200 Series CNC Integrated System Manufacturers' Manual

Version: 2019.03 1st Version
Author: Document Department

1 Overview

Through this section, you can quickly know components of **200 Series**.

200 Series include the following models:

- **NK260 integrated CNC system** (hereinafter referred to as **NK260**)
- **NK280 integrated CNC system** (hereinafter referred to as **NK280**)
- **NK280B integrated CNC system** (hereinafter referred to as **NK280B**)

You can see Model Selection for how to select the desired one among them.

All of them consist of hardware and software:

Hardware

- **NK260 / NK280 / NK280B** host
- PCBA EX9A terminal board
- DB9M/F cables

Note: For **NK280B**, the used terminal board is Lambda 5M RJ45.

Software

NK260 / NK280 / NK280B software

See Main Software Interface for the introduction to the main interface of the software.

1.1 Model Selection

You can select the desired integrated CNC system that matches with the structure of your machine tool and machining demands according to its application:

- **NK260**

  Three-axis engraving machine

- **NK280**

  - Three + one axis (double Y / single Y + single servo magazine)
  - Four-axis simultaneous movement, double Z engraving, three + two axis (single Y + double servo magazine, double Y + single servo magazine, double Y + double servo magazine)
  - Five-axis (four + one) engraving machine

- **NK280B**

  - Three-axis bilateral milling machine
  - Three-axis bus control, three + one axis (double Y / single Y + single servo magazine)
  - Bus control four-axis simultaneous movement, double Z engraving, three + two axis (single Y + double servo magazine, double Y + single servo magazine, double Y + double servo magazine)

1.2 Host

This section mainly takes **NK260** host as an example, and explains the differences among these CNC integrated systems separately as well.

1.2.1 Front View

Front view of the host is as follows:

1. Display screen
2. Main function buttons
3. Alpha-numeric keypad
4. Mode buttons
5. E-stop button
6. Auxiliary function buttons
7. Override switches
8. USB interface
9. Movement control buttons
10. Operational buttons

1.2.1.1 Main Function Buttons

They are used to switch to the following functional areas:

- 

: to enter into **State** functional area.
- 

: to enter into **Advanced** functional area.
- 

: to enter into **Program** functional area.
- 

: to enter into **Parameter** functional area.
- 

: to enter into **System** functional area.

1.2.1.2 Alpha-numeric Keypad

It consists of the set of keys for operating the integrated CNC system. And its use is generally the same with that of the computer keyboard, except the following:

- Enter an alphabet
  - For **NK260**, directly press the button to enter the alphabet on the left-upper part, and press **Shift** and an alphabet key to enter the alphabet on the right-lower part.
  - For **NK280 / NK280B**, the opposite is true.
- Numeric keys are used as axis direction keys in **Manual** mode.
  - Only press the axis direction key to move the axis at jog speed.
  - Press the axis direction key and numeric key **5** together to move the axis at rapid jog speed.

1.2.1.3 Mode Buttons

They are used to switch among the following modes:

- 

: in this mode, the machine tool will automatically run the program file.
- 

: in this mode, it is convenient to return to the machine origin or set datum.
- 

: in this mode, the machine tool will keep running until you release the button.
- 

: in this mode, the machine tool will be controlled by handwheel.
- 

: in this mode, the machine tool only will move **0.001(mm)**, **0.01(mm)**, **0.1(mm)** or a customized step size.

The last three modes are referred as **Manual** mode.

1.2.1.4 E-stop Button

It is used to stop the machine tool completely when it is in danger by triggering the emergency alarm.

When the danger is cleared, you can remove the alarm by rotating the button in the clockwise direction.

1.2.1.5 Auxiliary Function Buttons

They are used to turn on or off the following frequently used ports:

- 

: used to turn on/off blowing.

- 

: used to turn on/off the lubricant.
- 

: used to turn on/off the light.
- 

: used to turn on/off the coolant.
- 

: not customized yet.
- 

~ 

: used to customize function.

1.2.1.6 Override Switches

They include the following switches:

- 
: used to adjust the spindle override form 50% ~ 150%.
- 
: used to adjust the feed override from 0% ~ 120%.

1.2.1.7 Movement Control Buttons

They are used to control the movement of the machine tool:

- 

/ 

: used to start / stop the rotation of the spindle.
- 

/ 

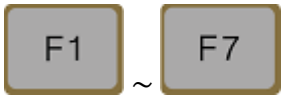
/ 

: used to control machining.

  See Execute machining for details.

1.2.1.8 Operational Buttons

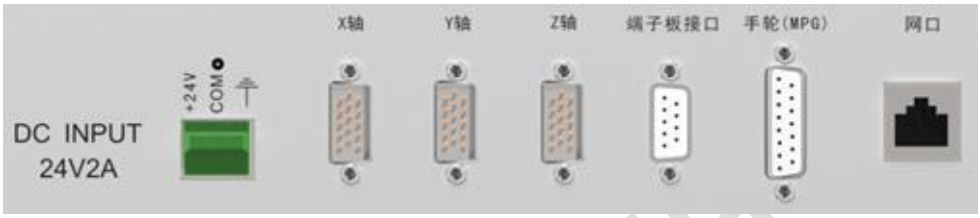
They are used to execute the corresponding operation in the display screen.



1.2.2 Rear View

The rear view of **NK260**, **NK280** and **NK280B** differs:

- **NK260**

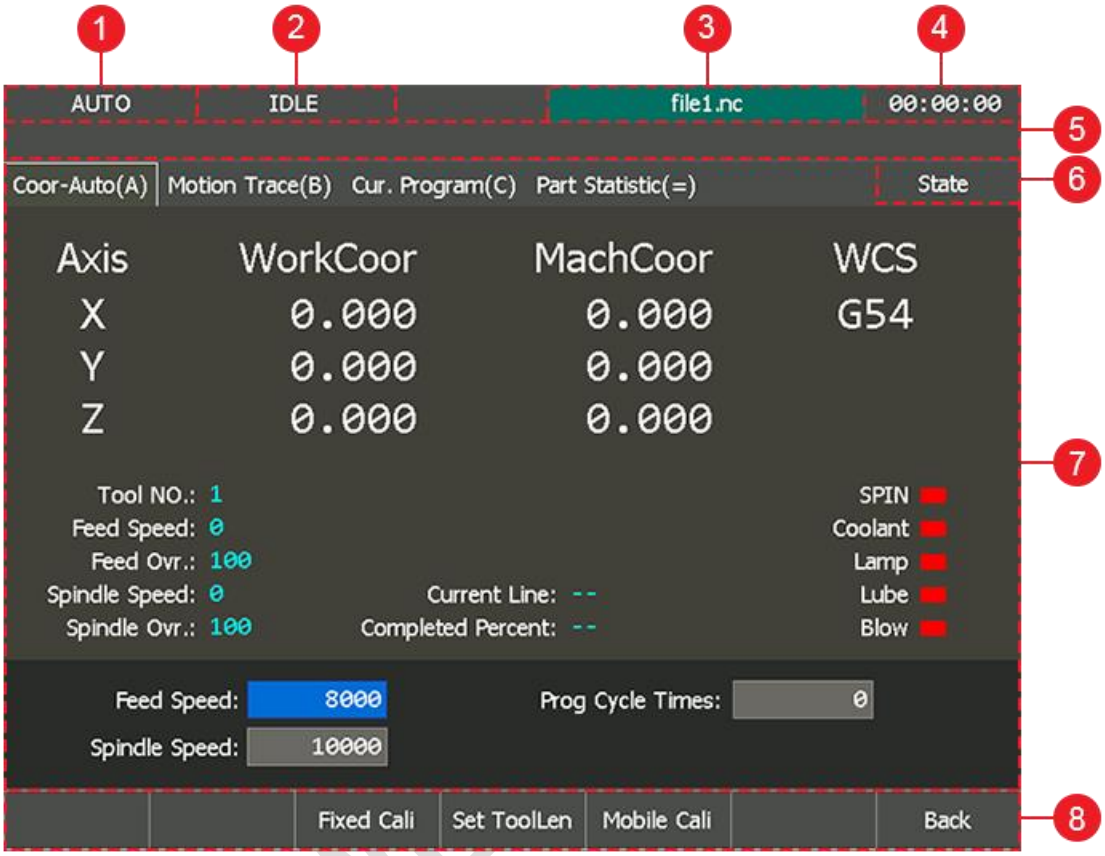


- **NK280**

- **NK280B**

1.3 Main Software Interface

The interactive operational interface of **NK260**, **NK280** and **NK280B** are almost the same:



1. Show the current mode, including **AUTO**, **JOG**, etc.
2. Show the current status of the machine tool, including **IDLE**, **RUNNING**, etc.
3. Show the name of the currently loaded program file or the currently loaded program task.
4. Show the machining time.
5. Show information such as working process or alarms.
6. Show the name of the current functional button.
7. Show child interfaces corresponding to each main function button.
8. Show commands corresponding to **F1** ~ **F7** on different operational interface.

