NC60A Multi-Process Control System Manufacturers' Manual

Version: 2018.10 1st Version
Author: Document Department

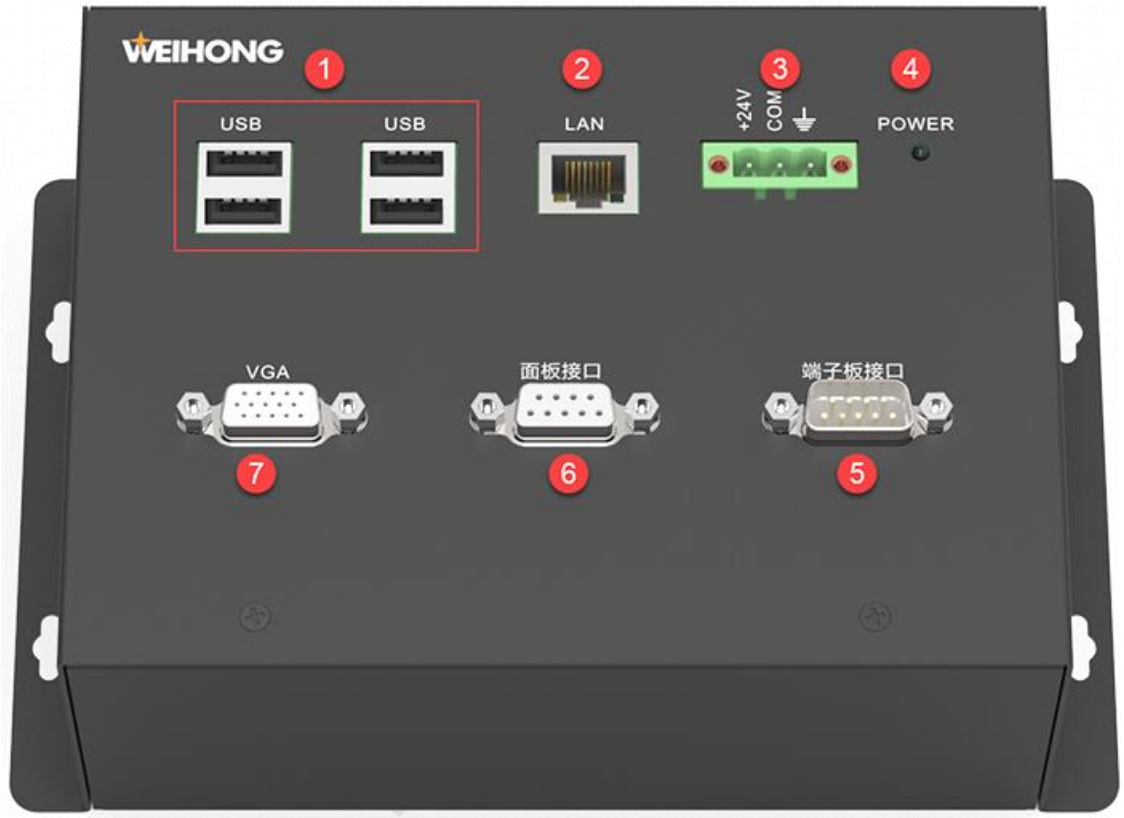
This manual is intended for manufacturers of machine tools. It mainly includes:

1. NC60A Industrial Personal Computer

NC60A Controller is a new member of WEIHONG industrial personal computers (PCs). Compared to other industrial PCs, it is more cost-effective and compact.

1.1. Layout

The interface layout of NC60A is as follows:



1. USB
2. LAN
3. 24V DC power
4. LED indicator
5. Lambda terminal board
6. Panel
7. VGA, display screen

1.1.1. LED Indicator

It is used to indicate the current status of the system.

- 2Hz flicker frequency shows good communication.
- 0.33Hz flicker frequency shows power disconnection.
- 10Hz flicker frequency shows exception.

1.1.2. Interface for Panel

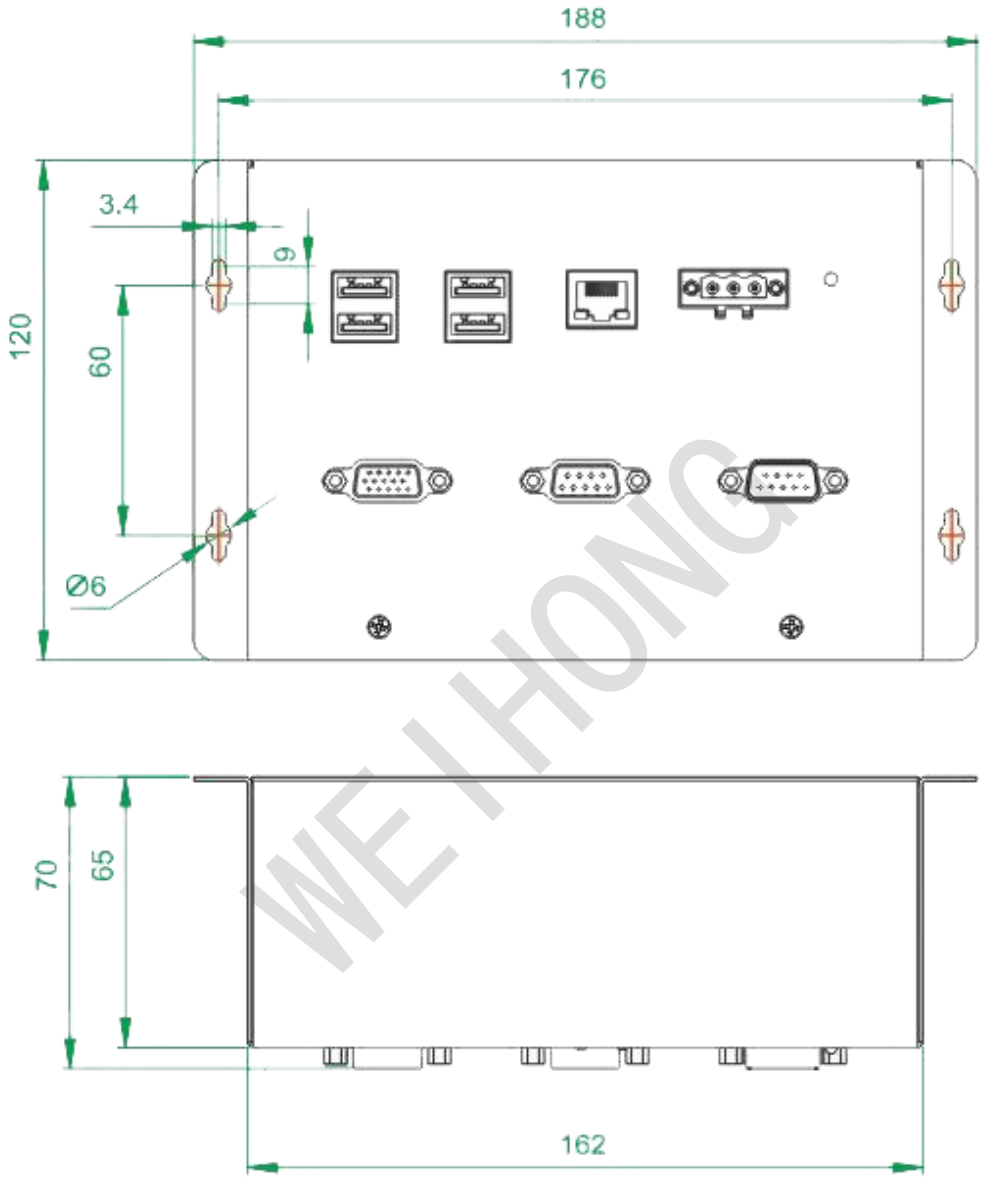
It is used to connect to operating panels.

1.1.3. Interface for Lambda Terminal Board

It is used to connect to Lambda terminal board.

1.2. Mounting Dimension

Mounting dimension of **NC60A Controller** is as follows:



2. Software Interface Overview

NC60A Multi-Process Control System is a CNC system solution for Weihong's multi-process cutting machine in woodworking industry. With simple and convenient operation and customized functions, the system can match with different machine models.

According to the operational role and usage scenarios, the system interface can be divided into the following:

- Operator interface

  Used for machining.

  It includes frequently used functions in machining for operators.

  It is the default interface.

- Technician interface

  Used for debugging the machine tool.

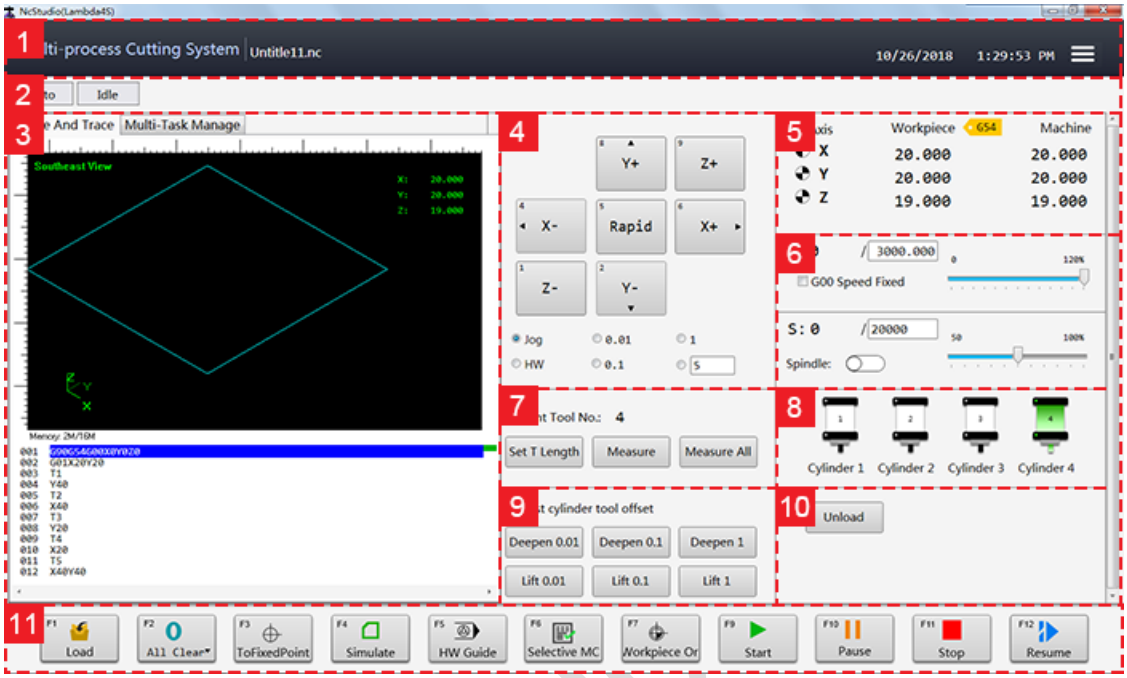
  It includes rich functions and machine parameters for technicians.

To switch between two interfaces, do one of the following:

- Under operator interface, to switch to the technician interface, click  → **Technician Interface** in the upper right corner.
- Under technician interface, to switch to the operator interface, do one of the following:
  - Click  in the upper right corner.
  - Click **System** → **Switch to Operator Interface**.

2.1. Operator Interface

Operator interface is shown as follows:



1. Title bar
2. CNC status bar
3. Function windows
4. Axis direction and mode selection area
5. Axis coordinates display area
6. Speed control area
7. Tool setting area
8. Cylinder control area
9. Area for adjusting cylinder tool offset
10. Loading and unloading control area
11. Operational buttons

2.1.1. Title Bar

This bar consists of the following:

- Name of the control system
- Current date and time
- Burger button 

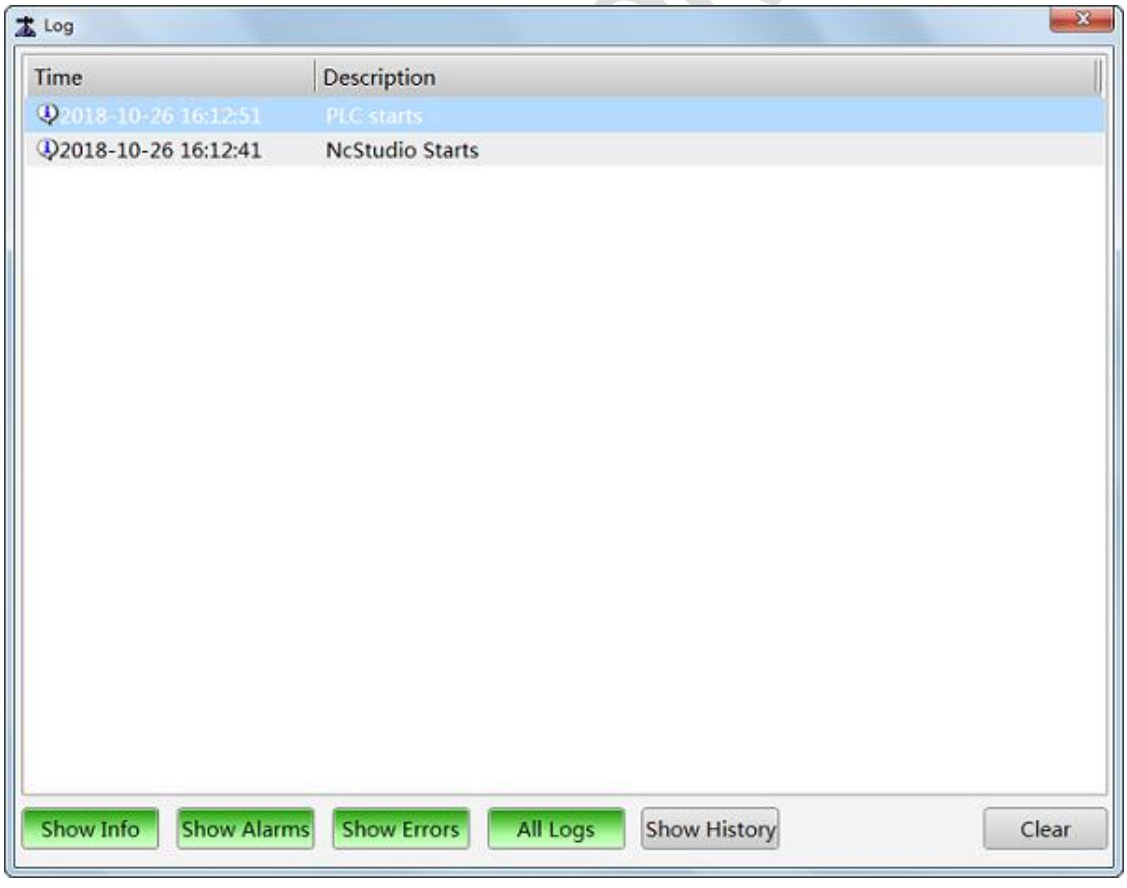
  Used to do the following:

  - Execute homing.
  - Check tool parameters.
  - Switch to technician interface.

2.1.2. CNC Status Bar

This bar shows current system mode and status, and prompt or alarm message.

Click the blank area. **Log** dialog box pops up:



In the dialog box, you can check different types of system logs that are good for troubleshooting.

2.1.3. Function Windows

Function windows include the following:

- **Ncfile and Trace** window

  In this window, you can see the following:

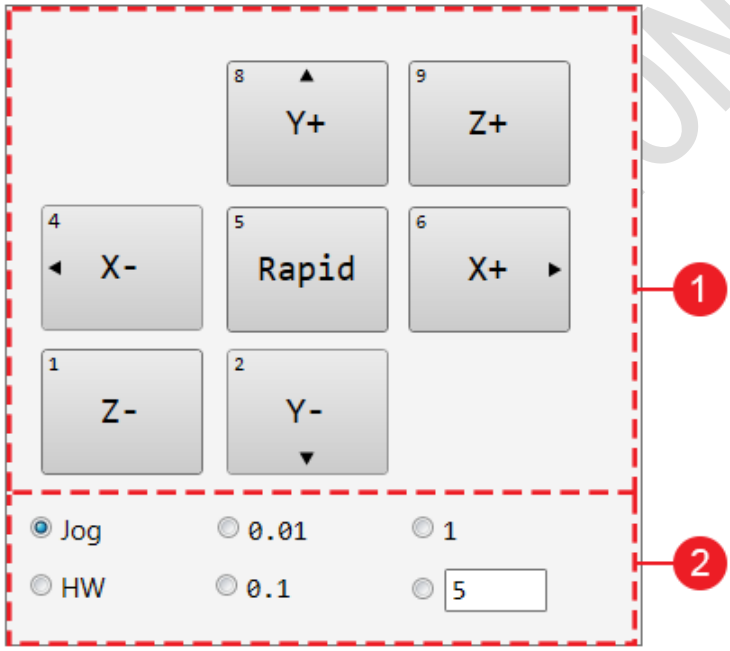
  - Tool path in real-time during machining or simulation so as to ensure the proper implementation of program file.
  - Related program file in details.

- **Multi-Task Manage** window

  In this window, you can manage machining tasks and check information of each task.

2.1.4. Axis Direction and Mode Selection Area

This area consists of the following:



1. Axis direction buttons:

   Used to move each axis towards positive or negative direction:

   - Press **X-** / **X+** / **Y-** / **Y+** / **Z-** / **Z+**. The machine moves at jogging speed.
   - Press **Rapid** and **X-** / **X+** / **Y-** / **Y+** / **Z-** / **Z+**. The machine moves at rapid jogging speed.

