execute at most 8 simple commands, so as to make machine tools move quickly or execute simple operations.

To execute MDI, do the following:

1. In the operation button bar, click [F4 MDI]. **MDI** dialog box pops up:
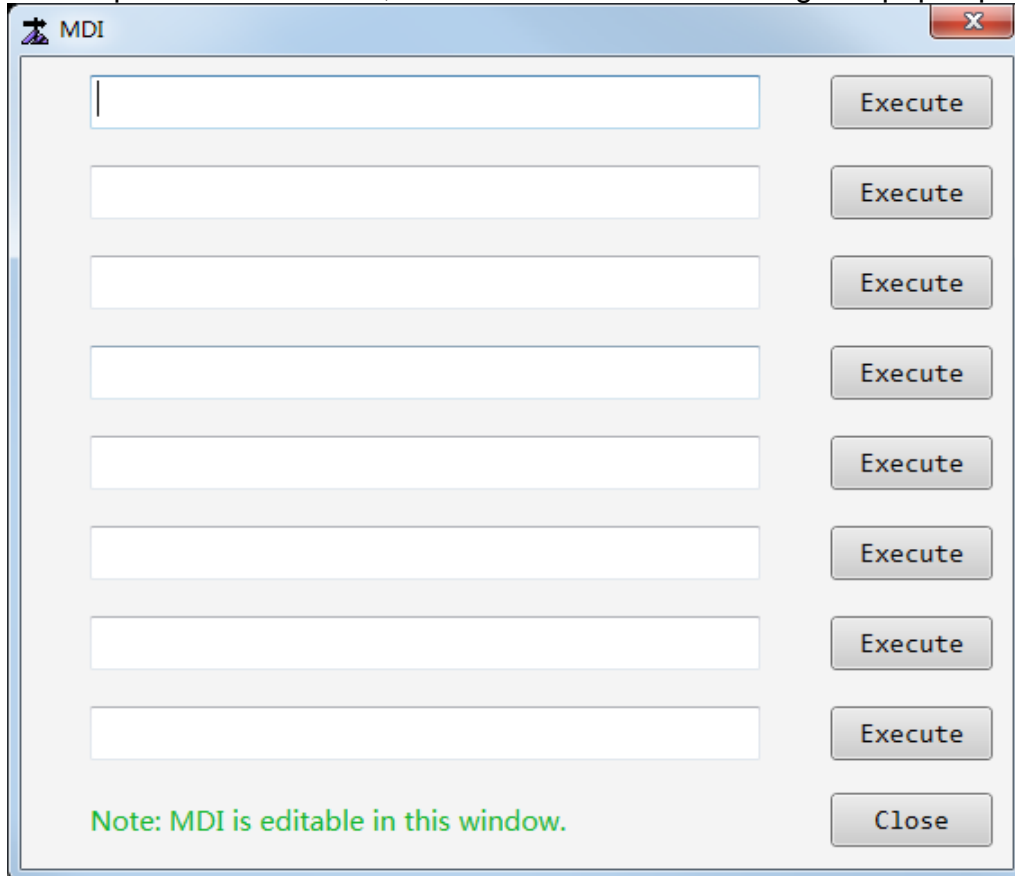
2. Enter the target commands.
3. Click **Execute**. The system starts to execute the input commands.

4.2.6 Set a Machining Task

It is used to start machining for all program files in a machining task. And you can set different machining interval, workpiece coordinate system, and machining times for each program file, and freely choose whether the target program file is involved in machining.

Before setting a machining task, make sure the following:

- Names of program files in the machining task are unique.
- The format of program file is NC.

To set a machining task, do the following:

1. To switch to **Multi-task Manage** window, in the function window area, click **Multi-task Manage**:

Number	File name	Area	State	Done	Times	Enable
1	1.nc	1	● Wait	0	1	☑
2	2.nc	2	● Wait	0	1	☑
3	3.nc	1	● Wait	0	1	☑
4	4.nc	2	● Wait	0	1	☑
5	5.nc	1	● Wait	0	1	☑

Tool Path&Trace | Multi-Task Manage | Offset Manage | Craftsmanship - **Cutting** | Drill