2 Basic Commissioning

Through this section, you can quickly get familiar with the commissioning process about **NK260/NK280/NK280B**:

1. Switch the system configuration.
2. Adjust I/O port polarity.
3. Set commissioning parameters.
4. Check the axis direction.
5. Return to the machine origin.
6. Set speed parameters.

2.1 Switch the System Configuration

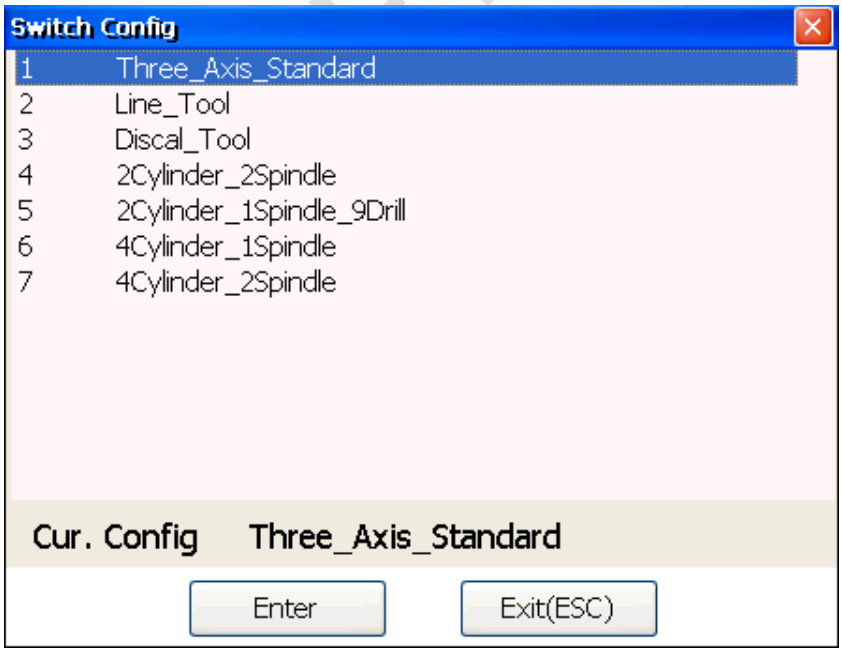
This operation is used to check if the default system configuration is what you need and change it according to the structure of the machine tool if not when the system is installed at the first time.

To switch the system configuration, do the following:

1. To enter into **System Info** interface, press 

→ **C**.

2. Press **F4**, and enter the manufacturer password. **Switch Config** dialog box pops up:



The current system configuration is shown in grey at the bottom of the dialog box.

3. **Optional:** If the current system configuration is not what you need, press ↑ / ↓ to move the cursor to the desired one, and press **Enter**.

Restart the software to validate the modification.

2.2 Adjust I/O Port Polarity

This operation is used to adjust polarities of input / output ports in the software in terms of the switch type, and clear alarms of I/O ports, so as to establish correct communication between the system and the drive:

- The polarity of normally closed switches should be **N**.
- The polarity of normally open switches should be **P**.

Except special customization, the polarities of output ports are **N** in general.

Before checking I/O ports, ensure the machine tool is connected well and powered on.

To adjust I/O port polarity, do the following:

1. To enter into **Port** interface, press 

→ **A**.

2. If the status of the following ports is different from the actual status of the machine tool, check the connection:
   - **ESTOP**
   - **Servo Alarm**
   - **Tool Calibration Signal of Z-axis**
   - **Mechanical Zero**
   - **Cycle Start**
   - **Cycle Stop**

   If the connection is not correct, tighten the connection; if it is, proceed to the next step.

3. Press ↑ / ↓ to select the target port, enter the manufacturer password, and click **Modify Polarity** to convert its polarity.

Restart the software to validate the modification.

In **Port** interface, you can also execute other operations about port. See Conduct a Simulation Test on Ports for details.

2.3 Set Commissioning Parameters

This operation is used to set parameters for commissioning, to avoid damage to the machine tool during its movement.

According to the type of control systems, it can be divided into the following:

- Set commissioning parameters for bus control system. (Exclusive to **NK280B**)
- Set commissioning parameters for non-bus control system.

2.3.1 Set Commissioning Parameters for Bus Control System

Before setting commissioning parameters for bus control system, ensure the following:

- Parameter **Control System Type** is set to **1**.
- The used terminal board is **Lambda 5M**.

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

1. To enter into **Machine Param** interface, press 

→ **A**.
2. To check manufacturer's parameters, press **F2**, and input the manufacturer password.
3. Find and set the following parameters:

   Drive Station Address
   It should match with toggle switch setting of the driver station address.
   Number of each drive station address should be unique, such as X-axis is set 1, Y-axis is set to 2, and Z-axis is set to 3…
   0 is invalid.

   Encoder Digit
   The encoder digit of a servo motor.

   Lead Screw Pitch
   It refers to the axial distance between the corresponding points of two adjacent teeth on the threads.

   TravelLimits-Positive / TravelLimits-Negative
   The movement range of each axis in X-axis, Y-axis and Z-axis direction.
   The system will carry out soft limit in terms of this range in order to protect the machine tool.

Electronic Gear Ratio
It refers to the ratio that the servo enlarges or shrinks the received pulse frequency.
It consists of parameter **Numerator of Electronic Gear Ratio** and **Denominator of Electronic Gear Ratio**.
B represents its numerator, while A represents its denominator.
If it is greater than 1, the servo enlarges the received pulse frequency; if it is less than 1, the servo shrinks the received pulse frequency.
It setting should match with the value of drive parameter **Electronic Gear Ratio**.
It is 1:1 by default.

$$\frac{B}{A} = \frac{F * p}{d} * \frac{m}{n}$$

F stands for encoder resolution; p stands for pulse equivalent; d stands for pitch; m/n stands for mechanical reducer ratio.

2.3.2 Set Commissioning Parameters for Non-bus Control System

For **NK280B**, before setting commissioning parameters for non-bus control system, ensure the following:

- Parameter **Control System Type** is set to **0**.
- Parameter **Encoder Type** is set to **0**.

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

1. To enter into **Machine Param** interface, press 

→ **A**.

2. To check manufacturer's parameters, press **F2**, and input manufacturer password.

3. Press **F2**, find and set the following axis parameters according to the actual situation:

   Pulse Equivalent
   It refers to the moving distance of screw or rotation degree of the 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; while larger value gets faster maximum feedrate.

   TravelLimits-Positive / TravelLimits-Negative
   The movement range of each axis in X-axis, Y-axis and Z-axis direction.

   The system will carry out soft limit in terms of this range in order to protect the machine tool.

4. Press **F4**, find and set the following origin parameters according to the actual situation:

   Motor Type
   The motor type.
   1: Stepping motor. 2: Servo motor.

   REF Switch Positioning Direction
   In REF switch positioning phase, the moving direction of each axis during returning to the machine origin.

   REF Switch Positioning Speed
   In REF switch positioning phase, the feedrate of each axis during returning to the machine origin.

   REF Encoder Positioning Speed
   In encoder positioning phase, the feedrate of {AXIS}-axis during returning to the machine origin.

   Lead Screw Pitch
   It refers to the axial distance between the corresponding points of two adjacent teeth on the threads.

   Back Distance
   After fine positioning stage of returning to the machine origin, the additional displacement of each axis.
   +: Positive direction; -: Negative direction.

   Coarse/Fine_Switches_Min_Dist
   Used to see if coarse and fine switches are too close.
   Valid range: [0, thread pitch/2].

2.4 Check the Axis Direction

This operation is used to ensure that the axis direction is the same with the direction stipulated by **Right Hand Rule**, to avoid damage to the machine tool due to incorrect direction.

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

1. Judge the positive direction of X-axis according to the **Right Hand Rule**.
2. To switch to **Manual** mode, press 

/ 

/ 

.
3. Press **4** / **6** or control the handwheel to move X-axis, and observe its moving direction.

4. **Optional:** If the moving direction is opposite to the judged direction, modify the setting value of parameter **Axis Direction(X)** to the opposite value.

2.5 Return to the Machine Origin

According to the encoder type, this operation can be divided into:

- Set Datum with an absolute encoder.
- Return to the machine origin with an incremental encoder.

2.5.1 Set Datum with an Absolute Encoder

This operation is used to return to the machine origin with an absolute encoder by directly setting datum when it is your first time to use the system. And it owns the following advantages compared to returning to the machine origin with an incremental encoder:

- No need to set returning orders for all axes in datum setting process.
- No need to set datum again after restarting the system, recovering from power interruption and E-stop because the system will automatically read the datum information.

Note: You need to set datum again when the absolute encoder is out of battery.

Before setting datum, ensure the following:

- Hardware devices have been well connected.
- The axis direction is correct. See Check the Axis Direction for details.
- Parameter **Enable Encoder Feedback** is set to **Yes**.
- Parameter **Encoder Type** is set to **1**.