2. Mode buttons:

   Used to switch to the following modes:

   - Jog

     Press an axis direction button. The machine keeps running until you release the button.

   - HW

     The machine is controlled by the handwheel.

   - Step

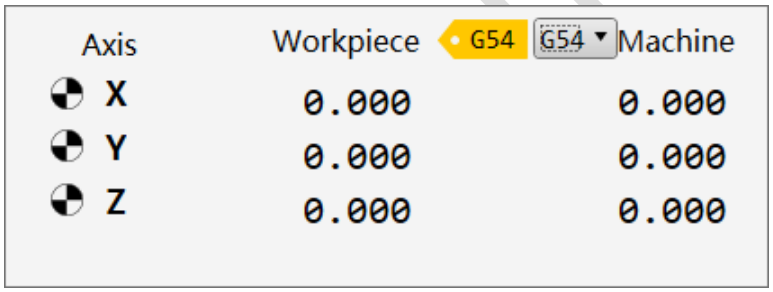
     Click an axis direction button. The machine moves 0.01 mm(inch), 0.1 mm(inch), 1 mm(inch) or customized step size.

     The default customized step size is 5mm(inch) and the value should not be too large to avoid damage due to misoperation.

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

2.1.5. Axis Coordinates Display Area

This area consists of the following:

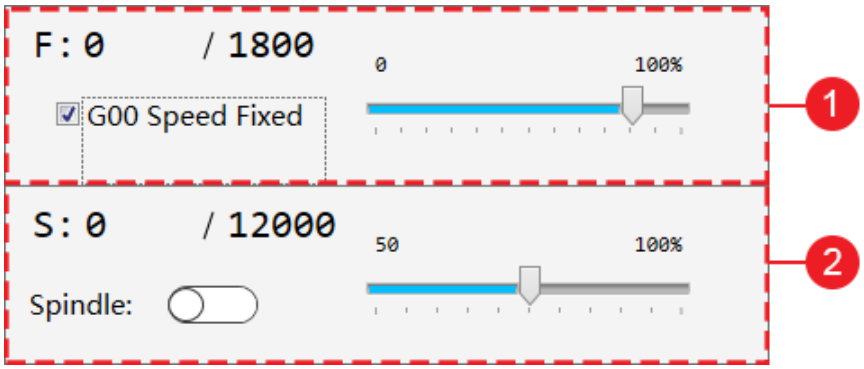


- Workpiece coordinates
- Machine coordinates

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

2.1.6. Speed Control Area

This area consists of the following:



1. Feed setting area

   Used to do the following:

   - Set feedrate override.
   - Modify setting speed.
   - Check actual speed.

   Relationship among them is as follows:

   ```
   Actual Speed = Setting Speed * Current Feedrate Override
   ```

2. Spindle setting area

   Used to do the following:

   - Set spindle override.
   - Modify spindle setting speed.
   - Check actual spindle speed.

   Relationship among them is as follows:

   ```
   Actual Spindle Speed = Setting Spindle Speed * Current Spindle Over
   ride
   ```

2.1.7. Tool Setting Area

In this area, you can check the current tool and do the following:

- Click **Set T Length** to modify Z-axis offset of current tool.
- Click **Measure** to measure length of current tool and set it to tool offset.
- Click **Measure All** to measure length of all tools and set it to tool offset.

2.1.8. Cylinder Control Area

In this area, you can control cylinders by clicking related cylinder to enable it.

The number of cylinder differs according to the cylinder selection in **Function Configuration** dialog box.

If no cylinder is selected in the dialog box, this area will be hidden.

See Customize Function Configuration for details of function configuration.

2.1.9. Area for Adjusting Cylinder Tool Offset

In this area, you can adjust cylinder tool offset.

If no cylinder is selected in **Function Configuration** dialog box, this area will be hidden.

See Customize Function Configuration for details of function configuration.

2.1.10. Loading and Unloading Control Area

In this area, you can enable loading and unloading by clicking related buttons.

The area differs according to the selection of loading or unloading in **Function Configuration** dialog box.

If loading or unloading is not selected in the dialog box, this area will be hidden.

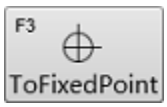
See Customize Function Configuration for details of function configuration.

2.1.11. Operational Buttons

These buttons are used to execute corresponding operation:

Used to load a program file.

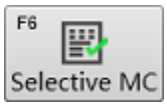
Used to clear a single axis or all axes.



Used to return to the set position of the fixed point.

Used to execute simulation to check the tool path in **Ncfile and Trace** window in real-time.

Used to enable handwheel to control the machine tool.



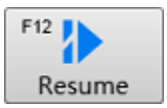
Used to select related lines of the program file to be machined.

Used to return to the workpiece origin.

Used to start machining.

Used to pause machining.

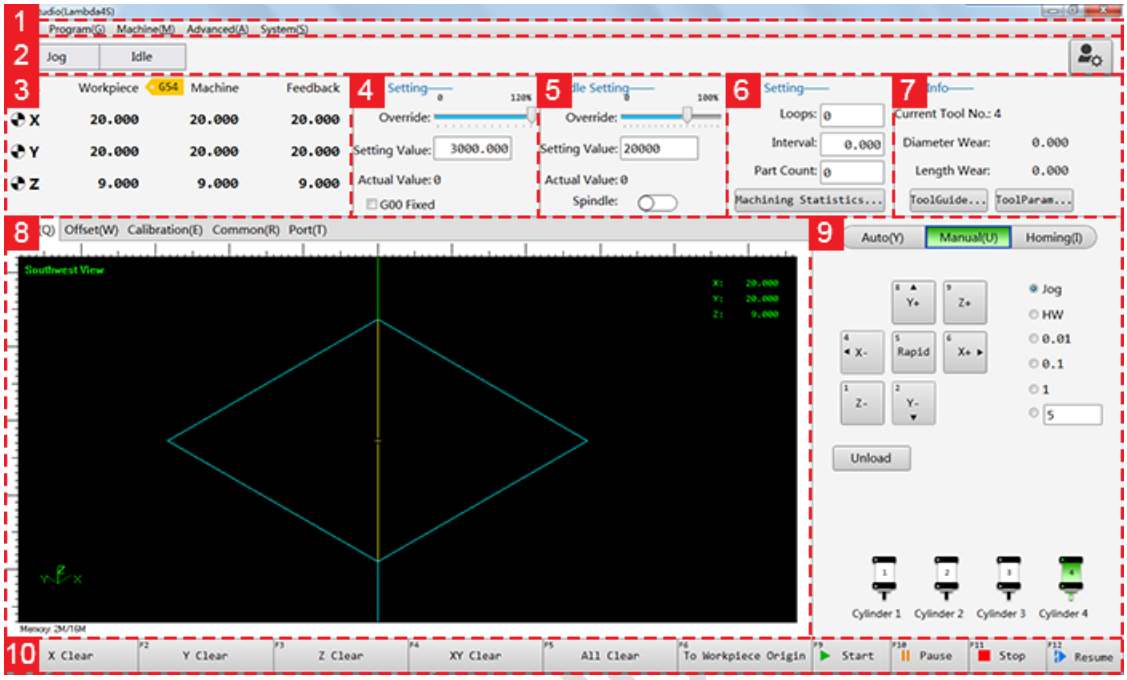
Used to stop machining.



Used to resume machining from the exact interrupted position when power interruption or e-stop occurs and the workpiece origin is secured.

2.2. Technician Interface

The interface diagram is as follows:



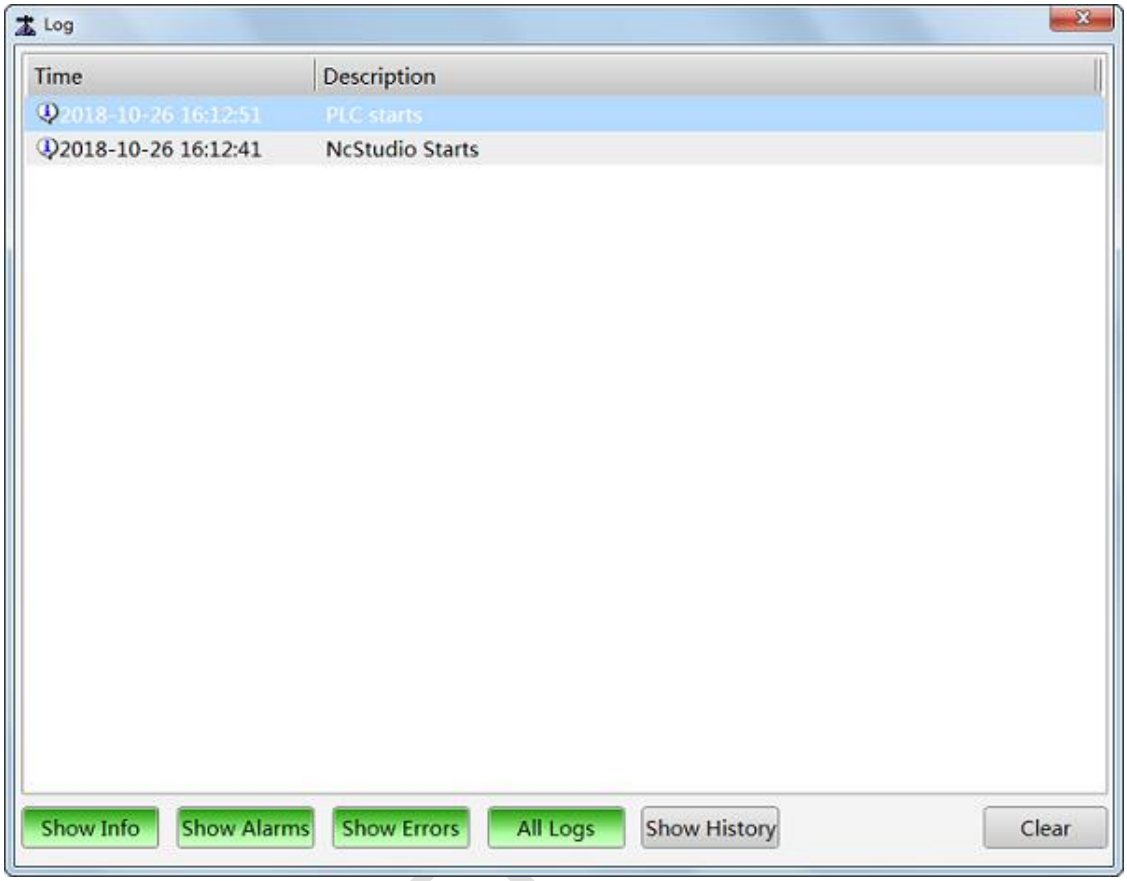
1. Menu bar
2. CNC status bar
3. Axis coordinates display area
4. Feed setting area
5. Spindle setting area
6. Cycle setting area
7. Tool information
8. Function windows
9. Mode control bar
10. Operational buttons

2.2.1. CNC Status Bar

This bar shows the following:

- Current system mode
- Current system status
- Prompt or alarm message
- Manufacturer permission button 

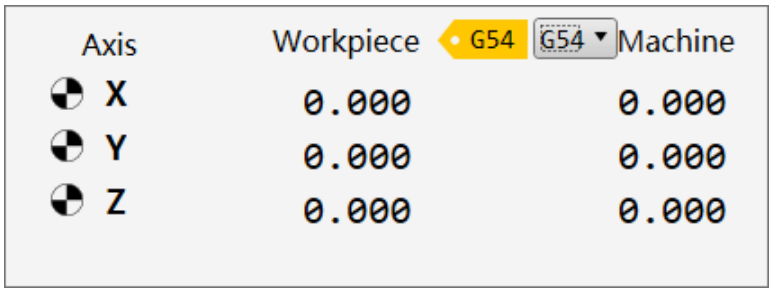
Click the blank area. **Log** dialog box pops up:



In the dialog box, you can check different types of system logs that are good for troubleshooting.

2.2.2. Axis Coordinates Display Area

This area consists of the following:



- Workpiece coordinates
- Machine coordinates

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

2.2.3. Feed Setting Area

In this area, you can do the following:

- Set feedrate override.
- Modify setting speed.
- Check actual speed.

Relationship among them is as follows:

```
Actual Speed = Setting Speed * Current Feedrate Override
```

2.2.4. Spindle Setting Area

In this area, you can do the following:

- Set spindle override.
- Modify spindle setting speed.
- Check actual spindle speed.

Relationship among them is as follows:

```
Actual Spindle Speed = Setting Spindle Speed * Current Spindle Override
```

2.2.5. Cycle Setting Area

In this area, you can do the following:

- Set cycle time.
- Set cycle interval.
- Reset workpiece count.
- Check machining statistics.

2.2.6. Tool Information

In this area, you can do the following:

- Check current tool number.
- Check diameter wear of current tool.
- Check length wear of current tool.
- Open **ToolGuide** dialog box and set parameters related to tool magazine.
- Set tool compensation parameters.

2.2.7. Function Windows

Function windows include the following:

- **Track** window:

  In this window, you can see the following:

  - Tool path in real-time during machining or simulation so as to ensure the proper implementation of program file.
  - Related program file in details.

- **Offset** window

  In this window, you can set the following:

  - Workpiece offset
  - Public offset

  See Set Offset for details.

- **Calibration** window

  In this window, you can do the following:

  - Change the active tool and its related parameters.
  - Set the position of fixed tool sensor.
  - Select tool calibration mode.

  See Calibrate the Tool for details.

- **Common** window

  In this window, you can do the following:

  1. Check common parameters.
  2. Modify values of common parameters.
  3. Customize common parameters.

  See Customize Common Parameters for details.

- **Port** window

  In this window, you can do the following:

  - Check port polarity
  - Conduct test.
  - Modify port polarity.
  - Set filter.

  See Execute Port Related Operations for details.

2.2.8. Mode Control Bar

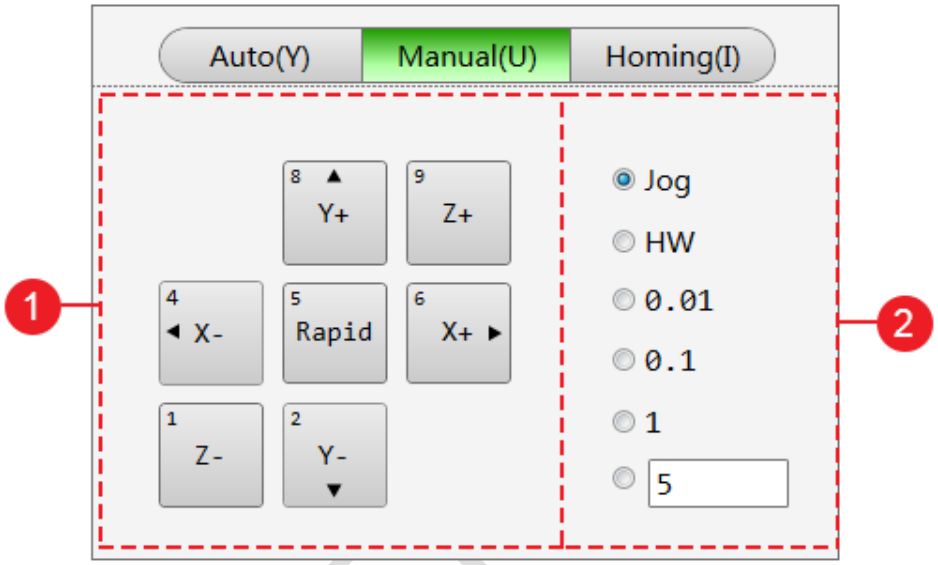
It includes the following modes:

- **Auto**

  Under this mode, it shows the loaded program file in details.

- **Manual**

  Under this mode, it shows the following:



1. Axis direction buttons

   Used to move each axis towards positive or negative direction:

   - Press **X-** / **X+** / **Y-** / **Y+** / **Z-** / **Z+**. The machine moves at jogging speed.
   - Press **Rapid** and **X-** / **X+** / **Y-** / **Y+** / **Z-** / **Z+**. The machine moves at rapid jogging speed.

2. Mode selection buttons

   Used to switch to the following modes:

   - Jog

     Press an axis direction button. The machine keeps running until you release the button.

   - HW

     The machine is controlled by the handwheel.