MultiTask Processing **ON** ○   Current task name: * Task

Cur Area: 0   Ready Area: 0

Save&Load   Delete All   Permutation setup   Reset area1 file   Reset area2 file
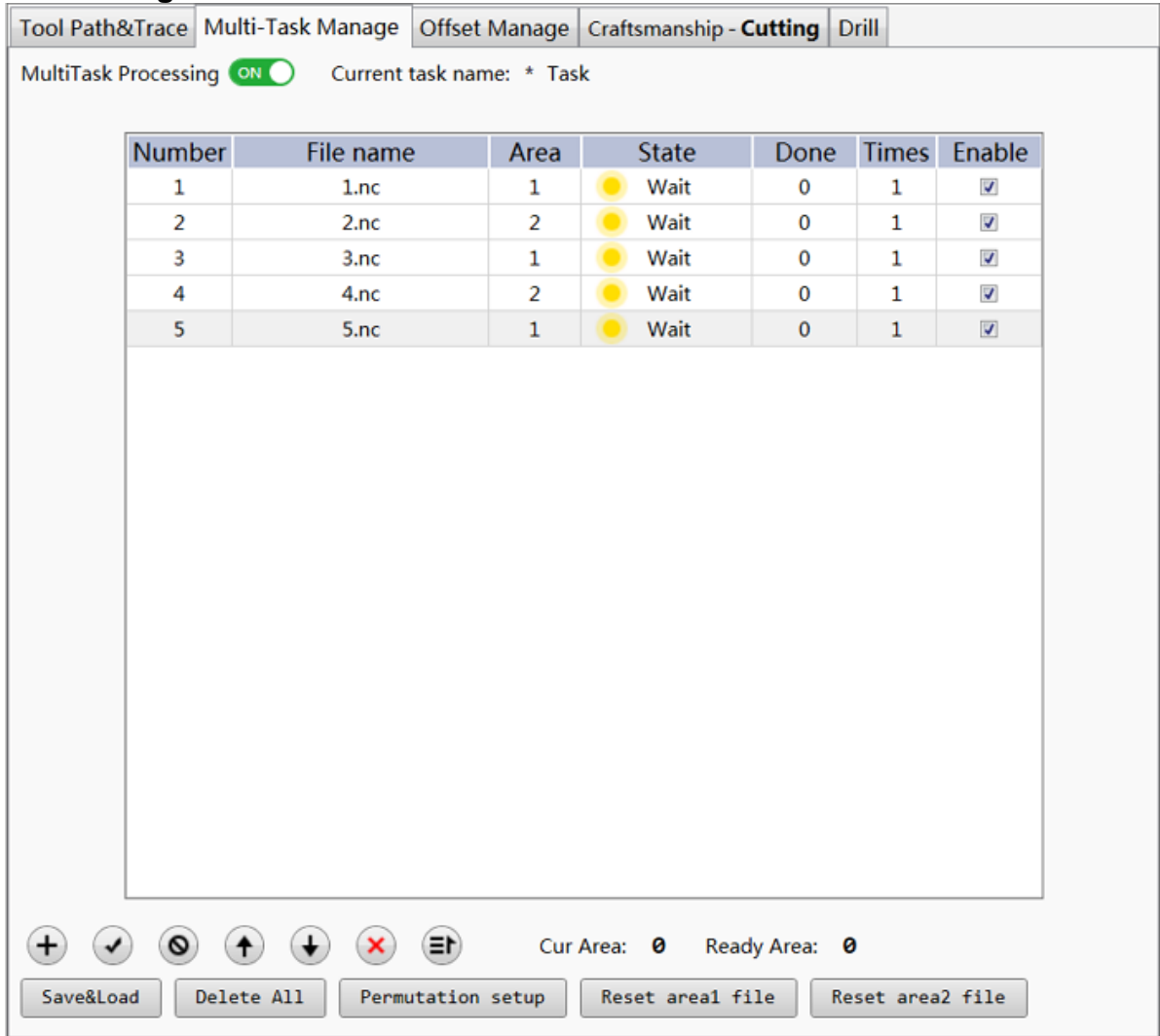
2. Change **Multi-task Processing** switch to **ON** status.

3. To add the target program file to the machining task, click ⊕, and select the target program file.
   You can at most add 60 program files to the machining task. And the status of added program files is waiting for machining.

4. To edit the machining task, do the following:

   ○ To enable all added program files, click ✔.

   ○ To disable all added program files, click ⊘.
   ○ The status of added program files is cancellation of machining.

   ○ To adjust the machining order of the selected program files, click ↑ or ↓.

   ○ To delete a program file, select the target program file, and click ✖.
   If you need to delete all program files, please click **Delete All**.

   ○ Click ≡↑. If a front / back identifier exists, the system firstly machines the program file with a back identifier and then the program file with a front identifier.
   See Sort Program Files in a Machining Task for details.

5. **Optional**: To replace the target program file with another one, double click the name of the target program file and select another program file.

6. To set the machining times for the target program file, double click the table cell in **Times** column, and set a value.

7. **Optional:** To set sorting rules, click **Permutation setup**, see Sort Program Files in a Machining Task for details.

8. **Optional:** To reset the status of station 1 or 2 to waiting for machining and clear machined time, click **Reset area 1 / 2 file**.

9. To save the current machining task to TSK format, click **Save&Load**.
   If the modification is not saved, sign * shows in **Current task name** in the upper right corner of **Multi-task Manage** window.

4.2.7 Start Machining

It is used to start automatic machining from the first program file to the last program file in the machining task.

Before start machining, make sure no alarms like E-stop alarm occur.

To start machining, do one of the following:

- In the operation button bar, click [F9 ▶ Start]. The system automatically starts machining the loaded program file.
  During automatic machining, you can also do the following:

   ○ To pause machining, click [F10 ‖ Pause].

   ○ To stop machining, click [F11 ■ Stop]. To resume machining from the exact interrupted position when power interruption or E-stop occurs, click [F12 ▮▶ Resume].

- In the idle status, in the operation button bar, click [F7 Selective], set the machining range by setting the start line and end line, and click **OK**. The system automatically starts machining from the start line to the end line.

- With the handwheel, in **Auto** mode, in the operation button bar, click [F6 HW Guide]. The system is controlled by the handwheel.

4.3 System Operations

4.3.1 Set Tool Compensation

Tool compensation includes tool length compensation and tool radius compensation.

The cutting part actually involved is the tool nose or the cutting edge which has dimensional variation with the tool center or the related point of the tool rest, so the CNC system has to compute the corresponding coordinates of the tool center or the related point of the tool rest according to the actual coordinate position of the tool rest or the cutting edge (namely the actual coordinate position of the part profile), which is called tool compensation.

Note: Please set tool compensation in the help of the technician.