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

1. To enter into **BACKREF** mode, press 

.
2. To enter into **Coor-Reference** interface, press 

→ **A**.
3. To enter into **Datum Setting** page, press **F5**.
4. Press **4** / **6**, and move X-axis to a fixed position.
5. Press **F1**. The system automatically reads X-axis position and sets it as X-axis datum.

After setting datum successfully, the sign appears in front of X-axis in coordinate display area.

Repeat the above steps, set datum for other axes, and restart the software to validate the setting.

After setting datum for all axes, to avoid datum loses after updating the software or during transporting machine tools, press **F6** to export datum to a USB flash disk for later use.

2.5.2 Return to the Machine Origin with an Incremental Encoder

Origin of Machine Coordinate System (MCS), also called machine origin, and home, is a fixed point assigned by design, manufacturing and debugging before the machine tool leaves factory. This operation is used to return to the point.

This operation is required each time you restart the system.

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

- Hardware devices have been well connected.
- The axis direction is correct. See Check the Axis Direction for details.

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

1. To enter into **BACKREF** mode, press 

.
2. To enter into **Coor-Reference** mode, press 

→ **A**.
3. Do one of the following:
   - Press **F7**. Z-axis returns to the machine origin firstly, then other axes return.
   - Press **F1** / **F2** / **F3**. The specified axis returns to the machine origin.

     For safety, it is suggested that Z-axis returns to the machine origin firstly.

After returning, the sign appears in front of related axes in coordinate display area.

2.6Set Speed Parameters

This operation is used to set parameters about speed for commissioning to get a good machining effect.

To set speed parameters, do the following:

1. To enter into **Machine Param** interface, press 

→ **A**.

2. Press **F2**, and input the manufacturer password.

3. Press **F1**, find and set the following operation parameters:

   JOG Jerk
   Jerk under rapid-jog mode.

   JOG Feedrate before Ref.
   The default maximum feedrate in jog mode before returning to the machine origin.

   Manual High
   The speed under rapid-jog mode.

   Manual Low
   The default speed under jog mode.

   Max Spindle Speed
   The maximum rotational speed of spindle.
   It should be consistent with the settings of inverter.

   Handwheel Acceleration
   Handwheel acceleration.
   Smaller value gets smoother handwheel movements.

4. Press **F3**, find and set the following program parameters:

   Feedrate
   The default feedrate during machining.

   Machining Acceleration
   The maximum acceleration for machining.

   Max Speed of Ref. Circle
   The maximal allowable speed of reference circle with 10mm diameter.

   Max Acceleration at Corners
   The maximum acceleration at corners.

   Machining Jerk
   Machining Jerk.
   It is only valid on GXX.

Maximum Axial Feedrate
The default maximum feedrate of each axis during machining

Maximum Axial Rapid Traverse Speed
The maximum speed of each axis during positioning.

Max Axial Speed
The maximum speed of each axis.

Max Axial Machining Acceleration
The maximum acceleration of each axis during machining.

Max Axial Rapid Traverse Acceleration
The maximum acceleration of each axis during positioning.

Max Axial Jerk
The maximum axial jerk.

Startup Speed
The initial also the minimum speed during machining.

Circular Processing Min Speed
The minimum speed during machining an arc.

Max Acceleration at Feed Override Change
When feed override changes, smaller max acceleration gets smoother movements.

Approaching F
During rapid positioning, the feedrate when the tool is approaching the workpiece.

5. Press **F5**, find and set the following parameters about tool change:

Rapid Traverse Speed in Tool Change
The rapid traverse speed during tool change.

Z-axis Speed in Tool Change
The default speed at upper and lower position of Z-axis during tool change.

Horizontal Speed when Tool In/Out Tool Mag.
The default speed of Z-axis moving into/out of tool magazine during tool change.

3 Quick Start

Through this section, you can quickly know how to use **NK260/NK280/NK280B**.

Machining processes include the following:

1. Return to the machine origin.
2. Load a program file.
3. Execute fixed calibration.
4. Set the workpiece origin.
5. Execute machining.

3.1 Return to the Machine Origin

This operation is used to adjust the coordinate system before machining.

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

- For an absolute encoder, skip the operation.

  See Set Datum with an Absolute Encoder for the reason.

- For an incremental encoder, return to the machine origin with an incremental encoder.

  See Return to the Machine Origin with an Incremental Encoder for details.

3.2 Load a Program File

This operation is used to load a program file stored in the host or a USB flash disk for machining.

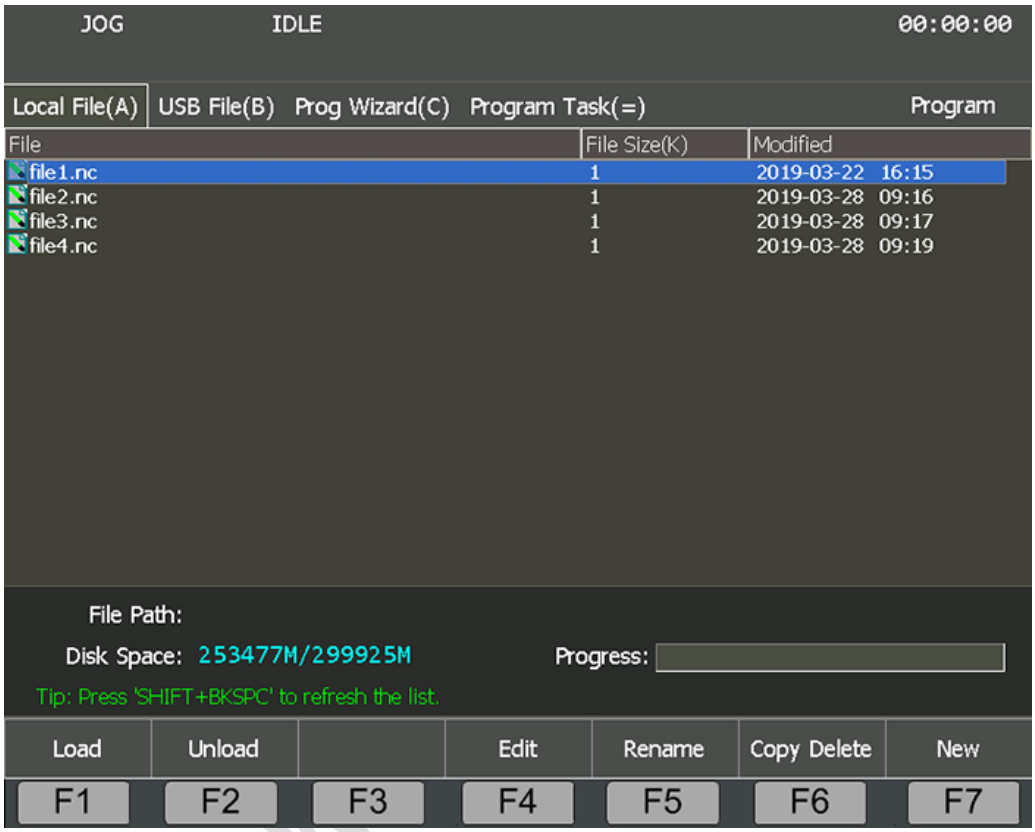
Before loading a program file, do one of the following to prepare a program file:

- Put a program file in the host or a USB flash disk.
- Press **F7** in **Local File** interface, create a program file, and press **F1** to save it.

Taking loading a program file in the local as an example, to load a program file, do the following:

1. To enter into **Local File** interface, press 

→ **A**:



2. Press ↑ / ↓, and select the target program file.
3. Press **F1**. The system automatically loads the file.

After finishing, the file name automatically shows on the top of the screen.

If you need to unload the file, press **F2**.

You can also load a program file generated by wizard or load a program task.

See Generate a Wizard File and Execute the Program Task for details.

3.3 Execute Fixed Calibration

This operation is used to measure the tool on a certain fixed position of the machine tool to reconfirm tool offset, so as to avoid tool length and the clamping position vary during calibration due to tool damage or other causes. It is mainly used for multi-tool mode and mainly used in machine tool with tool magazine.

Before executing fixed calibration, do the following:

1. Return to the machine origin.
2. Set the value of parameter **Fixed Presetter Position** according to the actual situation.

To execute fixed calibration, select a tool and do the following:

1. To enter into **AUTO** / **JOG** / **HW** mode, press 

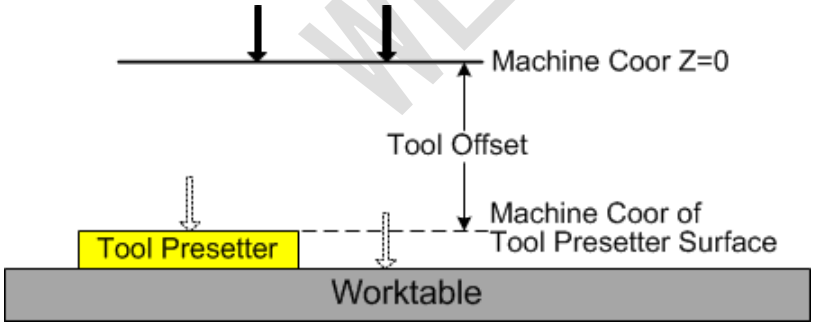
/ 

/ 

.
2. To enter into **Coor-Auto** / **Coor-Manual** interface, press 

→ **1**.
3. To start fixed calibration, press **F5** → **F3**.