- Step

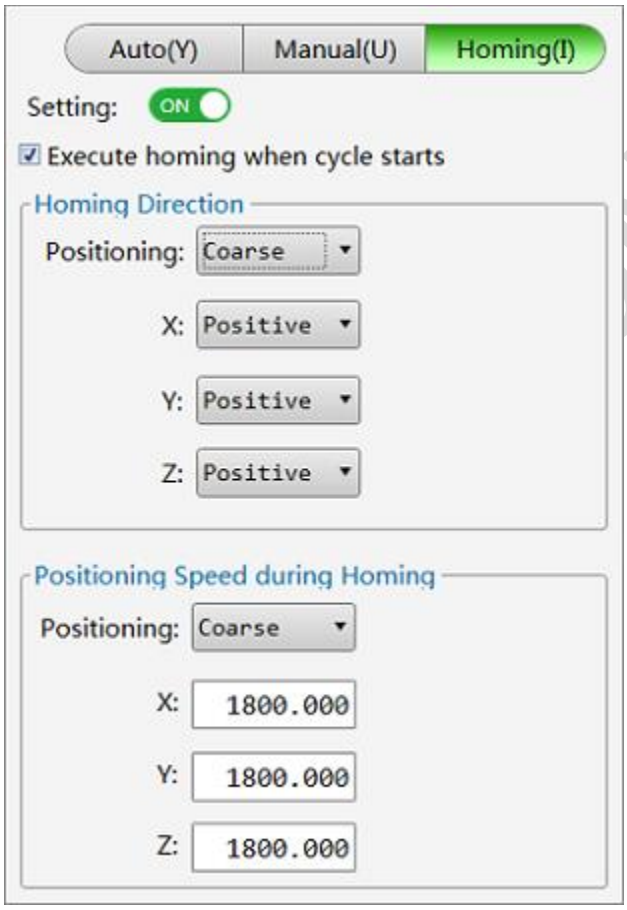
  Click an axis direction button. The machine moves 0.01 mm(inch), 0.1 mm(inch), 1 mm(inch) or customized step size.

  The default customized step size is 5mm(inch) and the value should not be too large to avoid damage due to misoperation.

  Note: Please do not click frequently for the system needs a certain time to execute the command.

- **Homing**

  Under this mode, it includes the following operations related to homing (returning to the machine origin):



  - Setting homing direction
  - Setting positioning speed during homing

2.2.9. Operational Buttons

It shows the related operations under different modes.

3. Getting Start

This section mainly introduces the following operations for operators to quickly know about how to use **NC60A Multi-Process Control System**:

- Return to the Machine Origin
- Load a Program File
- Set the Workpiece Origin
- Run the Program
- Execute Multi-tasking

All operations are done under **Operator Interface**.

3.1. Return to the Machine Origin

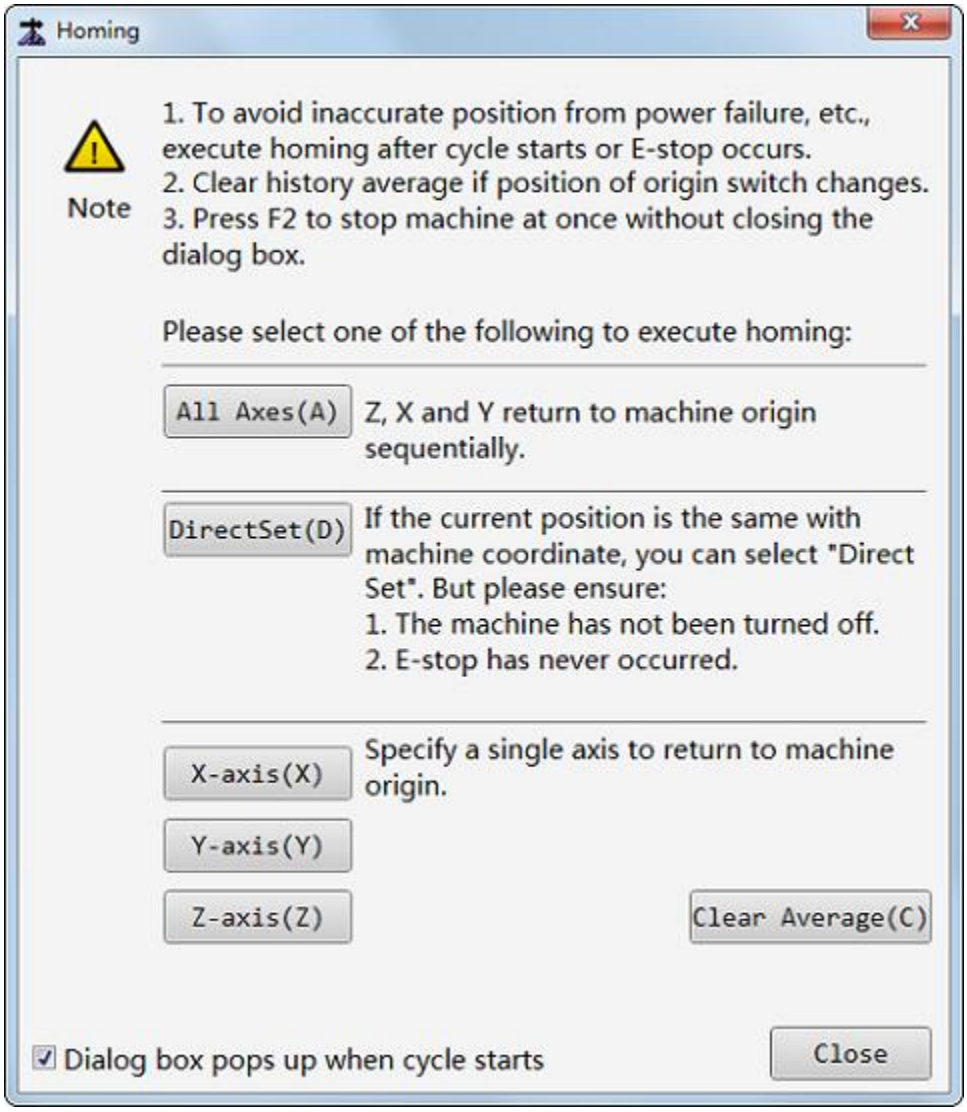
This operation is used to adjust coordinate position before machining.

Machine coordinate system(MCS) is unique for the machine tool. Its origin is called machine origin or machine zero. It is fixed and defined before delivery of the machine tool.

After software starts, returning to the machine origin is required before machining.

To return to the machine origin, do the following:

1. Click → **Homing**. **Homing** dialog box pops up:



   If **Dialog box pops up when software starts** is checked, the dialog box pops up automatically.

2. **Optional:** If the position of the machine origin changes, click **Clear Average** to clear historical averages.

   Otherwise, the prompt *Distance between coarse and fine positioning signals exceeds the allowable range compared to the historical average. Failed to return to the machine origin.* pops up.

3. Choose one of the following to return to the machine origin:

   – All axes

     All axes return to the machine origin in order of Z-axis and XY axes.

   – Direct set

     Directly set the current position as the machine origin.

     With the method, you MUST ensure:

     - The position is the same with the machine coordinate.
     - The machine has never been turned off.
     - E-stop has never occurred.

   – Specify an axis

     Specify an axis to return to the machine origin.

     For safety, it is suggested to firstly return Z-axis, then X-axis and Y-axis.

After setting, the sigh appears before related axis in axis coordinates display area.

3.2. Load a Program File

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

Before loading a program file, prepare a machining file.

To load a program file, click 

and select a program file.

After it is done, content of the program file shows in **Ncfile and Trace** window.

3.3. Set the Workpiece Origin

The workpiece origin is the origin of each axis in the program file. This operation is used to clear current workpiece coordinate of each axis and set workpiece offset to corresponding machine coordinate of each axis. As a result, the workpiece origin turns to zero.

To set the workpiece origin, do the following:

1. Manually move the machine tool to the position where you would like to set as the workpiece origin.

2. Click the lower part of the button 

and select related axis to do clearing:

   All Clear: set the current machine coordinates of all axes as the workpeice origin. **X Clear:** set the current machine coordinate of X axis as the workpeice origin. **Y Clear:** set the current machine coordinate of Y axs as the workpeice origin. **Z Clear:** set the current machine coordinate of Z axis as the workpeice origin. **XY Clear:** set the current machine coordinates of X-axis and Y-axes as the workpeice origin.

3. To clear current workpiece coordinate, click the upper part of the button 

and click **OK**.

After it is done, the workpiece origin turns to zero.

3.4. Run the Program

NC60A Multi-Process Control System supports the following three ways to run the program:

- Execute auto run

  Used to automatically run the program from beginning to the end.

- Execute selective machining

  Used to realize optional skip of the program.

- Enable handwheel guide

  Used to control the execution speed manually during auto run so as to avoid tool damage form wrongly loaded program or inappropriate toolpath.

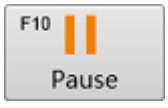
3.4.1. Execute Auto Run

To execute auto run, press 

.

The system automatically executes the program until end of block or any intentional interruption (e.g. Stop or Pause button is pressed).

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

- Click 

to pause machining.

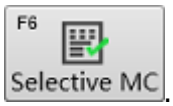
- Click 

to stop machining.
- Click 

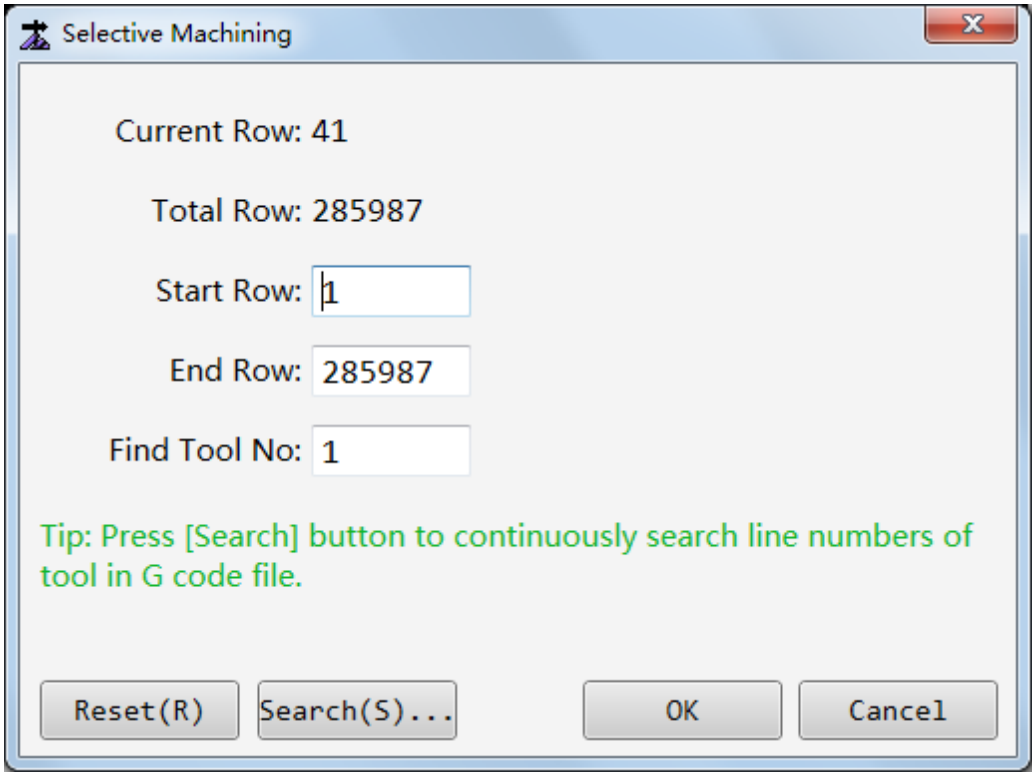
to resume machining from the exact interrupted position when power interruption or e-stop occurs and the workpiece origin is secured.

3.4.2. Execute Selective Machining

To execute selective machining, do the following:

1. Click 

. **Selective Machining** dialog box pops up:



2. To select machining rows, do one of the following:
   - Directly input row numbers in input boxes of **Start Row** and **End Row**.

– Input a tool number in **Find Tool No.** input box and click **Search**.

The system does the following:

- Search from the start row to the end row.
- Find the row where the set tool number appears at the first time.
- Automatically fill the found row number into **Start Row** input box.

If the tool appears several times in different rows, click **Search** several times to found the rows.

3. Click **OK**.

After setting, click 

. The system starts machining from the set start row to the set end row.

3.4.3. Enable Handwheel Guide

Before using handwheel guide, connect the handwheel to the system.

To use handwheel guide, press 

and turn the handwheel to control machining.

3.5. Execute Multi-tasking

Multi-process, also known as multi-tasking, is a solution to sequential machining of several toolpath files. For each toolpath, you can define machining time interval, workpiece coordinate system (WCS), usage and how many times that the toolpath needs to be machined.

Before executing multi-tasking, ensure the following:

- The filename of each toolpath file is unique.
- The toolpath files are NC files.

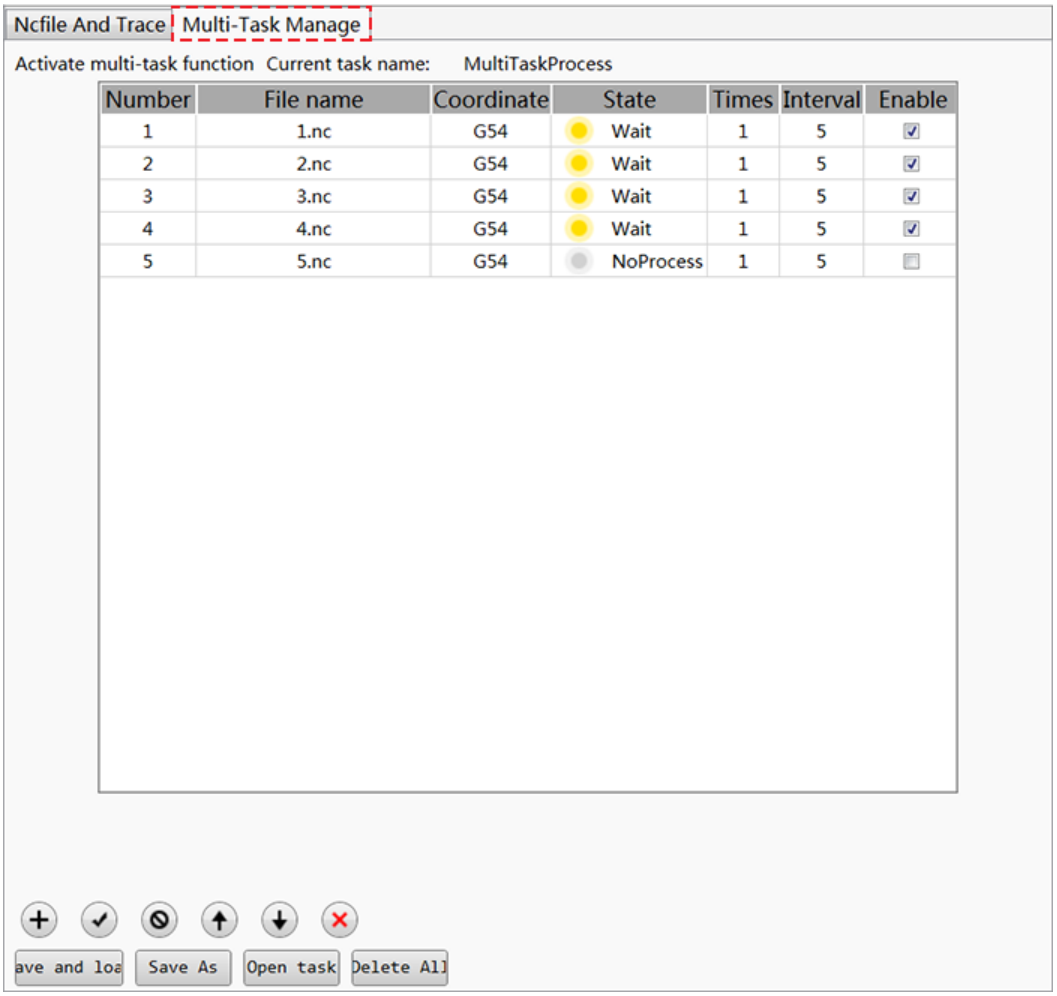
According to the selected software configuration, do one of the following to execute multi-tasking:

- Execute multi-process in Double-station Lambda 4S configuration
- Execute multi-process in other configurations

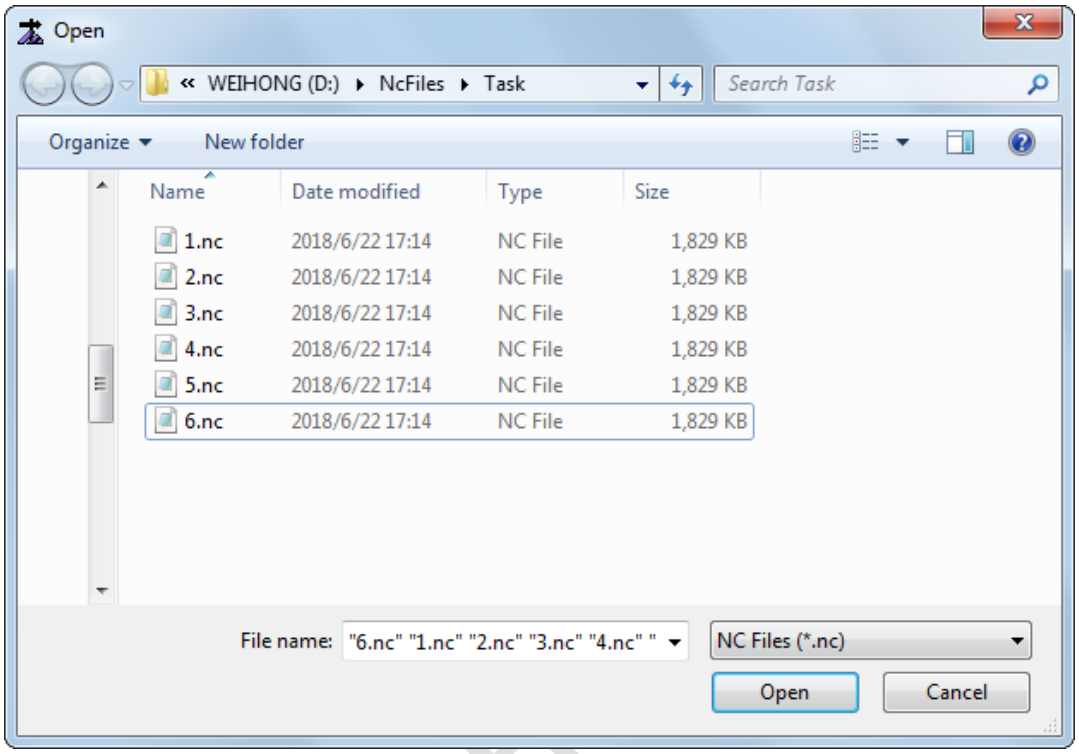
3.5.1. Execute Multi-process in Double-station Lambda 4S Configuration

If the current configuration is **Double-station Lambda 4S**, do the following to execute multi-process:

1. Switch to **Operator Interface** , and click **Multi-Task Manage** to enter into **Multi-Task Manage** window:



2. Click  to open **Open** dialog box, and select one or at most 60 NC toolpath file(s) that you need to add to the machining task:



3. Edit the toolpath file as desired:

   - Double click on the file name to select another toolpath file.
   - Double click on **WCS** to define the WCS for the toolpath.
   - Click on **Times** to set how many times the toolpath file will be machined in this task.
   - Click on **Interval** to set time interval between machining of two toolpath files.
   - Uncheck the box **Enable** to deselect the toolpath in the task.
   - Click  and  to adjust the position of the toolpath in the task.
   - Click  to remove the toolpath from the task.