Before setting tool compensation, set the following parameters to **Yes**:

- **Enable Tool Length Compensation**
- **Enable Tool Radius Compensation**

To set tool compensation, do the following:

1. In the upper right corner of the main interface of **NcStudio** software, click the hamburger menu ▤ → **Tool Parameter**. **Tool Parameter** dialog box pops up:

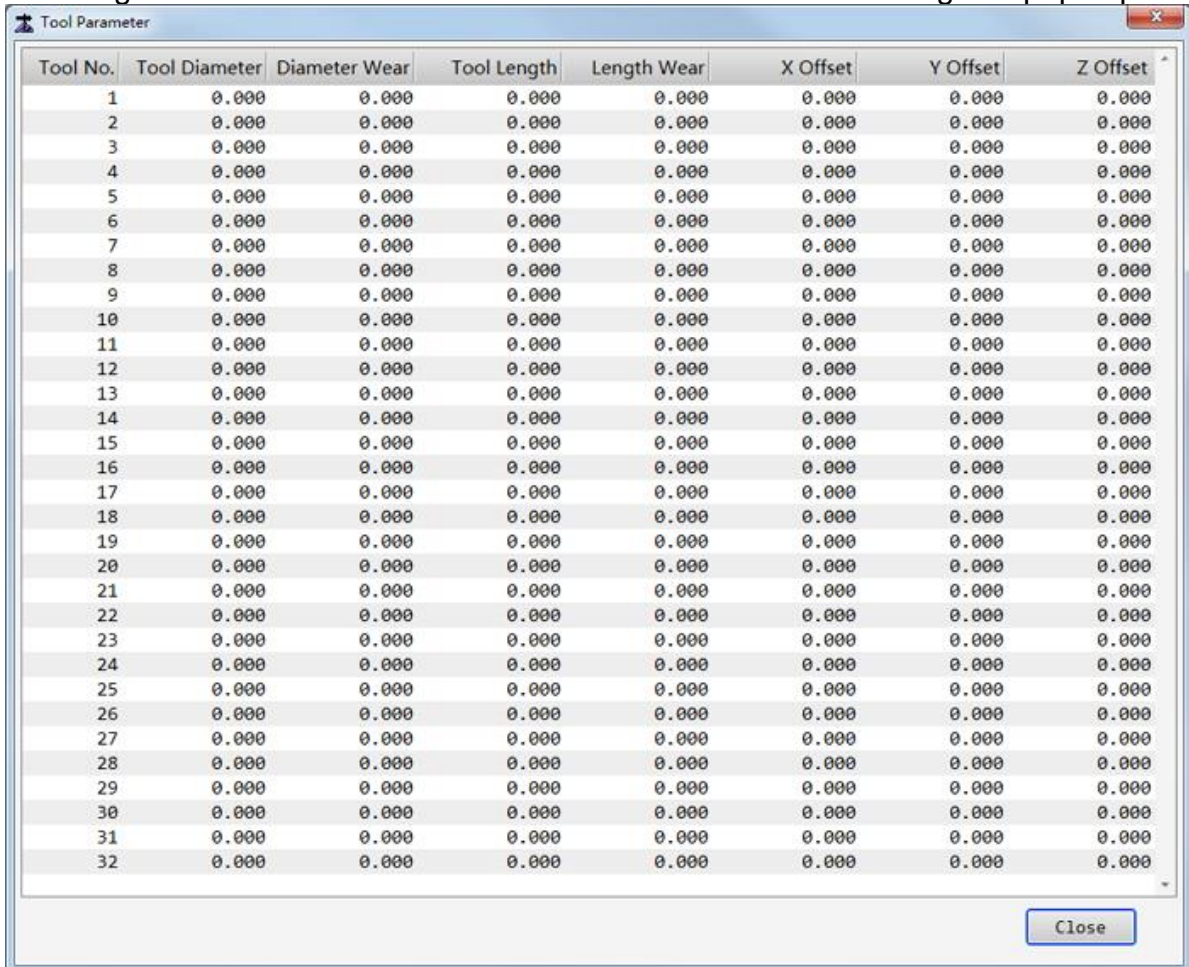| Tool No.	Tool Diameter	Diameter Wear	Tool Length	Length Wear	X Offset	Y Offset	Z Offset
1	0.000	0.000	0.000	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000	0.000	0.000	0.000
9	0.000	0.000	0.000	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000	0.000	0.000	0.000
11	0.000	0.000	0.000	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	0.000	0.000	0.000	0.000
13	0.000	0.000	0.000	0.000	0.000	0.000	0.000
14	0.000	0.000	0.000	0.000	0.000	0.000	0.000
15	0.000	0.000	0.000	0.000	0.000	0.000	0.000
16	0.000	0.000	0.000	0.000	0.000	0.000	0.000
17	0.000	0.000	0.000	0.000	0.000	0.000	0.000
18	0.000	0.000	0.000	0.000	0.000	0.000	0.000
19	0.000	0.000	0.000	0.000	0.000	0.000	0.000
20	0.000	0.000	0.000	0.000	0.000	0.000	0.000
21	0.000	0.000	0.000	0.000	0.000	0.000	0.000
22	0.000	0.000	0.000	0.000	0.000	0.000	0.000
23	0.000	0.000	0.000	0.000	0.000	0.000	0.000
24	0.000	0.000	0.000	0.000	0.000	0.000	0.000
25	0.000	0.000	0.000	0.000	0.000	0.000	0.000
26	0.000	0.000	0.000	0.000	0.000	0.000	0.000
27	0.000	0.000	0.000	0.000	0.000	0.000	0.000
28	0.000	0.000	0.000	0.000	0.000	0.000	0.000
29	0.000	0.000	0.000	0.000	0.000	0.000	0.000
30	0.000	0.000	0.000	0.000	0.000	0.000	0.000
31	0.000	0.000	0.000	0.000	0.000	0.000	0.000
32	0.000	0.000	0.000	0.000	0.000	0.000	0.000