The system automatically records the machine coordinate when the tool nose touches the surface of the tool presetter, and sets the recorded machine coordinate to the tool offset:




```
Tool offset = Machine coordinate - Machine coordinate of tool presetter
 surface
```

3.4Set the Workpiece Origin

This operation is used to define the origin of the workpiece coordinate system (WCS), that is, the workpiece origin.

It can be divided into the following according to the actual situation:

- Do centering.
- Do clearing.
- Execute mobile calibration.
- Set the offsets.

3.4.1 Do Centering

This operation is used to set the workpiece origin for regular-shaped workpieces.

It can be divided into the following according to the workpiece shape:

- Line centering: used to find the center point by two points on regular rectangular workpieces.
- Circle centering:used to find the center point by three points on circular workpieces.

Taking X-axis and line centering as an example, to do centering, do the following:

1. To enter into **Manual** mode, press 

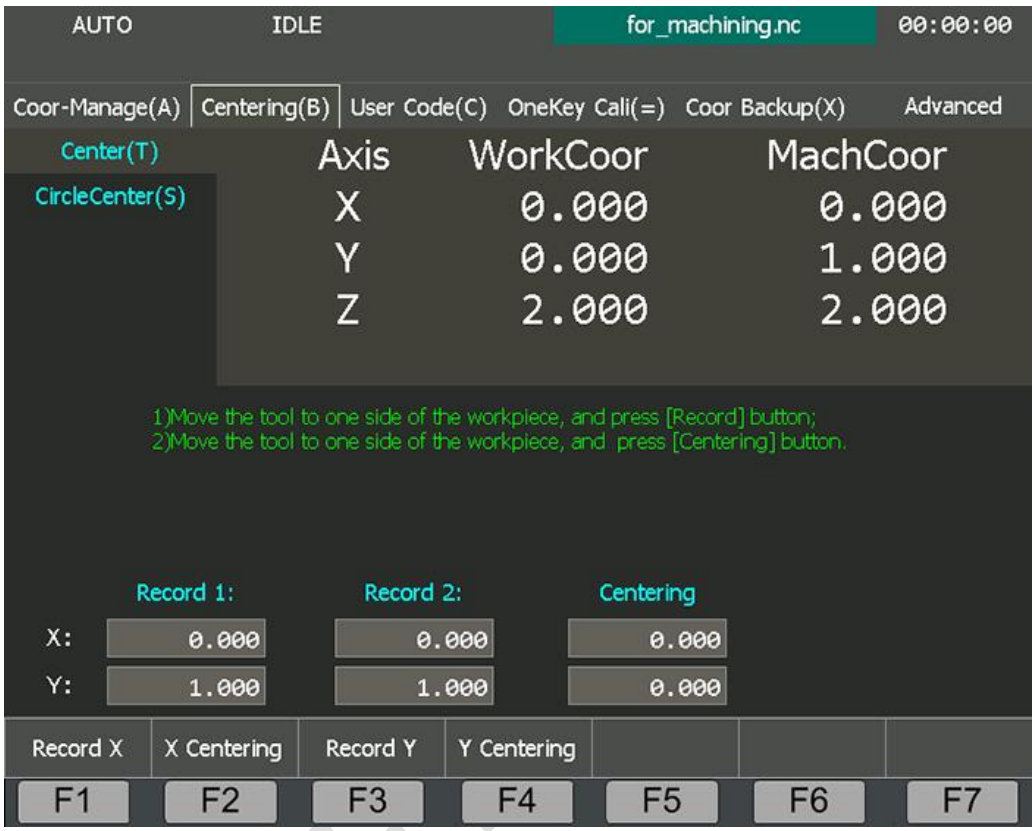
/ 

/ 

.

2. To enter into **Centering** interface, press 

→ **B**:



3. To enter into **Center** page for line centering, press **T**.

4. Press **4** / **6**, move X-axis to one side of the workpiece, and press **F1**. The system automatically records the machine coordinate of the current point.

5. Press **4** / **6**, move X-axis to the other side of the workpiece, and press **F2**. The system automatically calculates the midpoint coordinate based on the coordinate of the current position and records in the last step, and sets it as the workpiece origin.

3.4.2 Do Clearing

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

To do clearing, do the following:

1. To enter into **Coor-Auto** interface, press 

→ **A**.
2. Press **F6**, and select the target axis to do clearing:
   - **F1:** to clear the workpiece coordinates of X-axis.
   - **F2:** to clear the workpiece coordinates of Y-axis.
   - **F3:** to clear the workpiece coordinates of X-axis and Y-axis.
   - **F4:** to clear the workpiece coordinates of Z-axis.
   - **F5:** to clear the workpiece coordinates of all axes.

3.4.3 Execute Mobile Calibration

This operation is used to set the workpiece origin of Z-axis by executing calibration at the current position.

Before executing mobile calibration, do the following:

1. Return to the machine origin.
2. Set the value of parameter **Mobile Presetter Thickness**:
   1. Manually move Z-axis to a certain point over workpiece surface, shift down the tool nose until reaching the workpiece surface, and record the current coordinate of Z-axis (Z1).
   2. Uplift Z-axis, put a tool presetter on the workpiece surface, shift down Z-axis slowly until reaching the presetter and getting the tool presetter signal, and record the current coordinate of Z-axis (Z2).
   3. Get the result of Z2-axis minus Z1-axis, and set it as the value of parameter **Mobile Presetter Thickness**.

To execute mobile calibration, select a tool and do the following:

1. To enter into **AUTO** / **JOG** / **HW** mode, press 

/ 

/ 

.

2. To enter into **Coor-Auto / Coor-Manual** interface, press 

→ **A**.

3. To start mobile calibration, press **F5** → **F5**.

The system calculates calibration result and set it into the workpiece offset:

```
Workpiece offset = Machine coordinate - Mobile presetter thickness - Pu
blic offset - Tool offset
```

Generally, the default setting values of public offset and tool offset are both **0**.

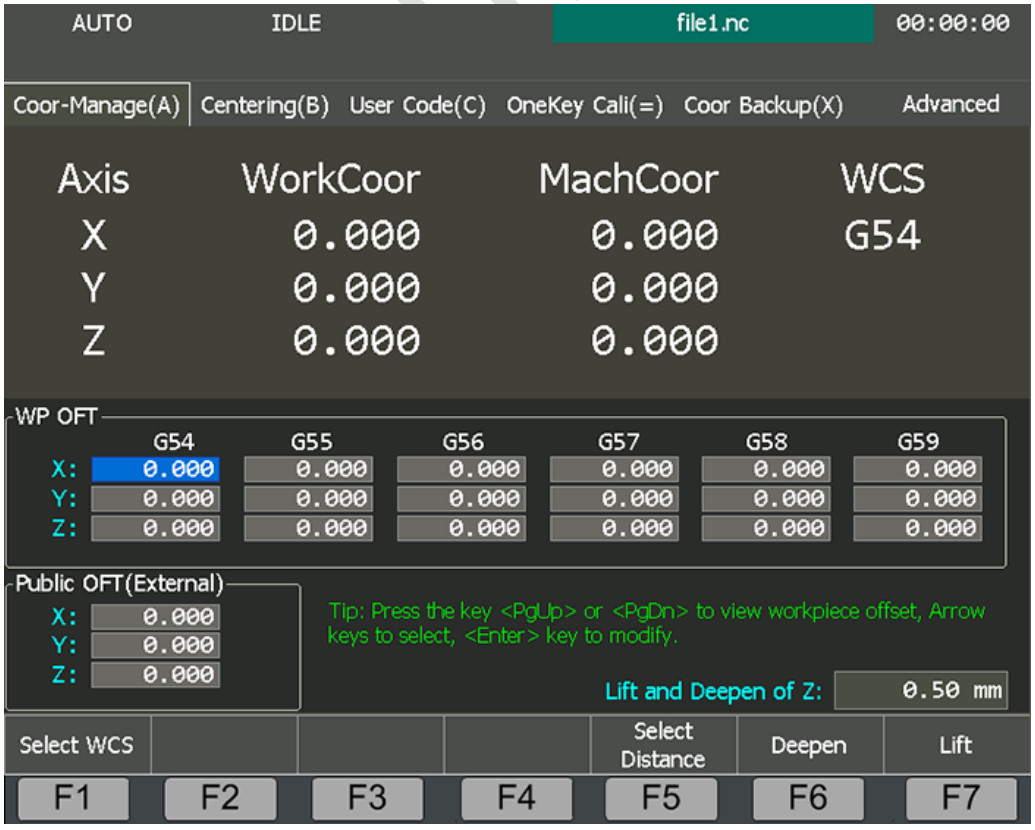
3.4.4 Set the Offsets

This operation is used to set the workpiece offset and the public offset. The former is used to show the distance of the workpiece origin relative to the machine origin, while the latter is used to adjust the workpiece origin of X-axis, Y-axis and Z-axis for all WCSs.