4. Click **Save & Load** to save the task and load it into the system.

   All settings are saved to a task file with extension filename TSK.

5. Return to **Operator Interface** and do one of the following to start machining the task:

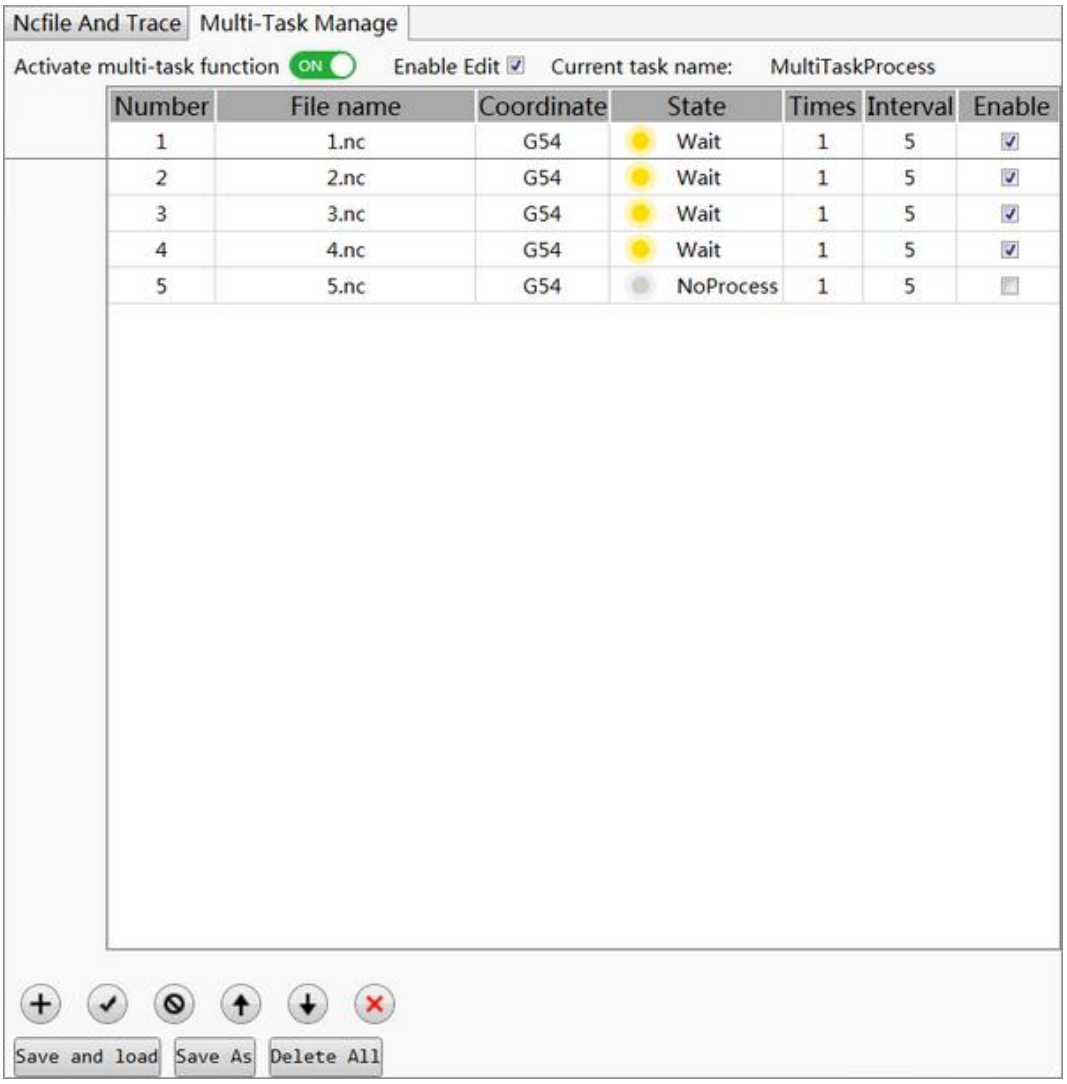
   - Switch to **Operator Interface**, and click **1 Start** or **2 Start** to enable station 1 or station 2 to start machining.
   - Click the button on the input port of Lamdba terminal board.

The system automatically executes toolpath files in sequence. Machining task finishes when all toolpath files have been finished.

3.5.2. Execute Multi-process in Other Configurations

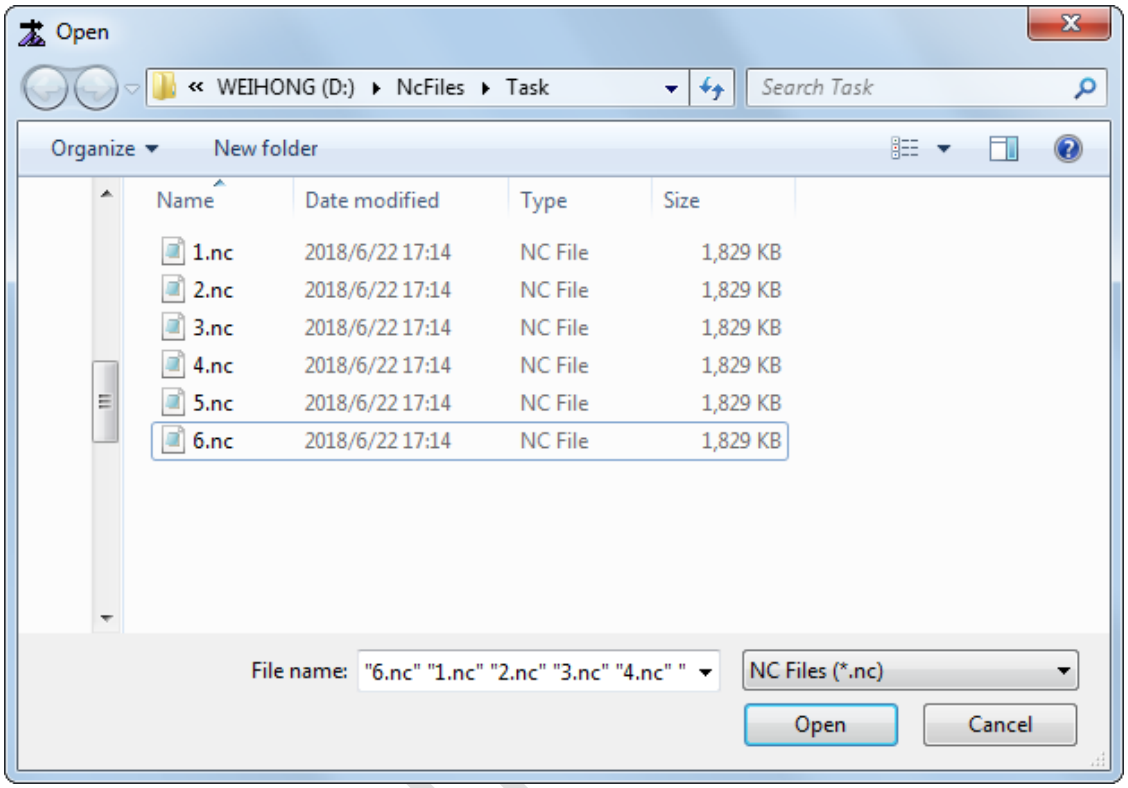
If the current configuration is not **Double-station Lambda 4S**, do the following to execute multi-process:

1. Switch to **Operator Interface** , and click **Multi-Task Manage** to enter into **Multi-Task Manage** window:



2. Click on **Activate multi-task function** to activate the function.

3. Click  to open **Open** dialog box, and select one or at most 60 NC toolpath file(s) that you need to add to the machining task.



4. Check the box **Enable Edit** and make modifications to toolpath files as desired:

   - Double click on the file name to select another toolpath file.
   - Double click on **WCS** to define the WCS for the toolpath.
   - Click on **Times** to set how many times the toolpath file will be machined in this task.
   - Click on **Interval** to set time interval between machining of two toolpath files.
   - Uncheck the box **Enable** to deselect the toolpath in the task.
   - Click  and  to adjust the position of the toolpath in the task.
   - Click  to remove the toolpath from the task.

5. To save the task setting, do one of the following:

   - Click **Save** to save it into default directory *D:* as default filename *MultiTaskProcess.tsk*.
   - Click **Save As** to save it into any directory as any filename.

6. To load the task into the system, do one of the following:
   - Switch to **Operator Interface**, and click **F1 Load**.
   - Switch to **Technician Interface**, and click **File** → **Open & Load** or click **F1 Load**.
7. To start machining the task, do one of the following:
   - Switch to **Operator Interface**, and click **F9 Start**.
   - Switch to **Technician Interface**, and click **Machine** → **Program Start** or **F9 Start**.

The system automatically executes the task from the first toolpath to the last one.

4. Quick Commissioning

This section mainly introduces the following operations for technicians to quickly know about how to do commissioning for **NC60A Multi-Process Control System**:

- Switch System Configuration
- Customize Function Configuration
- Clear I/O Port Alarms
- Set Commissioning Parameters
- Adjust Axis Direction
- Return to the Machine Origin

All operations are done under **Technician Interface**.

4.1. Switch System Configuration

The operation is used to choose the system configuration according to structure of the machine tools or terminal boards that are used. Currently, **Multi-process Control System** supports following configurations:

- Lamdba 3S/3L: applicable to Lamdba 3S/3L terminal board.
- Lamdba 4S: applicable to Lamdba 4S terminal board.
- Double-station Lamdba 4S: applicable to Lamdba 4S terminal board. Double-station machining improves efficiency.
- Lamdba 5M: applicable to Lamdba 5M terminal board and absolute type.

To switch the system configuration, do one of the following:

- To switch to Lambda 3S/3L, Lamdba 4S, or double-station Lamdba 4S configuration, do the following:
  1. Click  → **Technician Interface** to switch to technician interface.
  2. Click **File** → **Select Configuration** to select the target configuration.

- To switch to Lamdba 5M configuration, do the following:

  1. Open **Config** folder under the software installation directory *C:\Program Files\Weihong\NcStudio*
  2. Open file **controllertype.lua**, modify **controllerType** to **1** and save the file.
  3. Open folder **Std** and delete file **Ncstudio.plc.dll**.

  Start the software again. The system switches to Lambda 5M configuration.

- If Lamdba 5M configuration has been activated, and you want to switch back to other configurations, do the following:

  1. Open **Config** folder under the software installation directory *C:\Program Files\Weihong\NcStudio*
  2. Open file **controllertype.lua**, modify **controllerType** to **0** and save the file.
  3. Open folder **Std** and delete file **Ncstudio.plc.dll** file.

After setting, restart the software to make the setting effective.

4.2. Customize Function Configuration

NC60A Multi-Process Control System supports a variety of mechanical structures and customization of multi-cylinder-single-inverter and multi-cylinder-multi-inverter.

The supported mechanical structures include:

- Multi-cylinder-single-inverter

  There are multiple cylinders under one Z-axis, and one inverter controls multiple spindles ON and OFF.

  It includes:

  - Two-cylinder-single-inverter
  - Three-cylinder-single-inverter
  - Four-cylinder-single-inverter
- Multi-cylinder-multi-inverter

  There are multiple cylinders under one Z-axis, and multiple inverters control multiple spindles ON and OFF.

  It includes:

  - Two-cylinder-two-inverter
  - Three-cylinder-three-inverter
  - Four-cylinder-four-inverter

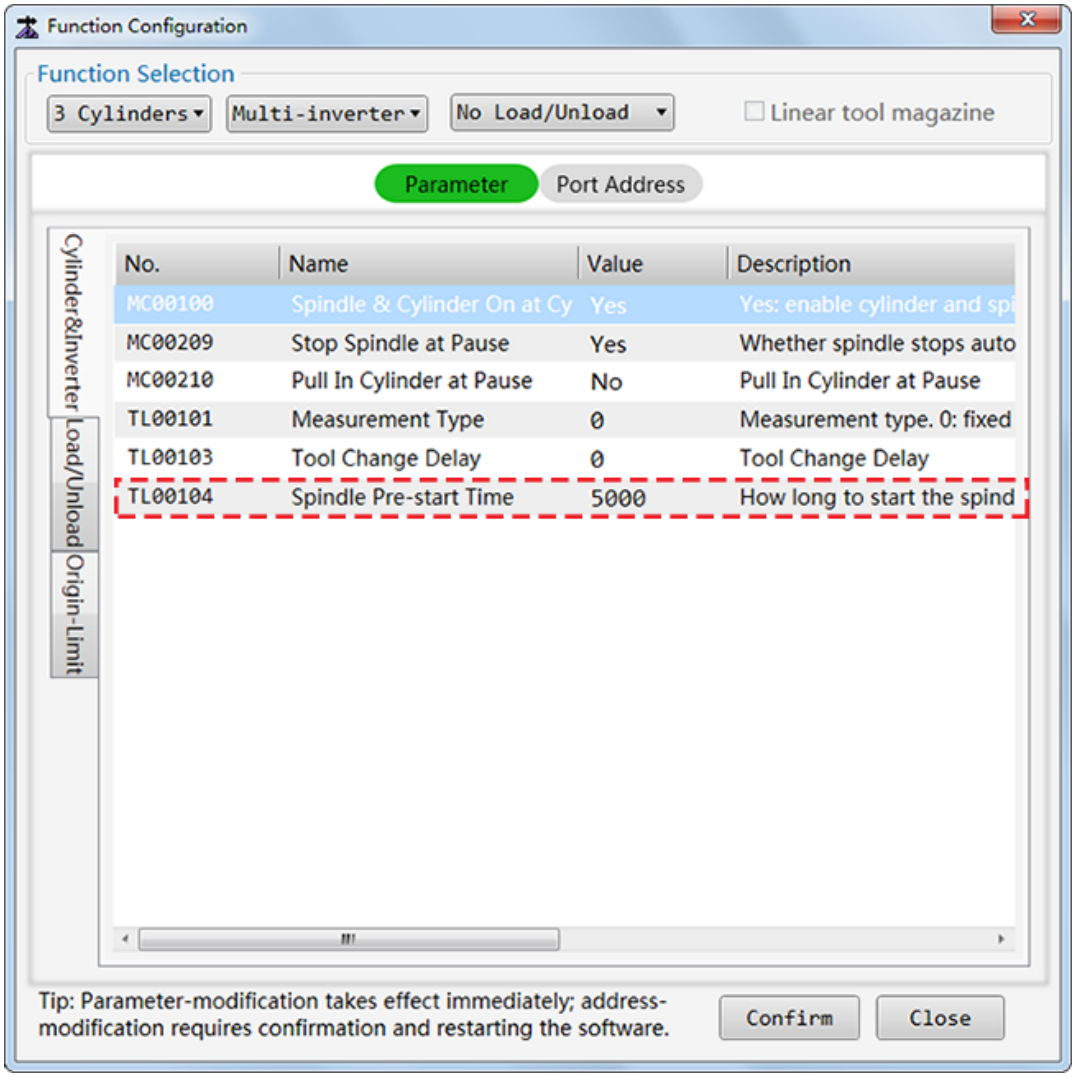
For relationship among the system configurations and customizable functions, see Software Configuration and Customizable Function Combination.

Note that under **Double-station Lambda 4S** configuration, no function can be customized.

4.2.1. Step-by-step

To customize function combination, do the following:

1. Click  → **Technician Interface** to switch to technician interface.
2. Click **File** → **Function Configuration**. **Functions Configuration** dialog box pops up:



3. Choose numbers of cylinders and inverters, according to actual mechanical structure.

4. Choose the functions you want to customize, including loading, unloading or linear tool magazine.

   Once configuration and functions you want to customize is defined, related parameters and IO ports are ready for modification.