Close

2. Set the following parameters for the target tool:
   o **Tool Diameter**: the diameter of the current tool.
   o **Diameter Wear**: the diameter wear of the current tool.
   o **Tool Length**: the length of the current tool.
   o **Length Wear**: the length wear of the current tool.
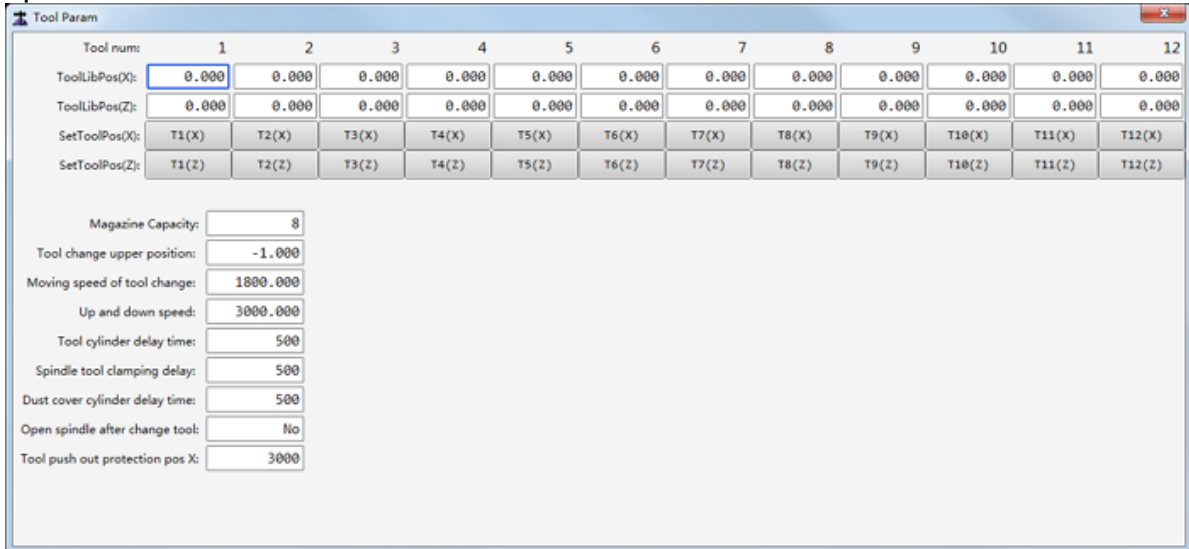   o **X / Y / Z Offset**: the offset of X-axis / Y-axis / Z-axis.

4.3.2 Set Parameters for the Tool Magazine

Before setting parameters for the tool magazine, make sure multi-cylinder configuration is not enabled.

Note: Please execute this operation in the help of the technician.

To set parameters for the tool magazine, do the following:

1. In the upper right corner of the main interface of **NcStudio** software, click the hamburger menu ☰ → **Tool Magazine Parameter**. **Tool Param** dialog box pops up:

2. In **Magazine Common Parameter** area, set the following parameters:
   o **Magazine capacity**: the total number of tools in tool magazine.
   o **Tool change upper position**: the upper position of Z-axis to change a tool.
   o **Speed in tool change**: the spindle speed during tool change.
   o **Speed of Z-axis**: the speed for Z-axis moving to the upper / lower position during tool change.
   o **Magazine cylinder delay**: the delay time to turn on the magazine cylinder.
   o **Spindle tool clamp delay**: the delay time to clamp / unclamp tool.
   o **Dustcover cylinder delay**: the delay time for the dustcover up and down.
   o **Open spindle after tool change**: whether to turn the spindle on after tool change.
   o **X safe position**: it is used to judge if the current machine coordinate of X-axis is less than this parameter.
3. To set the current position to the position of the tool magazine, modify the value of parameter **ToolLibPos(X/Z)**, and click the corresponding button of **SetToolPos(X/Z)**.

4.3.3 Set Parameters for Loading and Unloading

It is used to set parameters for loading and unloading, so as to improve machining accuracy and efficiency.

Note: Please execute this operation in the help of the technician.

Before setting parameters for loading and unloading, <u>make sure loading and unloading is enabled in **NcConfig** software.</u>

To set parameters for loading and unloading, do the following:

1. In the upper right corner of the main interface of **NcStudio** software, click the hamburger menu ▤ → **Load Unload Param**. **Load Unload Param** dialog box pops up:
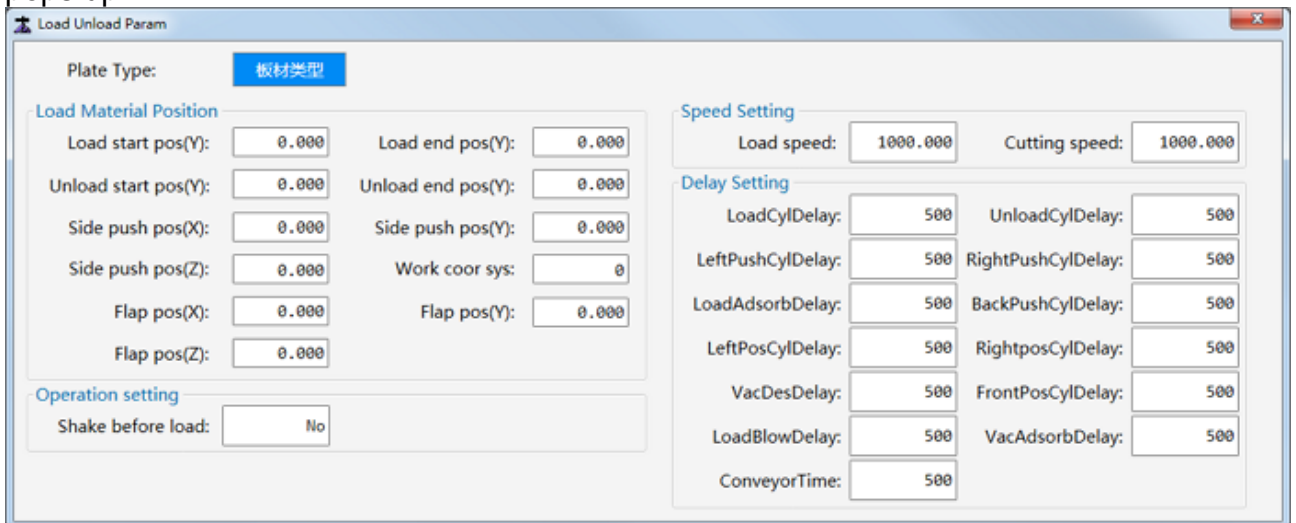
2. Double click 板材类型, and input the name for the plate type.
3. Set the following parameters according to your needs:
   - **Load start / end position**: the start / end position for loading material.
   - **Side push position**: the fixed position for side pushing.
   - **Unload start / end position**: the start / end position for unloading material.
   - **Vacuum adsorption / destroy delay**: the delay time for vacuum adsorption / destroy.
   - **Loading adsorption delay**: the delay time for loading adsorption.
   - **Front / Left / Right / Back position delay**: the delay time to turn on front / left / right / back positioning cylinder.
   - **Back / Left / Right / push delay**: the delay time to turn on back / left / right pushing cylinder.
   - **Load / unload delay**: the delay time to turn on loading / unloading cylinder.
   - **Load / unload speed**: the speed for loading and unloading.