To set the offsets, do the following:

1. To enter into **Coor-Manage** interface, press 

→ **A**:



2. **Optional:** To select the target WCS, do the following:
    1. Press ← / →, and move the cursor to the coordinate column.

       Its color turns into light blue.
    2. Press **F1** to select the coordinate system.

       The WCS name shows in the **WCS** column of the coordinate display area.
3. To set the workpiece offset, do the following:
    1. Press ↑ / ↓ / ← / →, and select the target input box.
    2. Press **Enter**, and input the workpiece offset.
4. To set the public offset, do the following:
    1. Press ↑ / ↓ / ← / →, and select the target input box.
    2. Press **Enter**, and input the workpiece offset.
    3. To adjust Z-axis public offset, press **F5**, set adjusted distance, and press **F6** / **F7** to deepen (reduce) / lift (increase) the public offset.

3.5 Execute Machining

This operation is used to run the program from the beginning to the end.

To execute machining, do the following:

1. To enter into **AUTO** mode, press 

.
2. To start machining, press 

.

The system automatically switches to **Coor-Auto** interface and executes the program file until the end of block or any intentional interruption.

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

- To stop machining in advance, press 

.
- To resume machining from the exact interrupted position when power interruption or E-stop occurs and the workpiece origin is secured, press 

.

4 Machining Operations

Through this section, you can mainly know the following machining related operations:

- Execute selective machining.
- Generate a wizard file.
- Execute cycle machining.
- Execute array machining.
- Execute the program task.
- Simulate machining.
- Execute MDI.
- Guide machining by the handwheel.

4.1 Execute Selective Machining

This operation is used for an optional skip of the program file, so that you can freely select the machining range.

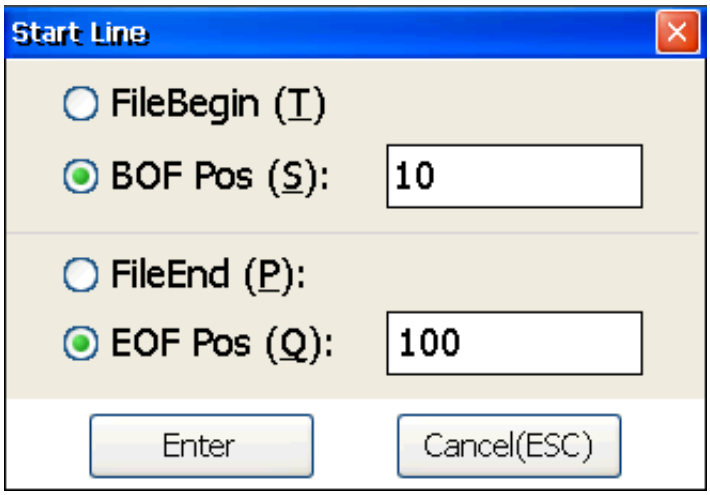
To execute selective machining, do the following:

1. To enter into **AUTO** mode, press 

.
2. To open **Start Line** dialog box, do one of the following:
   - Press 

→ **A** → **F2**.
   - Press 

→ **C** → **F1**.



3. To start machining according to the actual demand, select the start line:
   - To start machining from the beginning of the program file, press **T**.
   - To start machining from the set line, press **S** and manually input the start line.
4. To stop machining according to the actual demand, select the end line:
   - To stop machining till the end of the program file, press **P**.
   - To stop machining till the set line, press **Q**, and manually input the end line.

The system automatically starts machining from the set start line to the set end line.

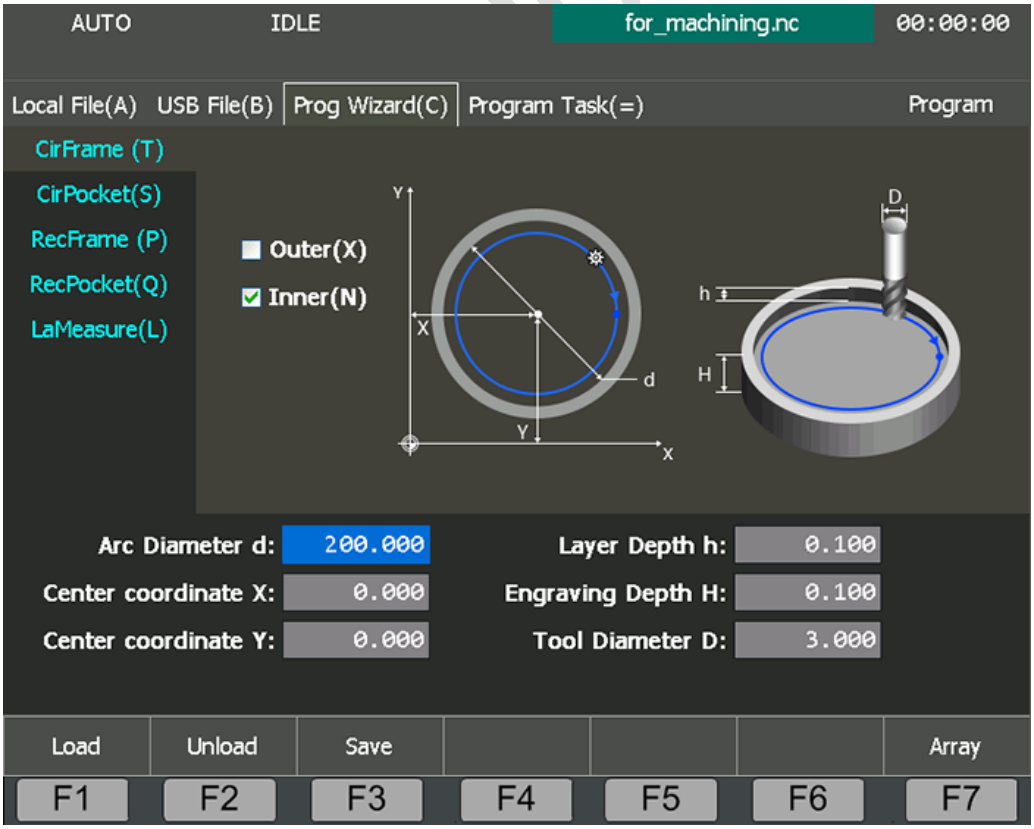
4.2 Generate a Wizard File

This operation is used to generate a wizard file by program wizard.

To generate a wizard file, do the following:

1. To enter into **Prog Wizard** interface, press 

→ **C**:



2. Select a kind of wizard:
   - **T:** CirFrame
   - **S:** CirPocket
   - **P:** RecFrame
   - **Q:** RecPocket
   - **L:** LaMeasure
3. Set related parameters.
4. To generate a program file and save it, press **F3**.
5. To load the program file, press **F1**.

If you need to unload the file, press **F2**.

4.3 Execute Cycle Machining

This operation is used to complete all machining actions with a program file for batch machining, carving machining, etc.

Before executing cycle machining, do the following:

1. Return to the machine origin.
2. Load a program file.

To execute cycle machining, do the following:

1. To enter into **AUTO** mode, press 

.
2. To enter to **Coor-Auto** interface, press 

→ **A**.
3. To set machining times of the current program file, set the value of parameter **Prog Cycle Times**.
4. Find Parameter **Cycle Machining Interval**, and set the interval between two machining tasks for cycle machining.

Note: After restating the software or reloading the program file, cycle times will be cleared and you need to set it again.