5. Modify parameters and IO ports:

   - Modification to parameters takes effect immediately without additional attention.
   - Modification to port addresses takes effect after clicking **Confirm** to confirm the modification and restarting the software.

6. Click **Close**.

You can go to **Port** window to check updated IO information.

4.2.2. Example

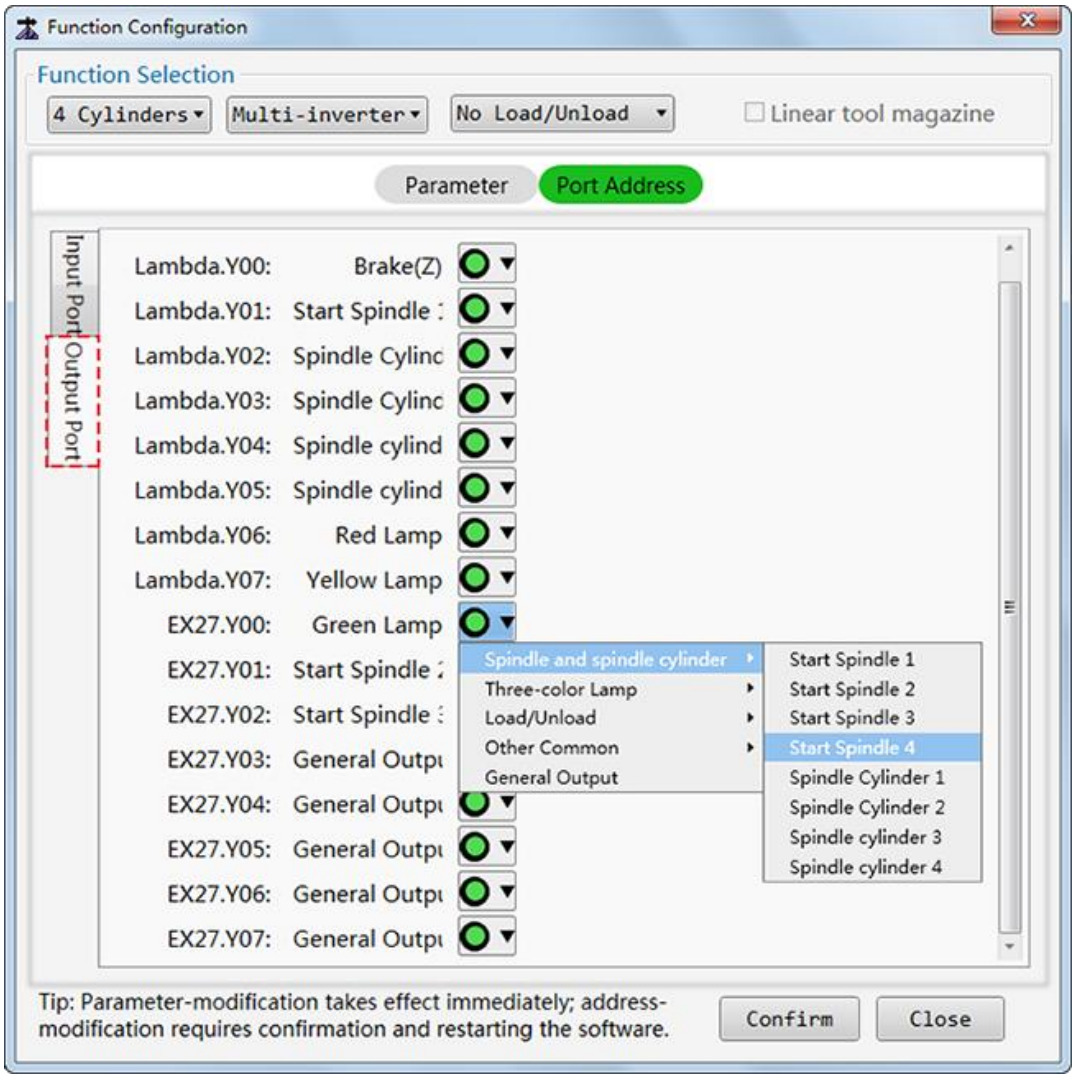
Taking **Lambda 4S** configuration, four-cylinders-four-inverters as an example, this section introduces how to customize cylinder ports.

To customize cylinder ports, do the following:

1. Switch to **Technician Interface**, and confirm that active configuration is **Lambda 4S**.
2. Click **File** → **Function Configuration**. **Functions Configuration** dialog box pops up.
3. In **Function Selection** area, select **4 cylinders** and **Multi-inverter**.
4. Click **Port Address** → **Output Port** tab to switch to the output port list.

5. Select any **General Output** port, and choose target definition on the pull-down list.

   You need to configure four spindle cylinder ports and four spindle ON ports in total and make sure all IO definitions are unique in the list.



6. Click **Confirm**, and restart the software.

Prompts may appear in following cases:

- If no cylinder-related port is configured or the configured ports are not enough, prompt will be *Missing mapping between spindle and spindle output ports.*
- If you have selected **None** in the **Function Selection** area, but you defined cylinder-related output ports, prompt will be *Not necessary spindle cylinder or spindle ON ports. It is recommended to delete them.*

After restarting the software, customization takes effect. Cylinder ports appear in both **Technician Interface** and **Operator Interface**.

4.3. Clear I/O Port Alarms

This operation is used to clear I/O port alarm so as to establish good communication between **NC60A Multi-Process Control System** and the servo driver.

The types of I/O alarms that need to be cleared can be divided into the following:

- Clear the alarm of terminal board disconnection
- Clear the E-stop alarm
- Clear the limit alarm
- Clear the driver alarm
- Clear the alarm of port polarity error

4.3.1. Clear the Alarm of Terminal Board Disconnection

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

1. Check if the wiring of terminal boards is connected correctly and securely:
   - If it is not, correct and tighten the wiring.
   - If it is, proceed to the next step.
2. Check if the port polarity in **Port** window is correct:
   - If it is not, modify the polarity of the corresponding port and the restart the software to make modification effective.
   - If it is, proceed to the next step.
3. Check if the terminal boards are damaged.

   If they are damaged, replace the terminal board.

4.3.2. 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 **Port** window.

4.3.3. Clear the Limit Alarm

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

The Limit alarm includes positive alarm and negative alarm.

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

4.3.4. Clear the Driver Alarm

To clear the driver alarm, do the following:

1. Check if the wiring of the driver is connected correctly and securely connected:
   - If it they are not, correct and tighten the wiring.
   - If they are, proceed to the next step.
2. Check if the polarity of **Servo Alarm** port is **NC**:

   If it is not, modify the port polarity to **NC**.

3. **Optional:** If the software is under Lamdba 5M configuration, check if the setting of parameter **Driver Station Address** matches with the driver station address of the machine tool:

   If it does not, modify the parameter setting according to the driver station address of the machine tool and restart the software to make modification effective.

4.3.5. Clear Alarm of Port Polarity Error

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

To clear alarm of port polarity error, modify the polarity of related ports in Port window.

4.4. Set Commissioning Parameters

This operation is used to set parameters for commissioning.

According to different configurations, this operation can be divided into the following:

- Set commissioning parameters for Lambda 5M configuration
- Set commissioning parameters for other configurations

For how to deal with related parameters, see Execute Parameter Related Operations.

4.4.1. Set Commissioning Parameters for Lambda 5M Configuration

If the system is under Lambda 5M configuration, set the following parameters:

Screw Pitch
Its setting should match with the actual condition.

Encoder Type
Its setting should match with the actual condition.
0: Incremental encoder
1: Absolute encoder
It is recommended to use absolute encoder and set the parameter to **1**.

Encoder Digit
It should match with the actual condition.

Electronic Gear Ratio
It consists of parameter **Numerator of Electronic Gear Ratio** and parameter **Denominator of Electronic Gear Ratio**.
Its setting should match with the actual condition.
It is set to **1:1** by default.

4.4.2. Set Commissioning Parameters for Other Configurations

If the control system is under other configurations, set ONE of the following parameters:

Pulse Equivalent
It refers to the moving distance of screw or rotation degree of rotary axis per pulse sent by the system, the minimum available distance controlled by the system as well.
It can be calculated in terms of the actual electronic gear ratio of the driver.
Smaller value gets higher machining precision and surface quality; larger value gets faster maximum feedrate.

Electronic Gear Ratio
It is used to set the electronic gear ratio of the driver.
It refers to 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 parameter **Pulse Equivalent**.

Calculation Formula

The calculation formula differs in motor type:

- Stepping motor

  The pulse equivalent equation is as follows:

```
Pulse Equivalent = Screw Pitch / [(360 / Stepping Angle) *
Subdivision of Stepping Motor * Mechanical Deceleration Ratio
```

- Servo motor

  In general, set the default value of pulse equivalent as 0.001mm/p and calculate electronic gear ratio (B/A).

  Thus, the electronic gear ratio equation is as follows:

  ```
  Electronic Gear Ratio = [(Encoder Resolution * Pulse Equivalent) /
  Screw Pitch] * Mechanical deceleration ratio
  ```

Among the calculation formulas, related concepts in the calculation formulas are as follows:

- Screw 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.
- 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.

  ```
  Mechanical Deceleration Ratio = Reducer Input Speed /
  Reducer Output Speed = Teeth No. of Driver Wheel / Teeth No. of Dri
  ving Wheel = Motor Rotational Speed / Screw Rotational Speed
  ```

- Encoder resolution: the pulse needed when the servo motor axis rotates one circle.

4.5. Adjust Axis Direction

This operation is used to adjust axis direction.

To adjust the moving direction of each axis, do the following(taking X-axis as an example):

1. Check current setting value of parameter **Axis Direction (X)**.
2. Judge the positive direction of X-axis according to the **Right Hand Principle**.
3. Switch to **Manual** mode, press **X+** or **X-** button to move X-axis and observe its moving direction.
4. **Optional** If the actual moving direction is opposite with the setting value, change the setting value to the opposite value.

Example

Current setting value for parameter **Axis Direction (X)** is 1. Manually move X-axis and find X-axis moves towards the negative direction.

Then, you need to change setting value of the parameter to **-1**.

4.6. Return to the Machine Origin

This operation is used to adjust coordinate position before machining.

According to encoders, it can be divided into:

- Return to the machine origin with incremental encoder.
- Return to the machine origin with absolute encoder.

4.6.1. Return to the Machine Origin with Incremental Encoder

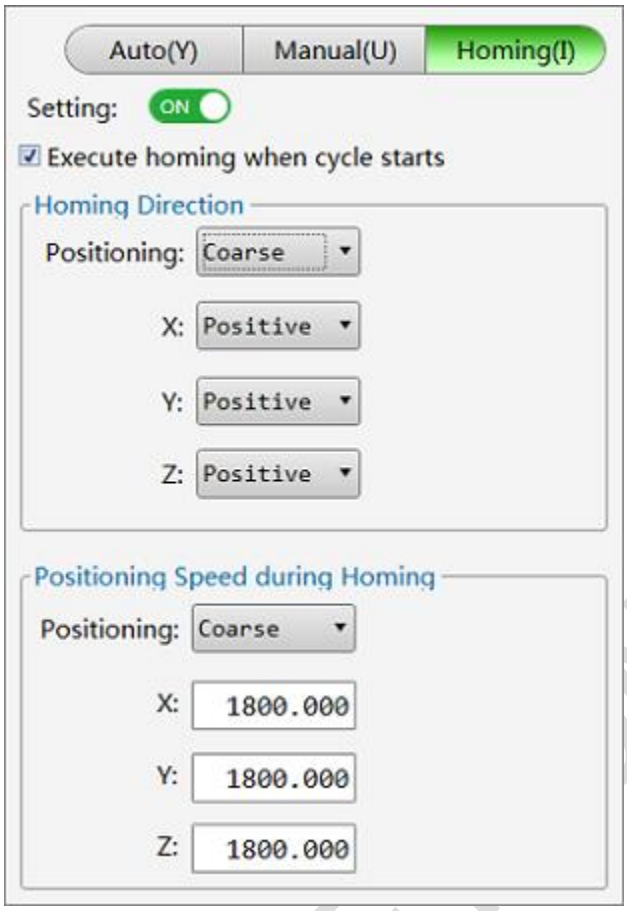
Machine coordinate system (MCS) is unique for the machine tool. Its origin is called machine origin or machine zero. It is fixed and defined before delivery of the machine tool.

After software starts, returning to the machine origin is required before machining.

Before returning to the machine origin with incremental encoder, do the following:

1. Check encoder type:
   - Check motor nameplate and ensure the used encoder is incremental encoder.
   - Set parameter **Encoder Type** to **0**.

2. Switch to **Homing** mode and check **Setting** to set related parameters:



   - **Homing Direction**
   - **Positioning Speed during Homing**
3. **Optional:** Check **Execute homing when software starts**. When software restarts, the system automatically returns to the machine origin.

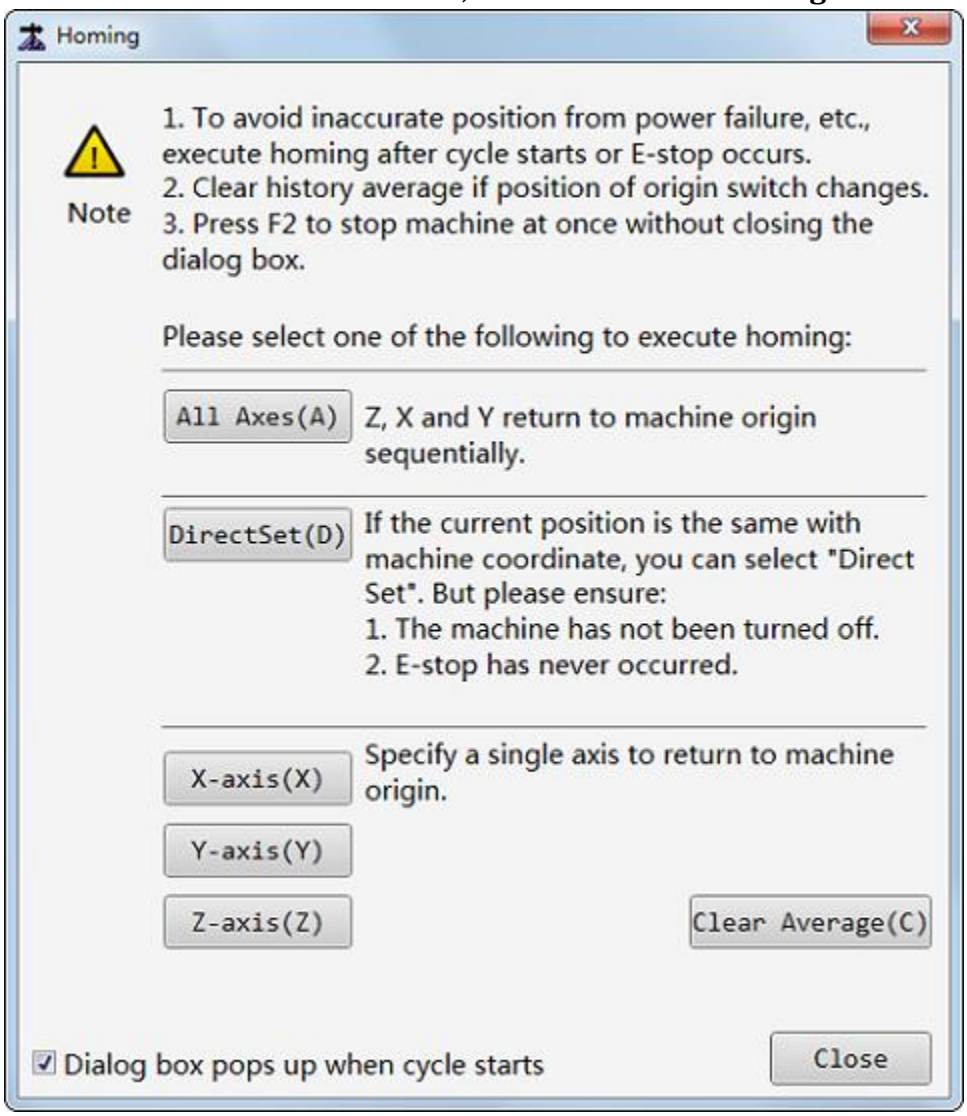
To return to the machine origin with incremental encoder, do one of the following:

- Return to the machine origin in **Homing** dialog box.
- Return to the machine origin through operational buttons.

4.6.1.1. Returning to the Machine Origin through Homing Dialog Box

To return to the machine origin through **Homing** dialog box, do the following:

1. To open **Homing** dialog box, do one of the following:
   - Under operator interface, click ☰ → **Homing**.
   - Under technician interface, click **Machine** → **Homing**.



If **Dialog box pops up when software starts** is checked, the dialog box pops up automatically.

2. **Optional:** If the position of the machine origin changes, click **Clear Average** to clear historical averages.

   Otherwise, the prompt *Distance between coarse and fine positioning signals exceeds the allowable range compared to the historical average. Failed to return to the machine origin.* pops up.

3. Choose one of the following to return to the machine origin:

   - All axes

     All axes return to the machine origin in order of Z-axis and XY axes.

   - Direct set

     Directly set the current position as the machine origin.