4.3.4 Set a Printer

During machining, the system automatically recognizes pictures that are put in the same path with program files and whose naming rule is the same with program files and print them.

Before setting a printer, in **NcConfig** software, in **Other** area of **Component** page, check **LabelPrint**.

To set a printer, do the following:

1. In the upper right corner of the main interface of **NcStudio** software, click the hamburger menu [≡] → **Printer Setting**. **Printer** dialog box pops up:
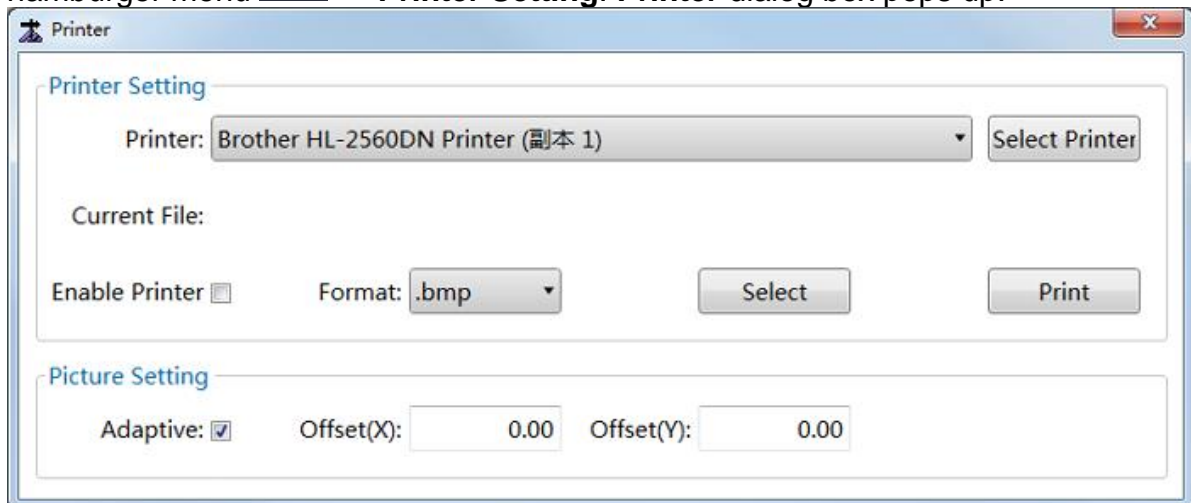

2. In **Printer Setting** area, do the following:
   a. To select the target printer, click the dropdown box of **Printer** and select the target printer.
      If you need to select the target printer to the default printer, please click **Select Printer**.
   b. **Optional:** To make the system automatically print pictures without editing, check **Enable Printer**.
   c. In the dropdown box of **Format**, select the picture format.
      The supported formats include BMP, JPG, PNG, GIF and TIF.
   d. To select the storage path for printed pictures, click **Select** and select the target path.
3. In **Picture Setting** area, do the following:
   a. **Optional**: To make the printed pictures adapt to the paper size, check **Adaptive**.
   b. To set X-axis coordinate of the upper left corner of the picture corresponding to the label paper, click input box **Offset(X)**, and set a value.
      Default value: 0.
   c. To set X-axis coordinate of the upper left corner of the picture corresponding to the label paper, click input box **Offset(Y)**, and set a value.
      Default value: 0.
4. Click **Print**:
   o When a program file is loaded, if a picture exists in the same path with the program file, the system prints these pictures. Otherwise, printing action will not occur.
   o When no program file is loaded, the system prints the selected picture.

4.3.5 Sort Program Files in a Machining Task

It is used to sort program files in the same machining task before start machining the task.

The naming rule of program files in the same machining task is as follows:

1. Board type: any characters, texts without limitation of the number of words.
2. Number: the digits should be the same and continuous.
3. Front/Back identifier: any characters, numbers, texts without limitation of the number of words.
    o If a front identifier shows in the file name, the front of the board will be machined.
    o If a back identifier shows in the file name, the back of the board will be machined.

    Note: Please set the front/back identifier in the prefix or suffix of the file name.