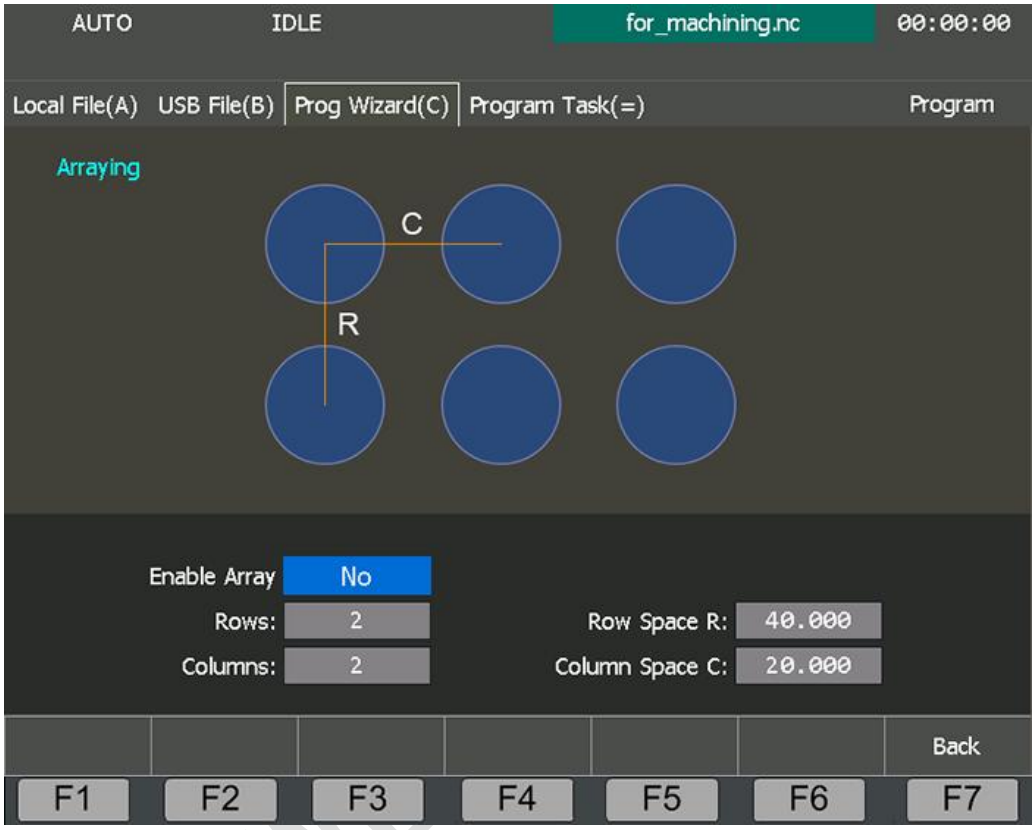
4.4 Execute Array Machining

This operation is used for batch machining of the currently loaded program file, so as to improve machining efficiency.

To execute array machining, do the following:

1. To enter into **Arraying** page, press 

→ **C** → **F7**:



2. To enable array machining, move the cursor to the input box of **Enable Array**, press **Enter**, and press **1**.

3. Press ↑ / ↓ / ← / → to move the cursor to the related input box, and set a value.

   Rows
   The number of rows for array machining.

   Columns
   The number of columns for array machining.

   Row Space
   The space between each row.

   Column Space
   The space between each column.

When machining starts, the system pops up a dialog box to confirm whether to enable array machining, and starts array machining after your confirmation.

4.5 Execute the Program Task

This operation is used to put several program files into a task by setting machining order, machining interval, used WCS and machining times for each program file.

Before executing the program task, ensure that there are several program files in the local or USB flash disk.

To execute the program task, do the following:

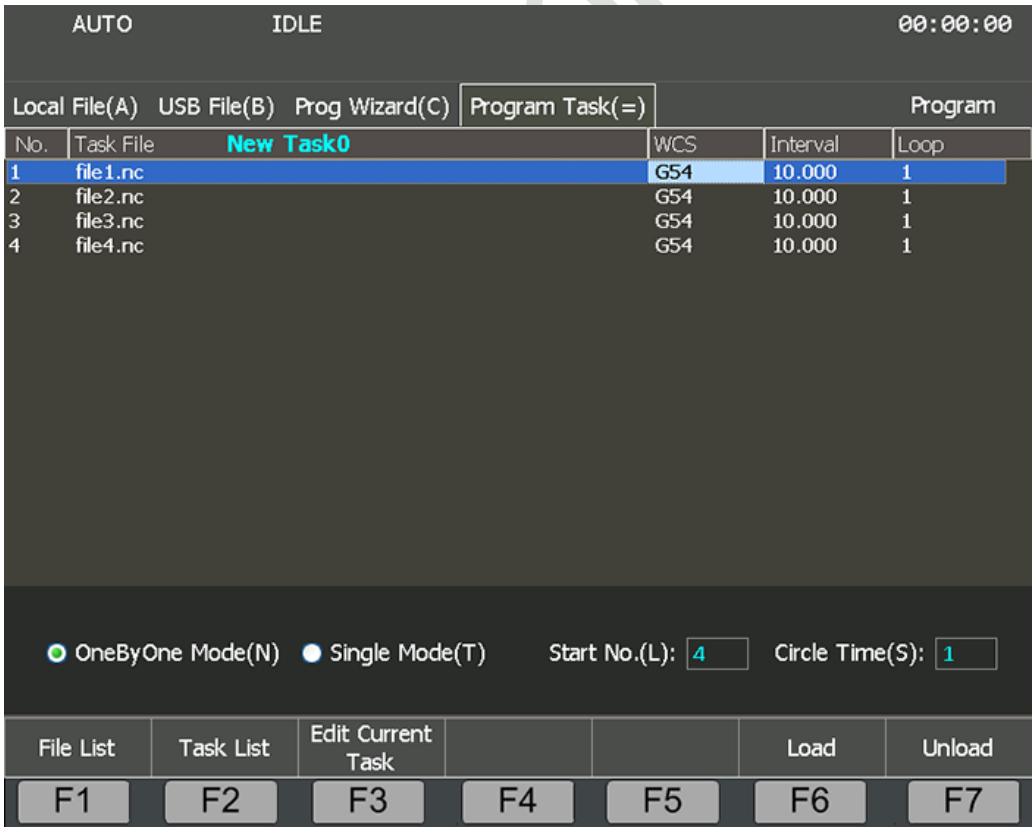
1. Create a program task.
2. Edit the program task.
3. Start the program task.

4.5.1 Create a Program Task

To create a new program task, do the following:

1. To enter into **Program Task** interface, press 

→ **=**:



2. To create a new program task, press **F2** → **F1**. A program task named as `New Task + No.` shows.

3. **Optional:** To rename the program task, press **F3**.
4. To open the program task for later editing, press **F6**.
5. To enter into **File List** page, press **F7** → **F1**.
6. **Optional:** To add program files in the USB flash disk into the program task, press **F6**.

   It is default to add local program files.
7. Press ↑ / ↓ and **F2** / **F3** / **F4** to select the target file, and press **F1** to add it into the program task.

Press **F7** to exit creating the program task.

4.5.2 Edit the Program Task

To edit the program task, do the following:

1. Press ↑ / ↓ / ← / → to select the target input box, press **Enter** and modify the following:
   - Used WCS
   - Machining interval
   - Machining times
2. Press **F3**, press ↑ / ↓, and do one of the following:
   - To modify the machining order in the program task, press **F1** / **F2**.
   - To remove the target program file from the program task, **F3**.
3. To save the modification, press **F6**.

4.5.3 Start the Program Task

To execute the program task, do the following:

1. To select a machining mode, do the following:
   - To select **One by One Mode**, press **N**.
   - To select **Single Mode**, press **T**.
2. **Optional:** To set cycle times for the program task, press **S**.
3. To load the program task, press **F6**. Names of the program task and the first program file in the task show above the display screen.
4. **Optional:** To set the first program file, press **L**.

5. To start machining the program task, press 

.

The system automatically does the following according to the set machining order:

1. Load the first program file, start to machine it, and unload it when machining ends.
2. Do one of the following:
   - In **One by One Mode**, load the second program file, and wait for the set interval, start to machine the program file, and unload it when machining ends, repeat these steps until all program files in the program task have been machined.
   - In **Single Mode**, load the second program file, and enter into **Pause** status.

     It will not start to machine the second program file unless you press 

manually.

4.6Simulate Machining

This operation is used to provide a fast but lifelike simulation machining environment to see the moving from of the machine tool in advance, so as to avoid damages to the machine tool due to programming mistakes.

During simulating machining, no actual machining occurs.