     With the method, you MUST ensure:

     - The position is the same with the machine coordinate.
     - The machine has never been turned off.
     - E-stop has never occurred.

   - Specify an axis

     Specify an axis to return to the machine origin.

     For safety, it is suggested to firstly return Z-axis, then X-axis and Y-axis.

After setting, the sigh appears before related axis in axis coordinates display area.

4.6.1.2. Return to the Machine Origin through Operational Buttons

To return to the machine origin through operational buttons, do the following:

1. Under technician interface, click **Homing** in mode control bar to switch to **Homing** mode.

2. Select related axis to return to the machine origin under the function window.

   - Click **X Homing** or **F1** to return X-axis to the machine origin.
   - Click **Y Homing** or **F2** to return Y-axis to the machine origin.
   - Click **Z Homing** or **F3** to return Z-axis to the machine origin.
   - Click **All Homing** or **F4** to return all axes to the machine origin in order of Z-axis and XY axes.

   For safety, it is suggested to firstly return Z-axis, then X-axis and Y-axis.

After setting, the sign appears before related axis in axis coordinates display area.

4.6.2. Return to the Machine Origin with Absolute Encoder

With absolute encoder, the machine origin is not defined before delivery of machine tools. You need to define it.

At this time, the machine origin is also called datum, the process of returning to the machine origin is also the process of setting datum.

This operation is required when the software is installed for the first time. Except updating or re-installing the system, there is no need to set datum because the system will automatically read datum of each axis.

Before setting datum, do the following:

1. Switch to the configuration of Lamdba 5M.

   See Switch System Configuration for details.

2. Check encoder type:

   - Check motor nameplate and ensure the used encoder is absolute encoder.
   - Set parameter **Encoder Type** to **1**.

To set datum, do the following:

1. Under technician interface, switch to **Manual** mode and move each axis to the position of the encoder origin.
2. Switch to **Homing** mode.
3. Click **Machine** → **Datum Setting** and do one of the following:

   - Select **Datum Setting(X)** to set the datum position of X-axis.
   - Select **Datum Setting(Y)** to set the datum position of Y-axis.
   - Select **Datum Setting(Z)** to set the datum position of Z-axis.
   - Select **Datum Setting(All)** to set the datum position of all axes.

5. Other Operations

This section introduces functions and operations including the following:

- Turn on the Spindle in Advance
- Execute Parameter Related Operations
- Set Offset
- Calibrate the Tool
- Execute Port Related Operations
- Re-connect Terminal Board with Control System ON
- Share Origin Switch Port with Limit Switches Ports
- Enable tool guide

5.1. Turn on the Spindle in Advance

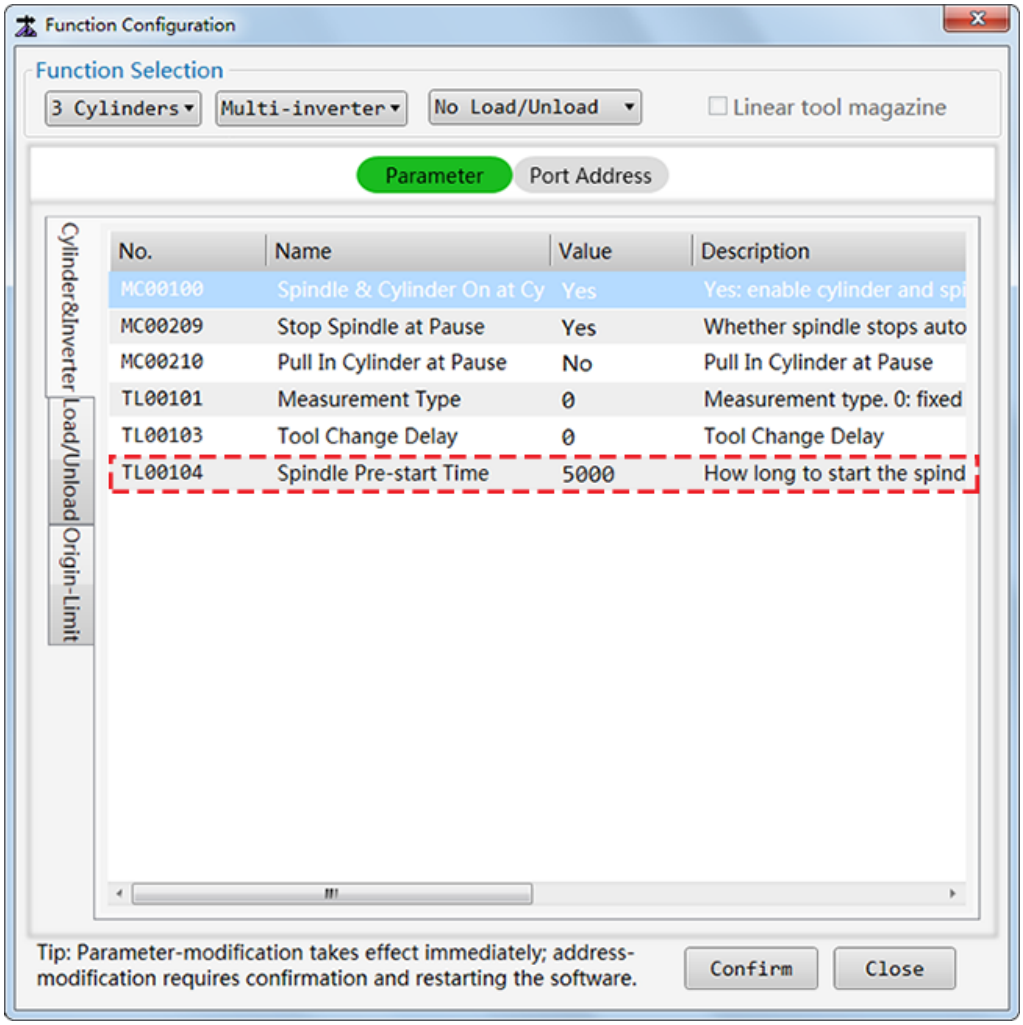
This operation is used to turn on the spindle of the target cylinder before switching to the target cylinder, so as to reduce the spindle preparation time and improve machining efficiency. It is controlled by the parameter **Spindle Pre-start Time**.

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

To turn on the spindle in advance, do one of the following:

- If the system is under Lamdba 3S/3L, Lamdba 4S, or Lamdba 5M configuration, do the following:

  1. Click  → **Technician Interface** to switch to technician interface.
  2. Click **File** → **Function Configuration**. **Function Configuration** dialog box pops up.



  3. Select functions in **Function Selection** area and ensure the selected cylinder combination belongs to multi-cylinder-multi-inverter.
  4. Click **Parameter** → **Cylinder&Inverter**, find the parameter **Spindle Pre-start Time** (Unit: ms) and modify the parameter value based on actual needs.

- If the system is under double-station Lamdba 4S configuration, do the following:

  1. Click  → **Technician Interface** to switch to technician interface.
  2. Click **System** → **Global Parameters**, select **Manufacturer** permission and enter the manufacturer password.
  3. Find the parameter **Spindle Pre-start Time** and modify the parameter value according to actual needs.

     See Execute Parameter Related Operations for details.

Example

If the parameter **Spindle Pre-start Time** is set to **1000** and changing from T1 to T3 for is required. Then:

- Before executing T3 command, the system turns on the spindle corresponding to T3 one second in advance.
- When executing the T3 command, the system does the following:
  - Close the spindle and cylinder corresponding to T1.
  - Pull down the cylinder corresponding to T3.

5.2. Enable Loading/Unloading

This operation is used to load/unload material.

It is only available to Lamdba 4S configuration and Lamdba 5M configuration.

Thus, before enabling this function, ensure the following:

- The used terminal board is Lambda 4S or Lambda 5M.
- The software has been switched to Lambda 4S configuration or Lambda 5M configuration.

  See Switch System Configuration for how to switch configuration.
- Related parameters and port addresses related to loading and unloading are configured, because the system does not configure the loading and unloading port by default.

  If the port address is not set, the related buttons are in gray and the loading and unloading function is disabled.

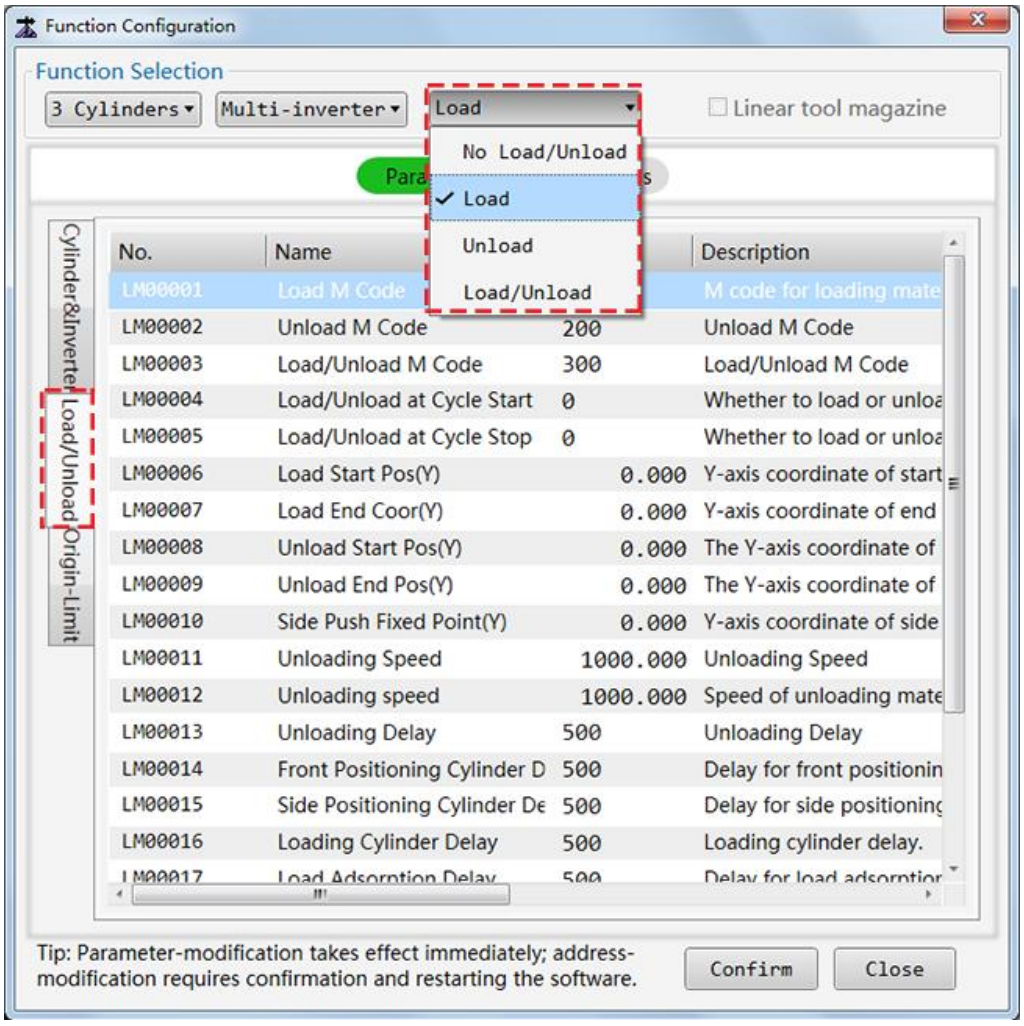
  See Customize Function Configuration for how to configure the function.

To enable loading and unloading material, do one of the following:

- Click  to switch to operator interface and click **Load**, **Unload** or **Load/Unload** in loading and unloading control area. The system automatically executes related operation.

- Click  → **Technician Interface** to switch to technician interface and do the following:

    1. Click **File** → **Function Configuration**. **Function Configuration** dialog box pops up:



    2. Select one of the following in **Function Selection** area:

        - **Load:** load material.
        - **Unload:** unload material.
        - **Load/Unload:** load and unload material.

    3. Select **Load/Unload** in the left of parameter list.

4. Set the following parameters to **1**, **2** or **3** in the parameter list:

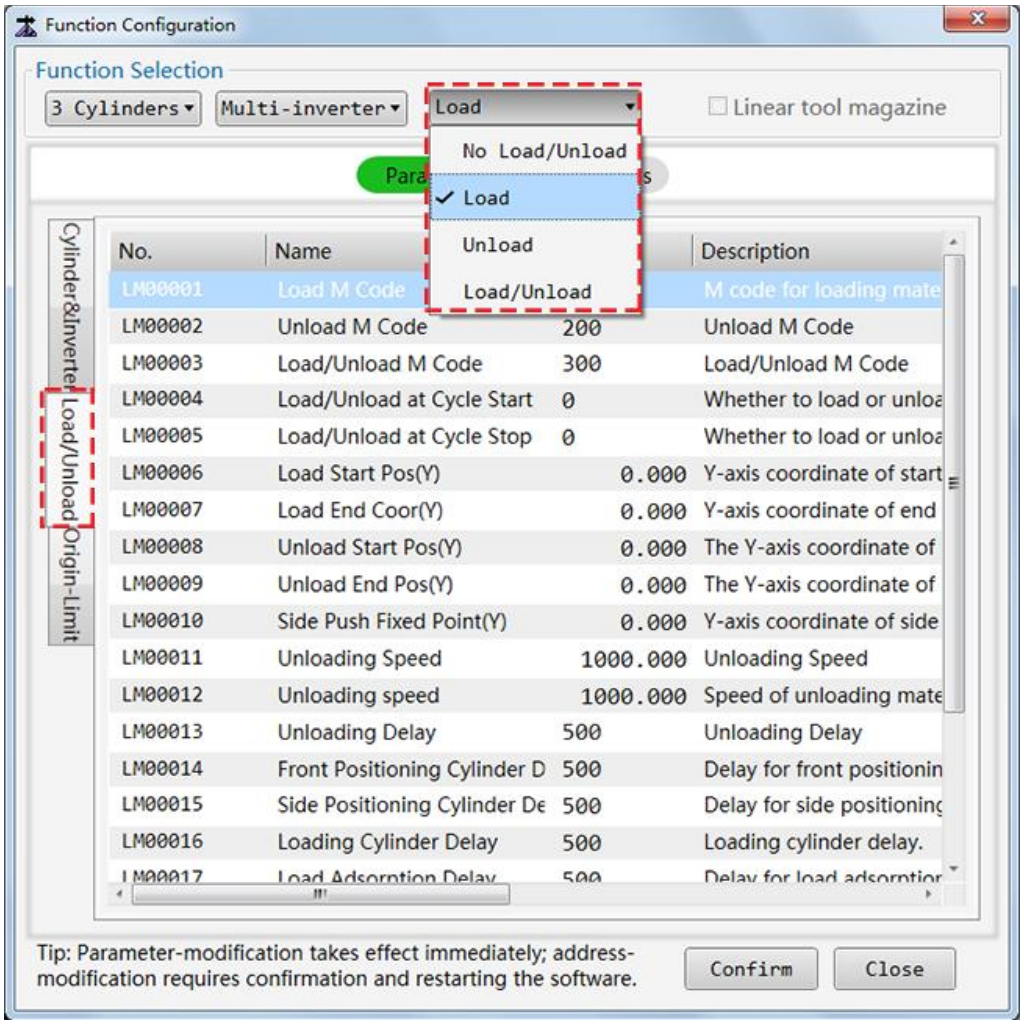
   Load/Unload at Cycle Start
   Whether to load or unload material when cycle starts.
   0: none.
   1: load material.
   2: unload material.
   3: load/unload material.

   Load/Unload at Cycle Stop
   Whether to load or unload material when cycle stops.
   0: none.
   1: load material.
   2: unload material.
   3: load/unload material.

   The system automatically loads/unloads material based on the set value.

- Click → **Technician Interface** to switch to technician interface and do the following:

1. Click **File** → **Function Configuration**. **Function Configuration** dialog box pops up:



2. Select **Load/unload** in the left of the parameter list and check load/unload related codes:

   - **Load M Code**
   - **Unload M Code**
   - **Load/unload M Code**

3. Do one of the following:

   - Click **Machine** → **MDI**, enter load/unload related code and execute **Start**.
   - When programming, write load/unload related code into tool path.

5.3. Execute Parameter Related Operations

NC60A Multi-process Cutting System provides a wide range of motion control parameters and technic parameters.