To sort program files in a machining task, do the following:

1. To switch to **Multi-task Manage** window, in the function window area, click **Multi-task Manage**:
2. Change **Multi-task Processing** switch to **ON** status.
3. Click **Permutation Setup**. **Permutation Setup** dialog box pops up:
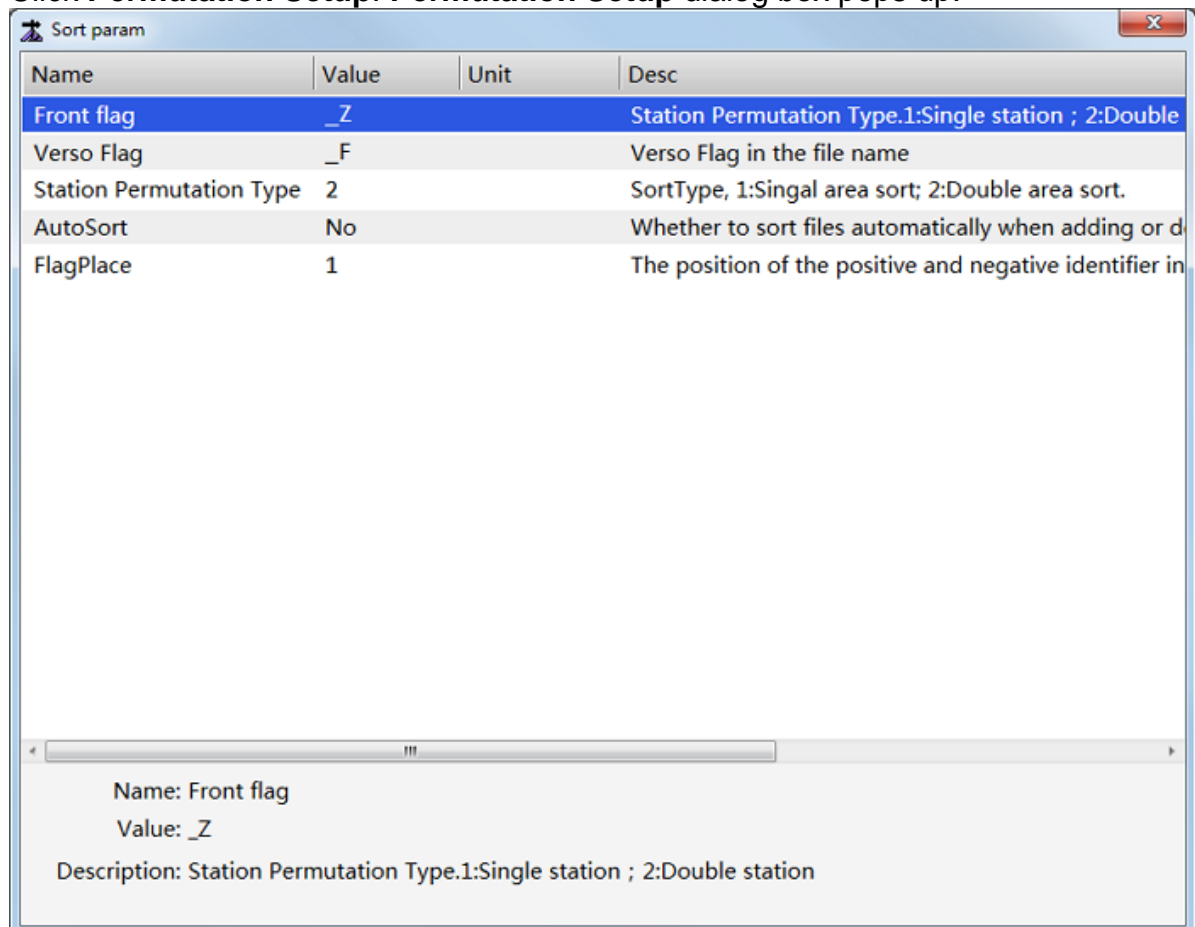
4. To set the front / back identifier of the file name, set parameters **Front Flag** and **Verso Flag**.
   The system will firstly machine the program file with a back identifier, then the program file with a front identifier, and lastly the program file with no identifier.
5. To set the position of the identifier in the file name, set parameter **Flag Place**:
   - o 1: the identifier is in the suffix of the file name.
   - o 2: the identifier is in the prefix of the file name.
6. **Optional**: To enable double stations for machining, set parameter **Station Permutation Type** to **2**.
   The system uses single station for machining by default.
7. Modify the value of parameter **Auto Sort**:
   - o When a program file is added into the machining task:
     - ▪ When the parameter is set to **Yes**, the system firstly machines the program file with a back identifier, then the program file with a front identifier and lastly the program file with no identifier.
     - ▪ When the parameter is set to **No**, the system lastly machines the added file.
   - o When a program file is deleted in the machining task:
     - ▪ When the parameter is set to **Yes**, the system lastly machines the other file (002_F in the following example) that is in the same group with the deleted file.
       Example
       When the machining order is 001_B(back identifier) → 001_F(front identifier) → 002_B(back identifier) → 002_F(front identifier) → 003_F(front identifier) → 006, and 002_B is deleted, the machining order turns into 001_B → 001_F → 003_F → 006 → 002_F.
     - ▪ When the parameter is set to **No**, the machining order remains unchanged.
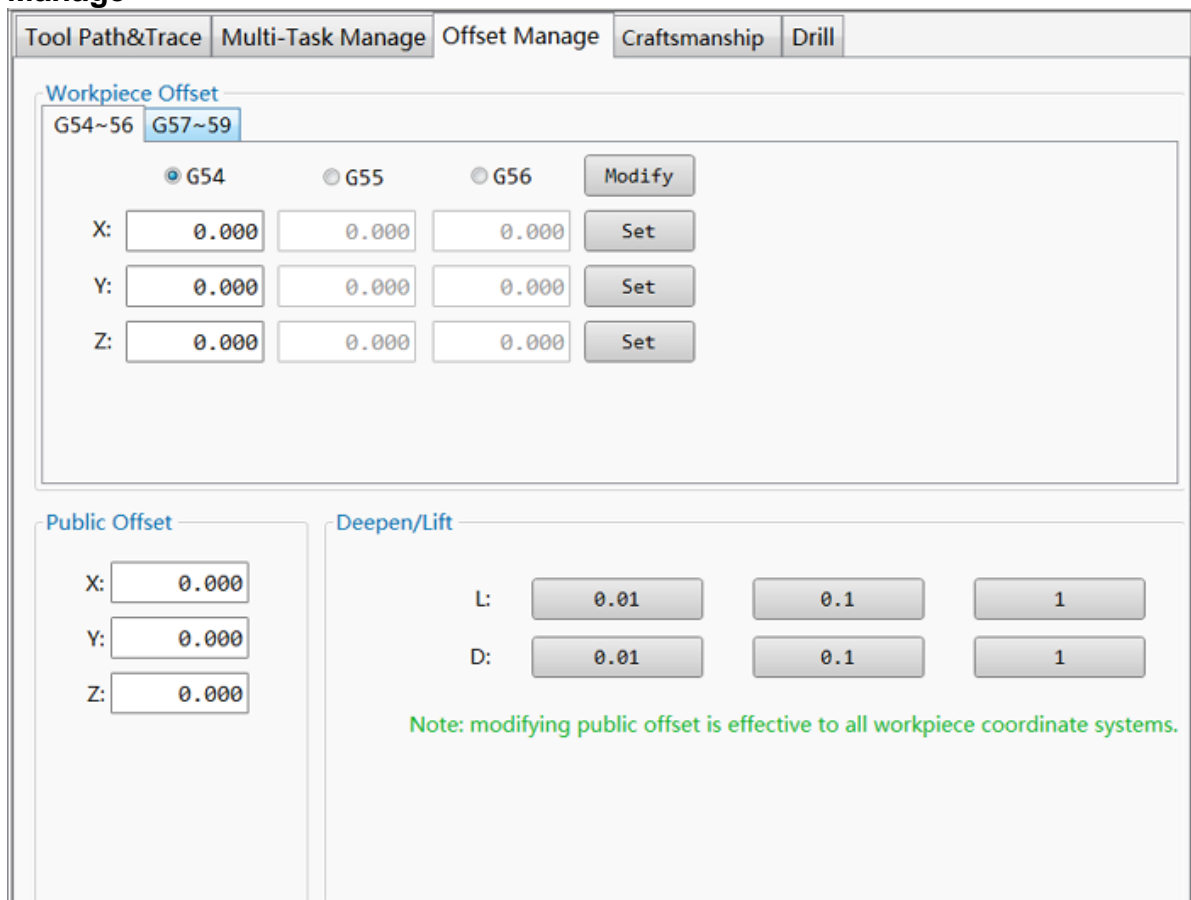
4.3.6 Manage Offsets