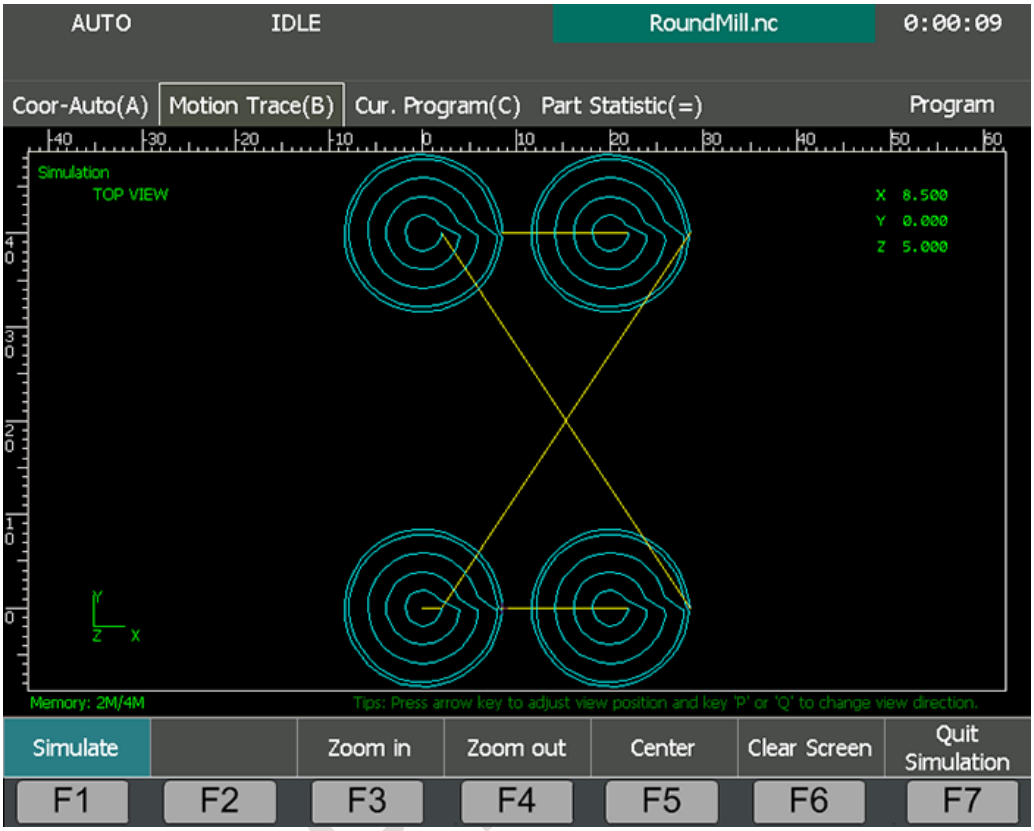
To simulate machining, do the following:

1. To enter into **AUTO** mode, press 

.

2. To enter into **Motion Trace** interface, press 

→ **B**:



3. To start simulating machining, press **F1**.

The moving trace of the machine tool shows in **Motion Trace** interface.

During simulating machining, you can also do the following:

- Press **F3** / **F4** / **F5** to zoom in / zoom out / center the moving trace.
- Press **F6** to clear the moving trace.
- Press **F7** to stop simulating machining.

4.7 Execute MDI

This operation is used to directly enter at most 7 commands to control the system, so as to perform fast movements, change system status or execute simple machining.

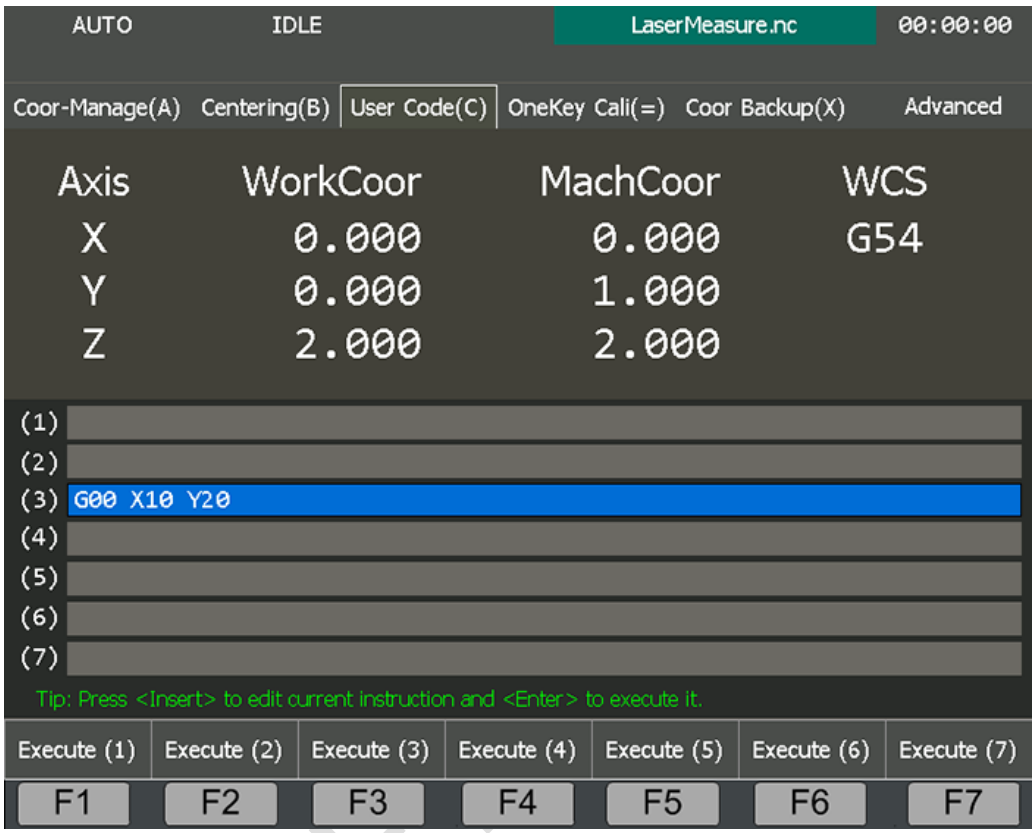
To execute MDI, do the following:

1. To enter into **AUTO** mode, press 

.

2. To enter into **User Code** interface, press 

→ **C**:



3. To move the cursor to the target line, e.g. line (3), press ↑ / ↓.
4. Press **Insert**, and input a command.
5. To execute the command, do one of the following :
   - Press **F3**.
   - Press **Enter**.

4.8Guide Machining by the Handwheel

This operation is used to manually control the automatic execution speed of the machining program during automatic machining, so as to guard against dangers caused by a wrongly loaded program or an inappropriate toolpath.

Before guiding machining by the handwheel, connect the handwheel.

To guide machining by the handwheel, do the following:

1. To enter into **AUTO** mode, press 

.
2. To enter into **Coor-Auto** interface, press 

→ **A**.
3. To enable handwheel guide, press **F1**.
4. To start machining by the handwheel, press 

.
5. Turn the handwheel in the clockwise direction.

The system starts machining and stops machining within 300ms until you stop turning the handwheel.

Machining speed varies with the turning speed of the handwheel.

5 Common Operations

This section introduces the following common operations:

- Conduct a simulation test on ports.
- Modify system parameters.
- Back up system parameters.
- Check system logs.
- Save and recover coordinates.
- Check machining statistics.
- Compensate the tool.
- Compensate screw error.
- Execute instant calibration.

5.1 Conduct a Simulation Test on Ports

This operation is used to conduct a simulation test on input ports and output ports to simulate hardware signal. And it helps to judge whether a port has output depending on the status of the indicator light.

Indicator status of input ports and output ports for simulation is as follows:

- Input ports: ● means having a signal; ● means no signal.
- Output ports: ● means having a signal; ● means no signal.

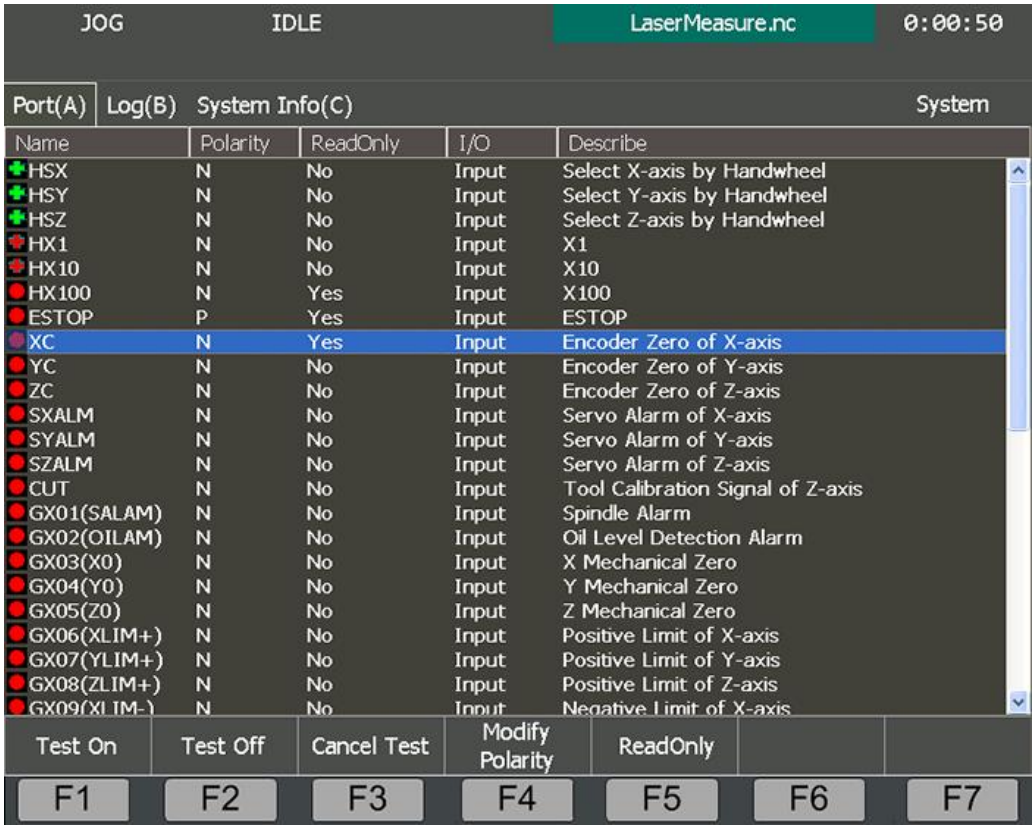
Indicator status is also connected with port polarity. See Adjust I/O Port Polarity for details.