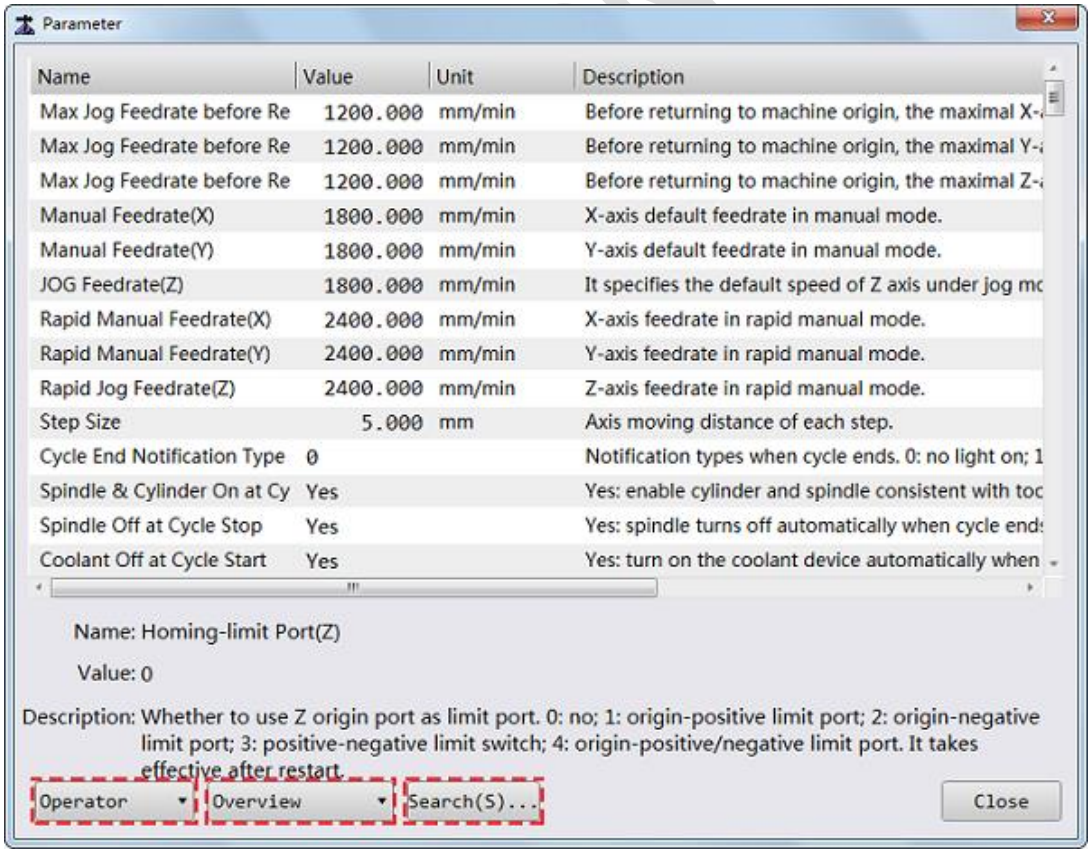
To execute parameter related operations, do the following:

1. Click → **Technician Interface** to switch to technician interface.
2. Do one of the following:
   - Check and modify parameters
   - Customize common parameters
   - Back up and restore parameter settings

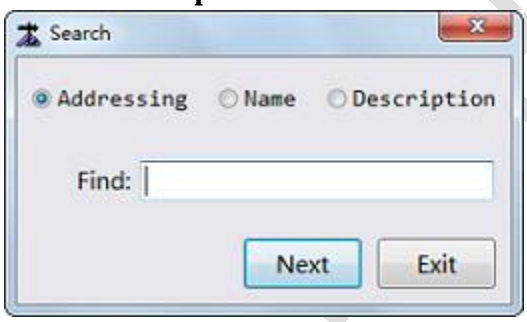
5.3.1. Check and Modify Parameters

To check and modify parameters, do the following:

1. Click **System** → **Global Parameters**. **Parameter** dialog box pops up:



2. Select access permission:
   - **Operator:** to show parameters under operator permission. It is the default permission.
   - **Manufacturer:** to show parameters under manufacturer permission. Enter the manufacturer password to access these parameters.
3. Select parameter classification to see related parameters:
   - **Overview:** to show all parameters with operator/manufacturer permission. It is the default classification.
   - **Axis:** to show axis parameters.
   - **Machining:** to show machining parameters.
   - **Planning:** to show planning parameters.
   - **Compensation:** to show compensation parameters.
   - **Tool:** to show tool parameters.
   - **Others:** to show other parameters.
4. **Optional:** Click **Search** to search parameters by:
   - **Addressing**
   - **Name**
   - **Description**



5. After finding the parameter, double-click the line of the parameter, enter a value and click **OK**.

If the effective time for the modified parameter is after restart, restart the software after all the parameters have been modified.

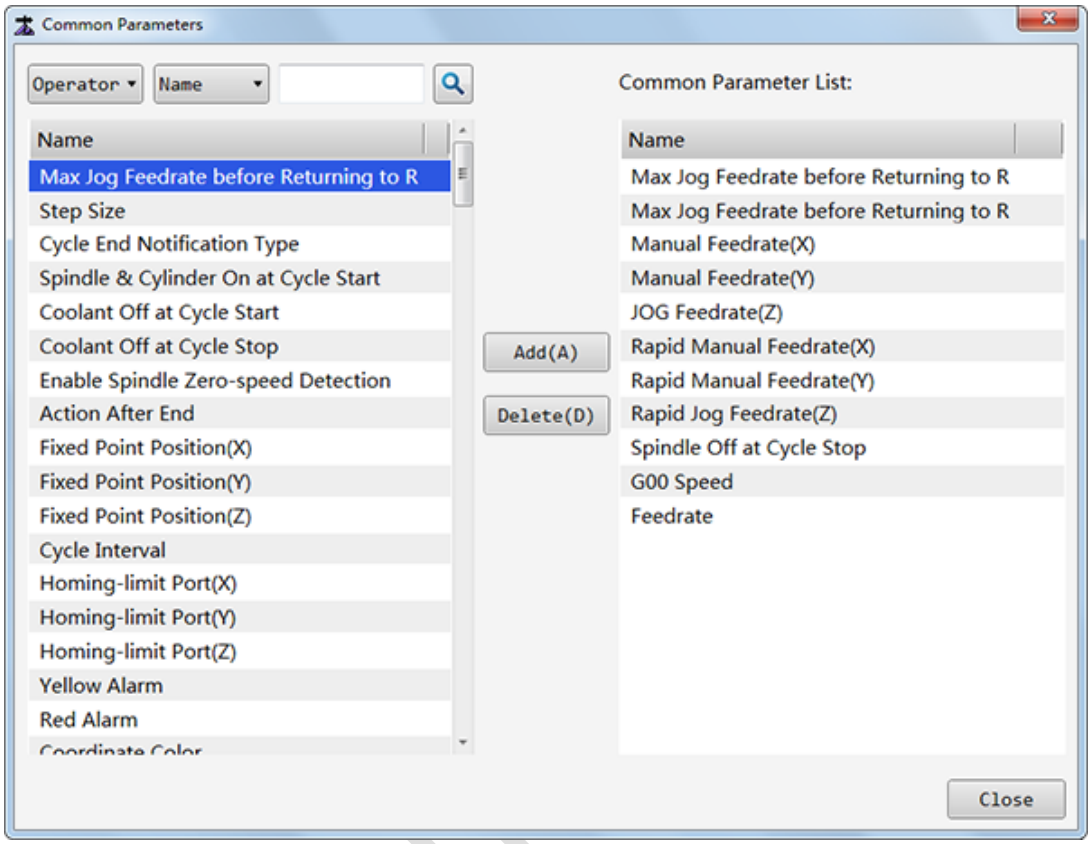
5.3.2. Customize Common Parameters

The system supports customizing common parameters, so that you can quickly find and manage frequently used parameters.

To customize common parameters, do the following:

1. Click → **Technician Interface** to switch to technician interface.
2. Click **Common** to enter into **Common** window.

3. Click **Set** in the lower right corner. **Common Parameters** dialog box pops up:



4. Find the parameter that you would like to add as common parameter by one of the following:
   - Permission: **Operator** and **Manufacturer**. Operator permission is the default.
   - Search: by **Name**, **Description** and **Addressing**. Search by name is the default.
5. Click **Add** to add the parameter to the right **Common Parameter List**.
6. Repeat step **3** and **4** to add all parameters that you would like to add as common parameters.
7. **Optional:** If you need to move a parameter out of the right **Common Parameter List**, select the parameter on the **Common Parameter List** and click **Delete**.

After setting, click **Close**. The parameters in the **Common Parameter List** instantly show in **Common** window.

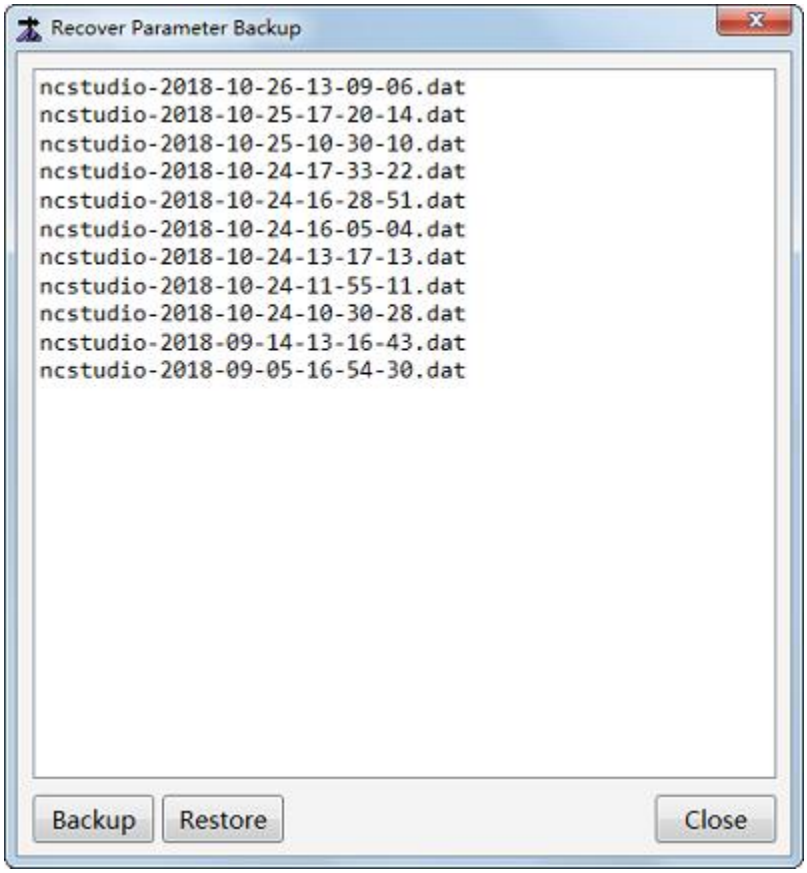
5.3.3. Back up and Restore Parameter Settings

This operation is used to back up current parameter settings for restoring them later.

Before backing up and restoring parameter settings, ensure parameters have been set correctly.

To back up and restore parameter settings, do the following:

1. Click **System** → **Recover Parameter Backup**. **Recover Parameter Backup** dialog box pops up:



2. Do one of the following:

   - To back up parameter settings, click **Backup** to start to back up current parameter settings.

     The backup file are saved as a DAT file and saved under path *Weihong\NcStudio\Config*. And the default file name consists of ncstudio and system time.

   - To restore parameter settings, select a backup file to be restored and click **Restore**.

5.4. Set Offset

Offset in the system includes the following:

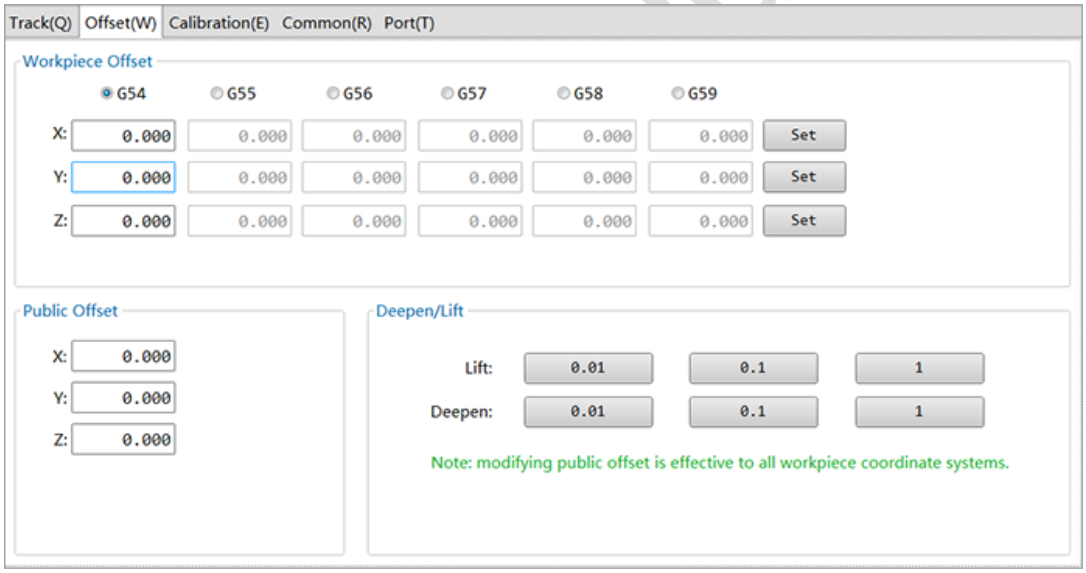
- Workpiece offset: it shows the distance of the workpiece origin relative to the machine origin.
- Public offset: aiming at all WCSs, it is used to adjust workpiece origin of X, Y, and Z-axis.

The relationship among workpiece origin, tool offset and public offset is as follows:

```
Workpiece Coordinate = Machine Coordinate - Workpiece Offset –
Public Offset - Tool Offset
```

To set offset, do the following:

1. Click  → **Technician Interface** to switch to technician interface.
2. Click **Offset** to enter into **Offset** window:



3. To set the workpiece offset, do the following:
    1. Select a workpiece coordinate system.
    2. Click the input box of related axis and set a value.

    Click **Set** to set the current position as workpiece origin.
4. To set the public offset, do one of the following:
    - Click the input box of related axis and set a value.
    - Use **Deepen/Lift** to deepen/lift Z-axis **0.01mm**, **0.1mm** or **1mm** to adjust the public offset of Z-axis.

5.5. Calibrate the Tool

This operation introduces how to execute tool calibration through setting parameters that are related to tool calibration.

In terms of permission, it can be divided into the following:

- Calibrate the tool in operator interface
- Calibrate the tool in technician interface

5.5.1. Calibrate the Tool in Operator Interface

To calibrate the tool, do the following:

1. Switch to **Operator** interface.
2. Do one of the following:
   - To calibrate the current tool, do one of the following:
     - Click **Set T Length** to modify Z-axis offset of current tool.
     - Pull down a cylinder and click **Measure** to measure length of current tool and set it to tool offset.
   - To calibrate all tools, click **Measure All** to measure length of all tools and set it to tool offset.

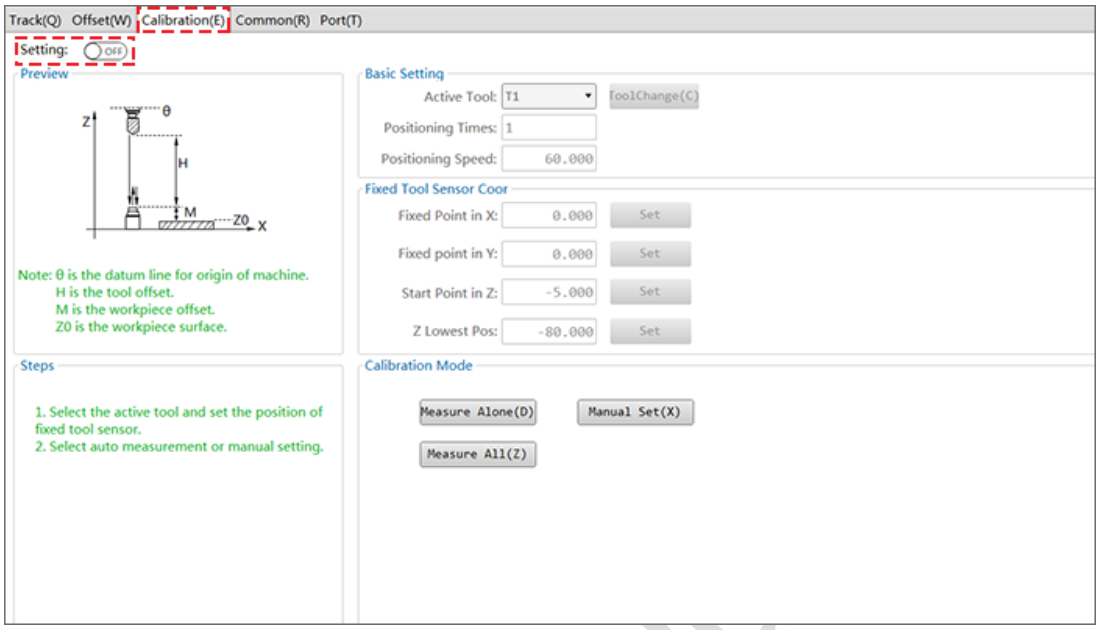
     The system starts calibrating the current tool and then the second tool until it finishes calibrating all tools.

5.5.2. Calibrate the Tool in Technician Interface

To calibrate the tool, do the following:

1. Click  → **Technician Interface** to switch to technician interface.

2. Click **Calibration** to enter into **Calibration** window:



3. Click on **Setting**, and enter manufacturer's password.

   All options are available to modification.

4. In the **Basic Setting** area, do the following:

   - To prepare the spindle tool, select the tool number from the drop-down list, and click **Tool Change** button.
   - To improve tool calibration accuracy, set **Fine Positioning Times**.
   - To improve tool calibration efficiency, set **Fine Positioning Speed**.