Offsets include the following:

- **Workpiece offset**: the distance between the workpiece origin and the machine origin.
- **Public offset**: the distance of the X-axis, Y-axis and Z-axis relative to the machine origin for all workpiece coordinate systems.
- **Tool offset**: the distance between the tool and standard tool. The relationship between workpiece coordinate and offsets is as follows:

Workpiece coordinate = Machine coordinate - Workpiece offset - Public offset - Tool offset
To manage offsets, do the following:

1. To switch to **Offset Manage** window, in the function window area, click **Offset Manage**

2. To set the workpiece offset, do the following:
   a. In **Workpiece Offset** area, select the target workpiece coordinate system from G54 to G56, and click **Modify**.
   b. To set the current position to the workpiece offset, input the workpiece offset of each axis, and click **Set**.
3. To set the public offset, in **Public Offset** area, input the public offset of each axis.
4. **Optional:** To adjust Z-axis public offset, do one of the following:
   o To lift Z-axis **0.01mm** / **0.1mm** / **1mm**, click **0.01** / **0.1** / **1** after **L** to.
   o To deepen Z-axis **0.01mm** / **0.1mm** / **1mm**, click **0.01** / **0.1** / **1** after **D**.
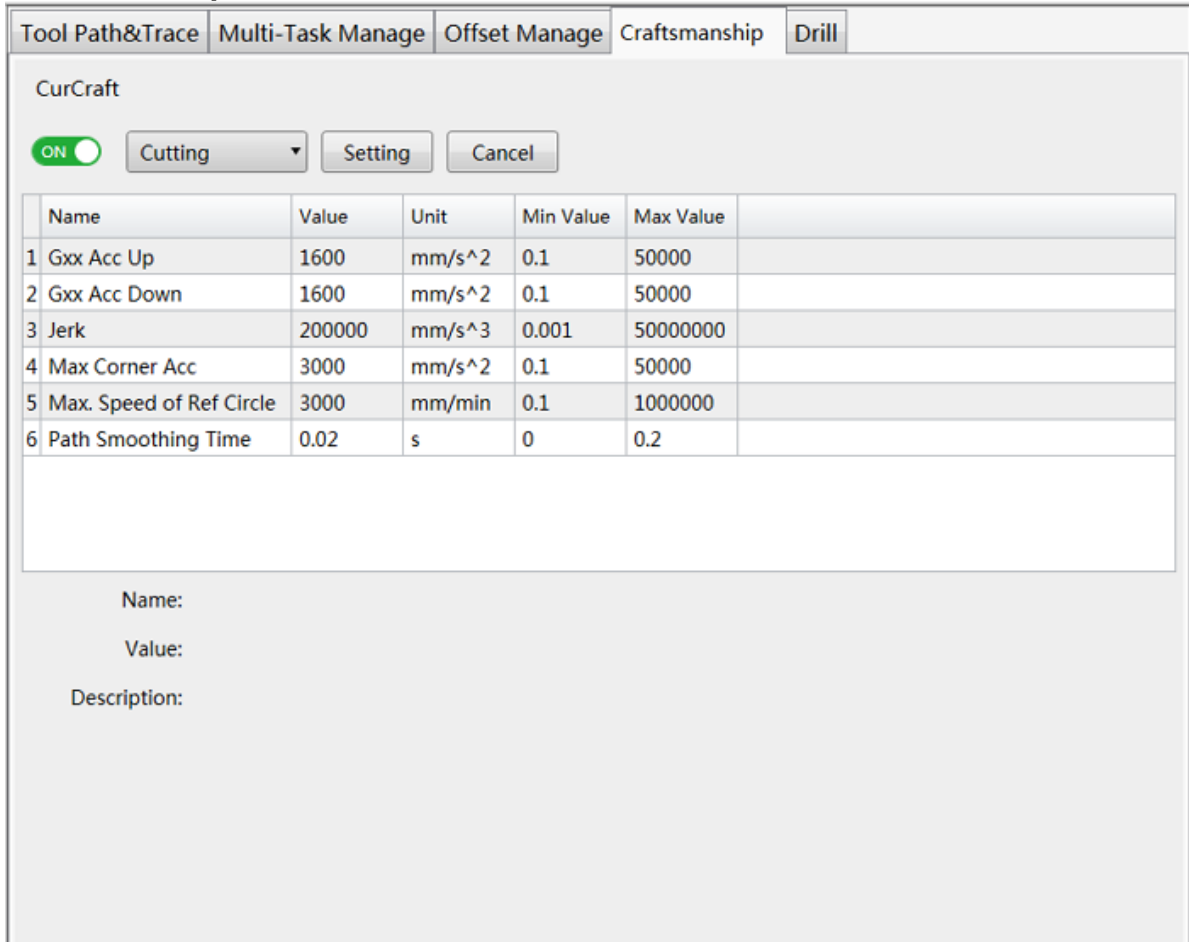
4.3.7 Set Technics

It is used to set parameters for different technic types, so as to improve machining accuracy and efficiency.

Technic types include the following:

- **Cutting**: used to cut boards.
- **Wood**: Carve patterns in the boards.

To set technics, do the following:

1. To switch to **Craftsmanship** window, in the function window area, click **Craftsmanship**:

	Name	Value	Unit	Min Value	Max Value
1	Gxx Acc Up	1600	mm/s^2	0.1	50000
2	Gxx Acc Down	1600	mm/s^2	0.1	50000
3	Jerk	200000	mm/s^3	0.001	50000000
4	Max Corner Acc	3000	mm/s^2	0.1	50000
5	Max. Speed of Ref Circle	3000	mm/min	0.1	1000000
6	Path Smoothing Time	0.02	s	0	0.2

Tool Path&Trace | Multi-Task Manage | Offset Manage | Craftsmanship | Drill

CurCraft

ON | Cutting | Setting | Cancel

Name:

Value:

Description:

2. To modify technic parameters, change **Turn on** switch to **ON** status.
   Otherwise, you are not allowed to modify technic parameters.
3. Select the technic type, and click **Setting**.
   If you need to cancel this operation, please click **Cancel**.
4. In the **Value** column, double click the table cell, and modify values of the following parameters:
   - **Gxx Acc Up**: the acceleration in accelerating phrase during machining.
   - **Gxx Acc Down**: the deceleration in decelerating phrase during machining.
   - **Jerk**: the change rate of acceleration.
   - **Max Corner Acc**: the maximum feedrate at corners. It is suggested to set it 1~2 times of acceleration of a single axis.
   - **Max Speed of Ref Circle**: the maximum feedrate when machining 10mm-diameter reference circle.
   - **Path Smoothing Time**: the longer the time, the smoother the surface of the workpiece. But some of the details may be weakened.

5   FAQs

It is used to clear alarms so as to establish good communication between WEIHONG **NcStudio CNC System for Cutting Machines** and the servo drive.

The types of alarms that need to be cleared include the following:

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

5.1   Clear the Alarm of Terminal Board Disconnection

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

1. Make sure the wiring of terminal boards is connected correctly and securely.
   If it is not connected correctly and securely, correct and tighten the wiring.
2. Make sure the drive program is correctly installed.
   If it is not correctly installed, install the drive program again.
3. Make sure the terminal boards are not damaged.
   If it is damaged, replace the terminal board.

5.2   Clear the E-stop Alarm

To clear the E-stop alarm, check if E-stop button on the panel is pressed:

- If it is, release the button.
- If it is not, modify the polarity of **E-stop** port in **NcTune** software.

5.3   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.

5.4   Clear the Servo Drive Alarm

Before clearing the servo drive alarm, make sure no alarm in the servo drive.

To clear the servo drive alarm, do the following:

1. Make sure the wiring of the servo drive is connected correctly and securely.
   If it is not connected correctly and securely, correct and tighten the wiring.
2. Make sure the polarity of **Servo Alarm** port is **NC**.
   If it is not, modify the port polarity to **NC**.
3. **Optional:** Make sure the setting of parameter **Drive Station Address** matches with the drive station address of the machine tool.
   If it does not, modify the parameter setting according to the drive station address of the machine tool and restart the software to make modification effective.

5.5   Clear the Alarm of Port Polarity Error

When the switch type of port polarity does not match with that of the machine tool, related ports will send alarms.

To clear the alarm of port polarity error, modify the polarity of related ports in **NcTune** software.

6   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.

6.1   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.

6.2   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.

6.3   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