5. In **Fixed Tool Sensor Coor** area, set following parameters to define the position of the tool sensor.

   - X/Y fixed point
   - Z start point
   - Z lowest point

6. Do one of the following:

   - To measure length of current tool, pull down a cylinder and click **Measure Alone**. The system automatically executes tool calibration, and saves the result to Z-axis offset for the tool.

   - To measure length of all tools, click **Measure All**. The system automatically executes tool calibration, and saves the results to Z-axis for all tools.

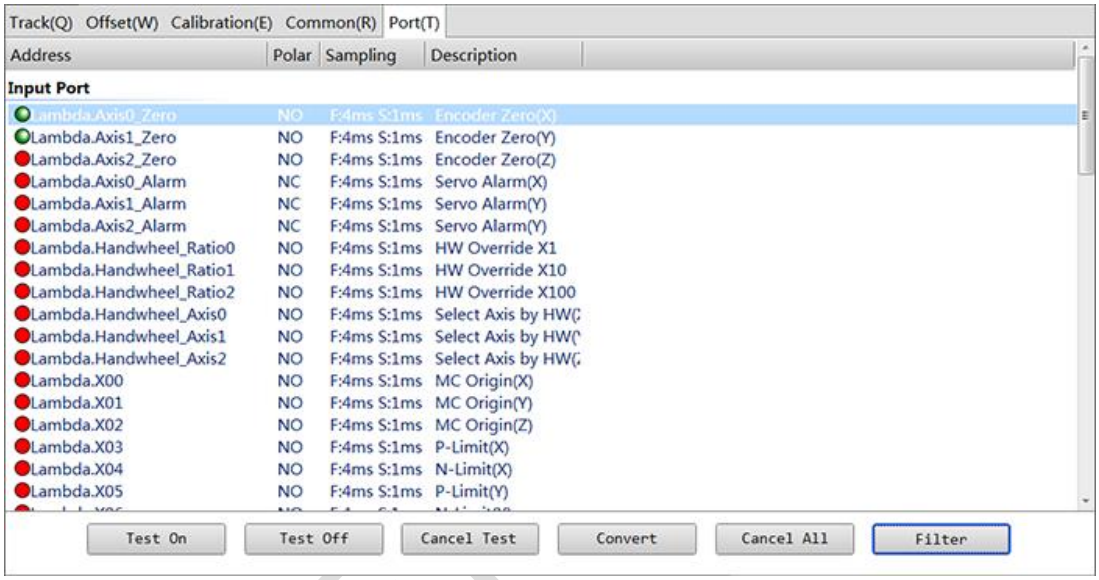
   - To set the current Z-axis machine coordinate to the tool offset of current tool, click Manual Set.

5.6. Execute Port Related Operations

By controlling over the input or output ports, you can monitor the status of the machine tool, conduct simulation test and so on.

To execute port related operations, do the following:

1. Click → **Technician Interface** to switch to technician interface.
2. Click **Port** to enter into **Port** window:



3. Do one of the following:
   - Conduct test: click **Test On** and **Test Off** to conduct or close simulation test and click **Cancel Test** to cancel test.
   - Modify port polarity: click **Convert** to modify port polarity.
   - Set sampling: click **Filter** to set sampling interval and enable/disable filter.

Except these operations, you can also modify and configure port according to the actual usage. See Customize Function Configuration for details.

5.7. Re-connect Terminal Board with Control Software ON

When the control system is running and terminal board disconnects with power supply, the control system will prompt with alarm and turn into emergency stop state. With this function, you can re-power on the terminal board and remove alarm, without restarting the software. After that, the software and terminal board can resume good communication.

To use the function, make sure that the terminal board and the software work well before power interruption to the terminal board occurs.

To re-connect the terminal board with control software on, do the following:

1. Click  → **Technician Interface** to switch to technician interface.
2. Click **System** → **Re-connect Lambda Controller**.

5.8. Share Origin Switch Port with Limit Switches Ports

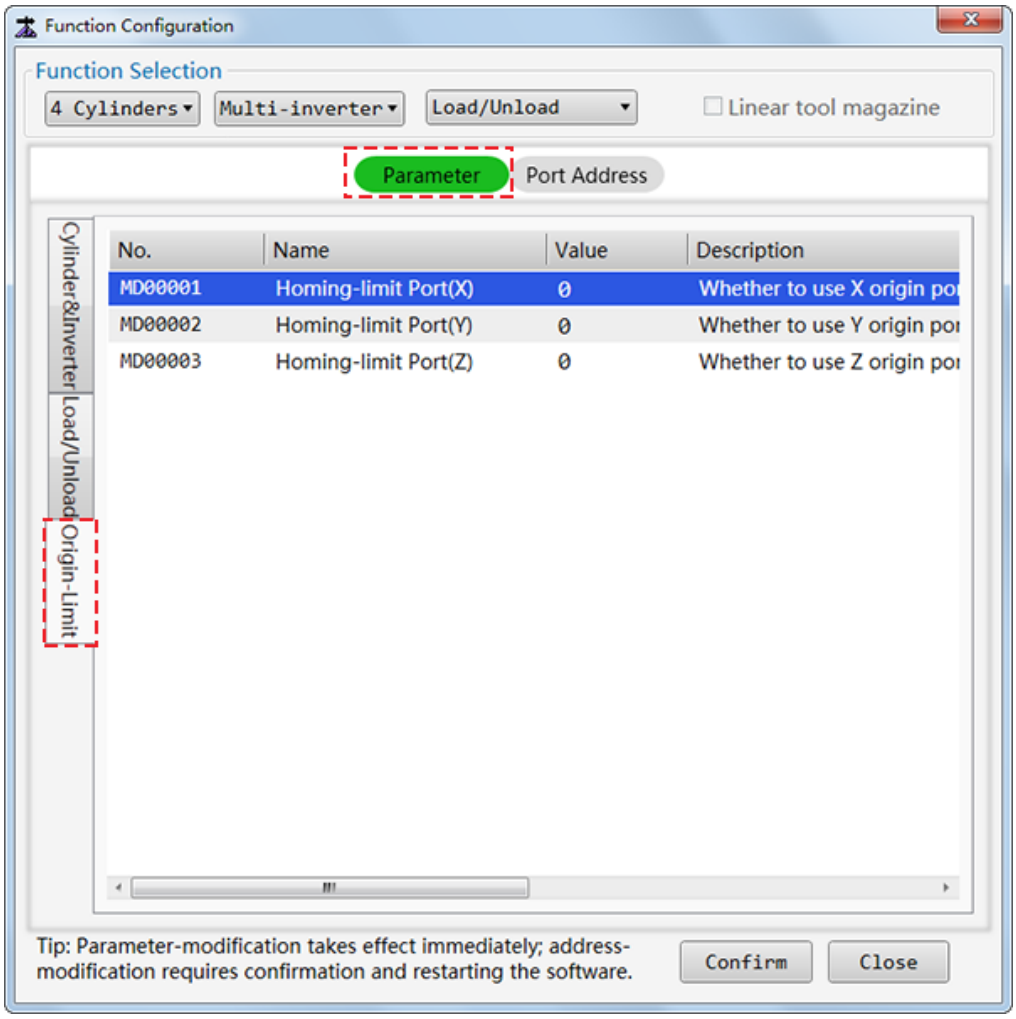
This function is used to save IO ports, for example, origin switch port can be used as limit switch port at the same time.

Origin switch port and limit switches ports share in following ways:

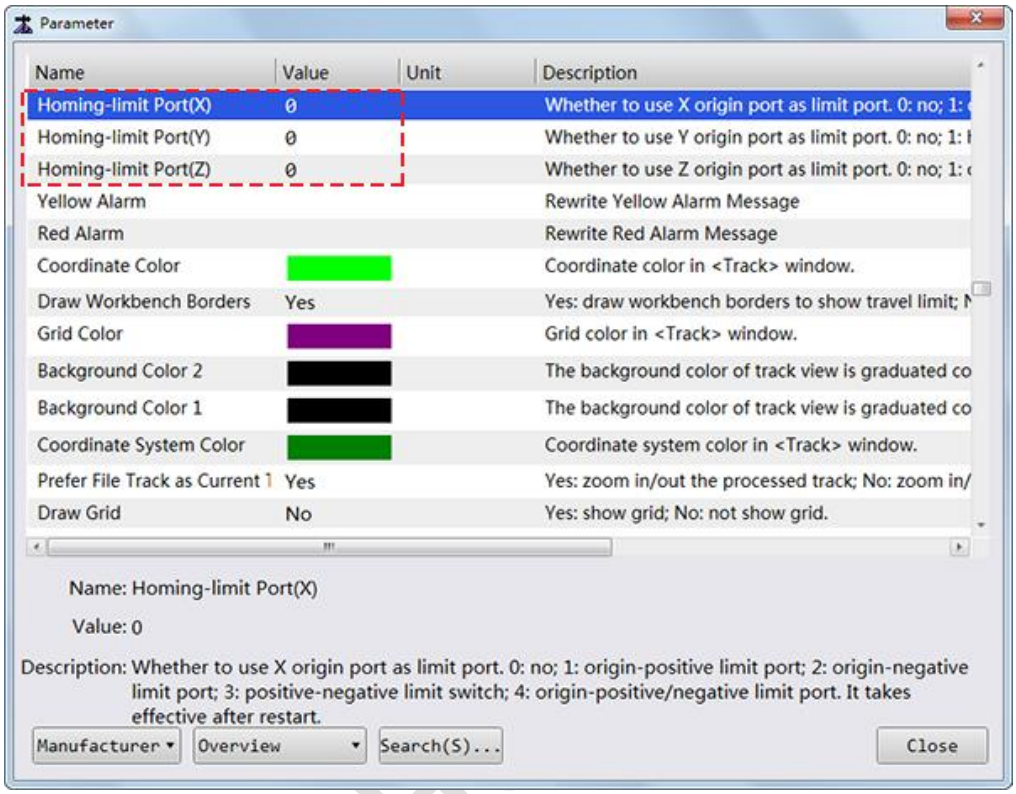
- Origin switch port used as positive limit switch port
- Origin switch port used as negative limit switch port
- Positive switch port used as negative limit switch port
- Origin switch port, positive limit switch port and negative limit switch port share the same port

To share the origin switch port and limit switches ports, do the following:

1. Click → **Technician Interface** to switch to technician interface.
2. To locate parameters **Origin-Limit Port**, do one of the following:
   - Click **File** → **Function Configuration** → **Parameter** → **Origin-Limit**:

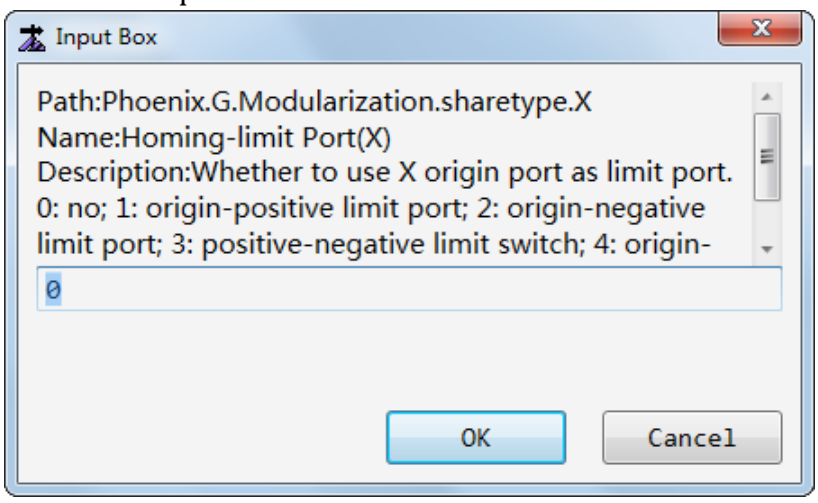


– Click **System** → **Global Parameters**, and find parameter in the list:



3. Select the axis, its origin and limit switch and modify their parameter settings:

    1. N/A (default)
    2. Share origin port with positive limit switch port
    3. Share origin port with negative limit switch port
    4. Share positive and negative limit switch port
    5. Origin port, positive limit switch and negative limit switch share the same port



4. Restart the software to validate modifications.

All saved ports are reset to **General Output** ports.

In case that origin switch, positive limit switch and negative limit switch share the same port, when port signal is triggered, decide which limit switch is working according to the current machine coordinate, as follows:

- If current coordinate < half of travel limit, negative limit switch works.
- If current coordinate > half of travel limit, positive limit switch works.
- If current coordinate = half of travel limit, further consider last moving direction, as follows:
  - If the axis is moving towards the positive direction, positive limit switch works.
  - If the axis is moving towards the negative direction, negative limit switch works.

5.9. Enable Tool Guide

Tool Guide function is available only when linear tool magazine is used.

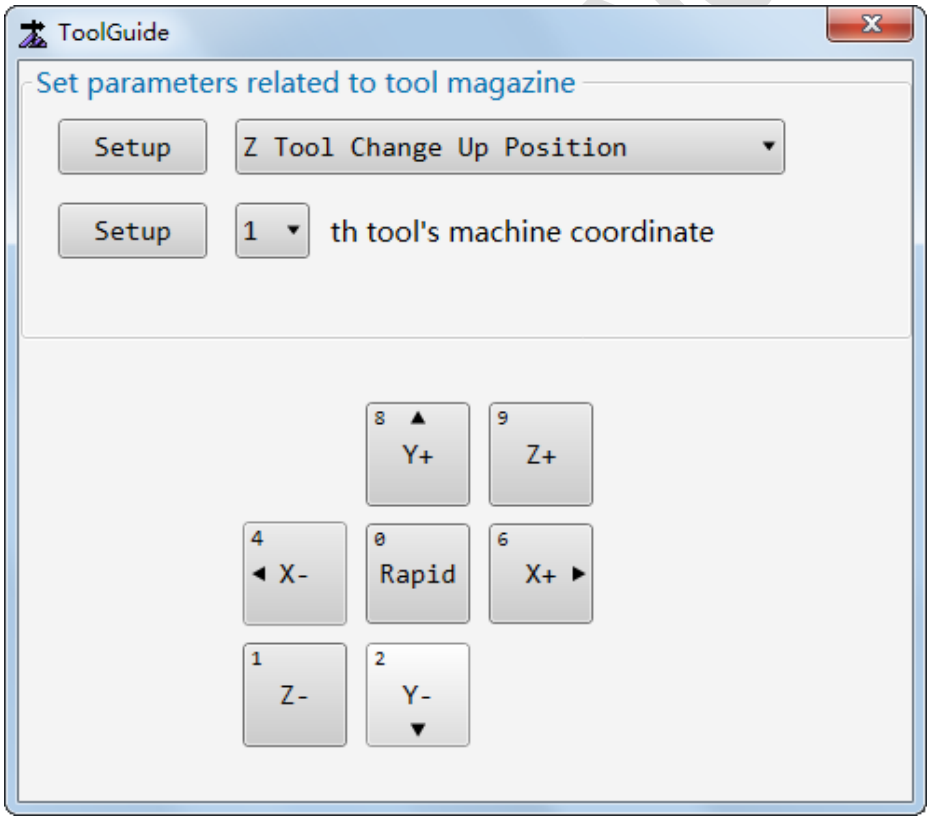
Enable **Tool Guide** function to do the following:

- Change tool ahead position
- Change tool upper position
- Change tool lower position
- Tool positions in tool magazine

Before enabling tool guide, make sure **Linear tool magazine** has been configured. See Customize Function Configuration for details.

To enable **Tool Guide** function, do the following:

1. Click → **Technician Interface** to switch to technician interface.
2. Click **ToolGuide** in **Tool Info** area. **ToolGuide** dialog box pops up:



3. Select the item to be set.
4. Click axis direction button to manually move the axis to desired position, and click Setup to set current machine coordinate to the item.

6. Software Configuration and Customizable Function Combination

In **Multi-process Control System**, you can customize function combinations under the following configurations:

- Lamdba 3S/3L configuration
- Lamdba 4S configuration
- Lamdba 5M configuration

Available function combinations under different configurations are as follows:

- Lamdba 3S/3L configuration

Mechanical Structure	Load/Unload	Linear Tool Magazine	Origin-Limit
None	N/A	Suitable	Suitable
Two-cylinder-single-inverter	N/A	N/A	Suitable
Two-cylinder-multi-inverter	N/A	N/A	Suitable
Three-cylinder-single-inverter	N/A	N/A	Suitable
Four-cylinder-single-inverter	N/A	N/A	Suitable

- Lamdba 4S configuration and Lamdba 5M configuration

Mechanical Structure	Load/Unload	Linear Tool Magazine	Origin-Limit
None	Suitable	Suitable	Suitable
Two-cylinder-single-inverter	Suitable	N/A	Suitable
Two cylinder-multi-inverter	Suitable	N/A	Suitable
Three-cylinder-single-inverter	Suitable	N/A	Suitable
Three-cylinder-multi-inverter	Suitable	N/A	Suitable
Four-cylinder-single-inverter	Suitable	N/A	Suitable
Four-cylinder-multi-inverter	Suitable	N/A	Suitable

SPECIALIZED / CONCENTRATED / FOCUSED

Shanghai Weihong Electronic Technology Co., Ltd.

Address: No. 1590, Huhang Rd., Fengxian, Shanghai, China, 201401

Hot-line: 400 882 9188

Website: www.weihong.com.cn/en