Before conducting a simulation test, ensure that **Read-only** attribute for the select port is **No**. You can press **F5** in **Port** interface to change the read-only attribute.

To conduct a simulation test on ports, do the following:

1. To enter into **Port** interface, press 

→ **A**:



2. To make the indicator light before the selected port shift between ✚ (test on) and ✚ (test off), press **F1** or **F2**.

To exit the simulation test on all ports, press **F3**.

5.2 Modify System Parameters

This operation is used to tell how to change system parameters in **Machine Param** interface.

According to user' roles and permission, system parameters can be divided into:

- User's parameters: default parameters
- Manufacturer's parameters: manufacturer password is required.
- Developer's parameters: developer password is required.

To modify system parameters, do the following:

1. To enter into **Machine Param** interface, press 

→ **A**.
2. To find the target parameters, press ↑ / ↓.
3. Press **Enter**, and input a value according to your need.
4. **Optional:** Do one of the following according to valid time of these modified parameters:
   - **Now:** skip this step.
   - **Restart:** restart the system to make modification effective.
   - **Reload:** reload the program file to make modification effective.

After system parameters have been modified well, back up system parameters. See Back up System Parameters for details.

5.3 Back up System Parameters

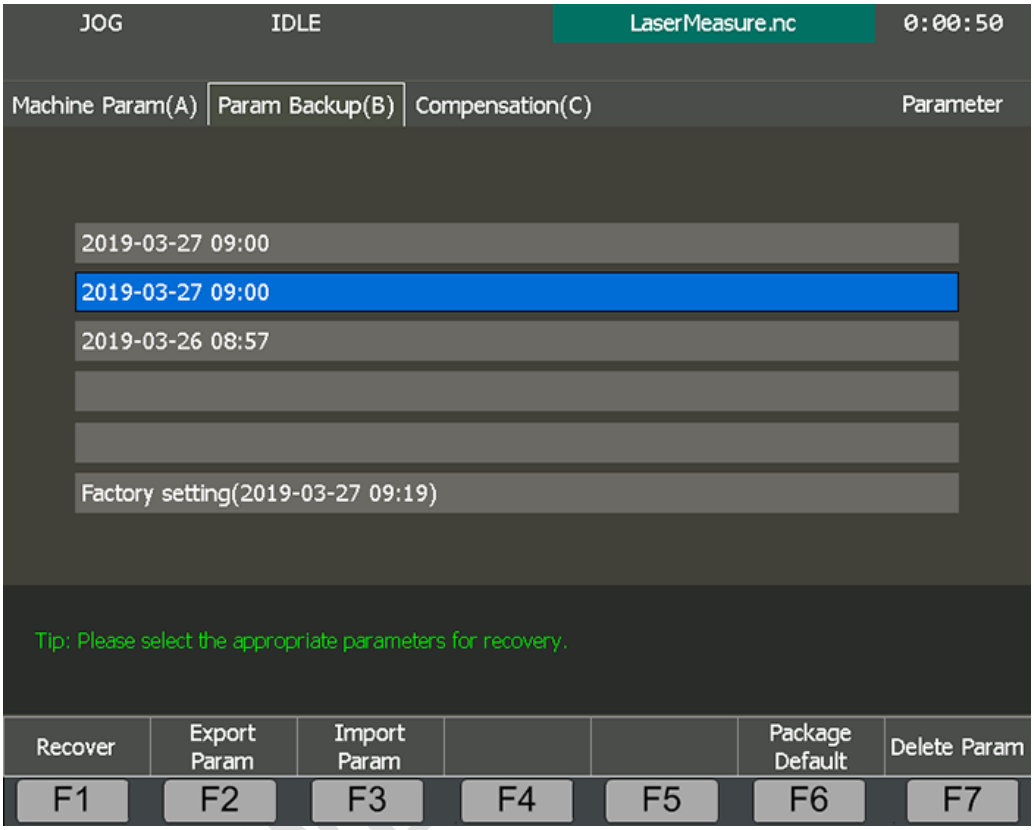
This operation is used to prepare a second copy of the parameter settings to restore the settings in the future or for other CNC integrated systems, so as to save the trouble of setting them again.

Before backing up system parameters, ensure that system parameters have been set well. For how to set system parameters, see Modify System Parameters for details.

To back up system parameters, do the following:

1. To enter into **Param Backup** interface, press 

→ **B**:



2. To back up current parameter settings to factory setting, press **F6**.
3. To export parameter settings to a USB flash disk, press **F2**.

   After restarting the system, the exported time shows in **Param Backup** interface.

In **Param Backup** interface, you can also do the following:

- To import a parameter file in a USB flash disk into the system, press **F3**.
- To recover parameter setting to a desired backup time, press ↑ / ↓ to select the desired time, and press **F1**.
- To delete backup parameters, press **F7**. The system deletes backup time and backup parameter settings.

5.4Check System Logs

This operation is used to view important operations and system events since this time start-up and in history for troubleshooting.

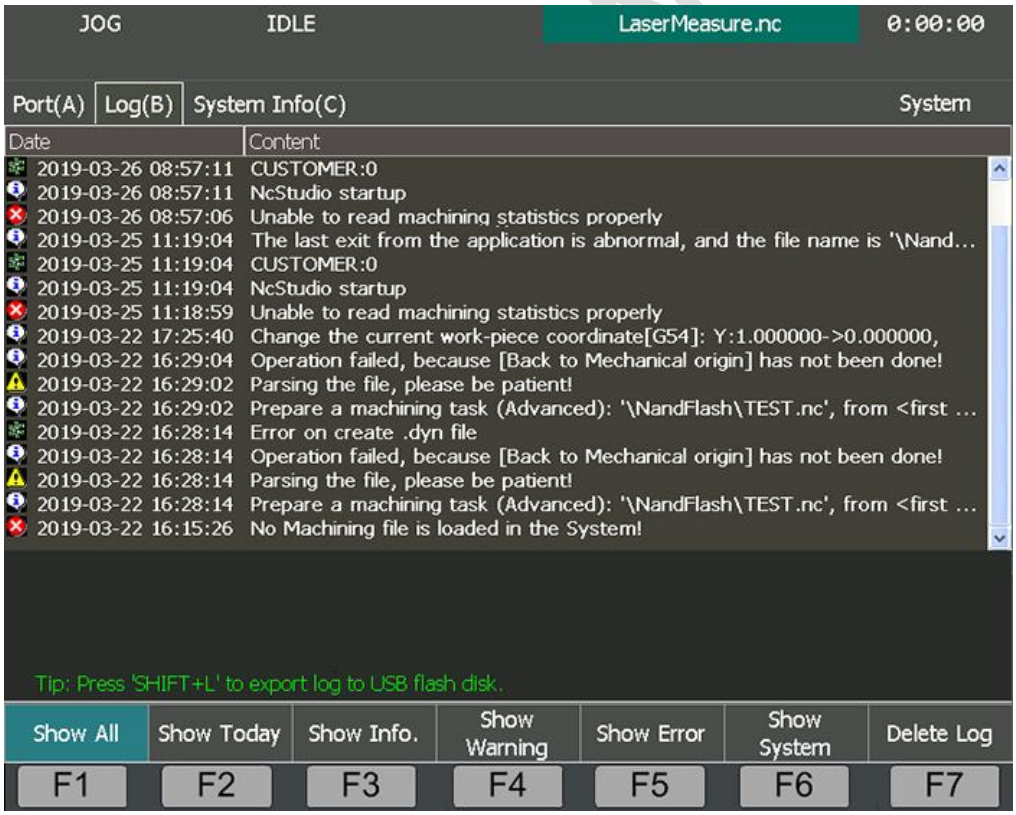
System logs include the following types:

- System startup and shutdown
- Start and stop of automatic machining
- Change of workpiece offsets
- System alarms
- Prompts about machining
- Other system information

To check system logs, do the following:

1. To enter into **Log** interface, press 

→ **B**:



2. Check different types of system logs:
   - To check all logs, press **F1**. It is pressed by default.
   - To check today's logs, press **F2**.

- To check logs about running status, press **F3**. Logs with icon show in **Log** interface.
- To check warning logs, press **F4**. Logs with icon show in **Log** interface.
- To check error logs, press **F5**. Logs with icon show in **Log** interface.
- To check system logs, press **F6**. Logs with icon show in **Log** interface.

You can also do the following in **Log** interface:

- To delete all system logs, press **F7**.

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

- To export system logs to a USB flash disk, press **shift** and **L**.

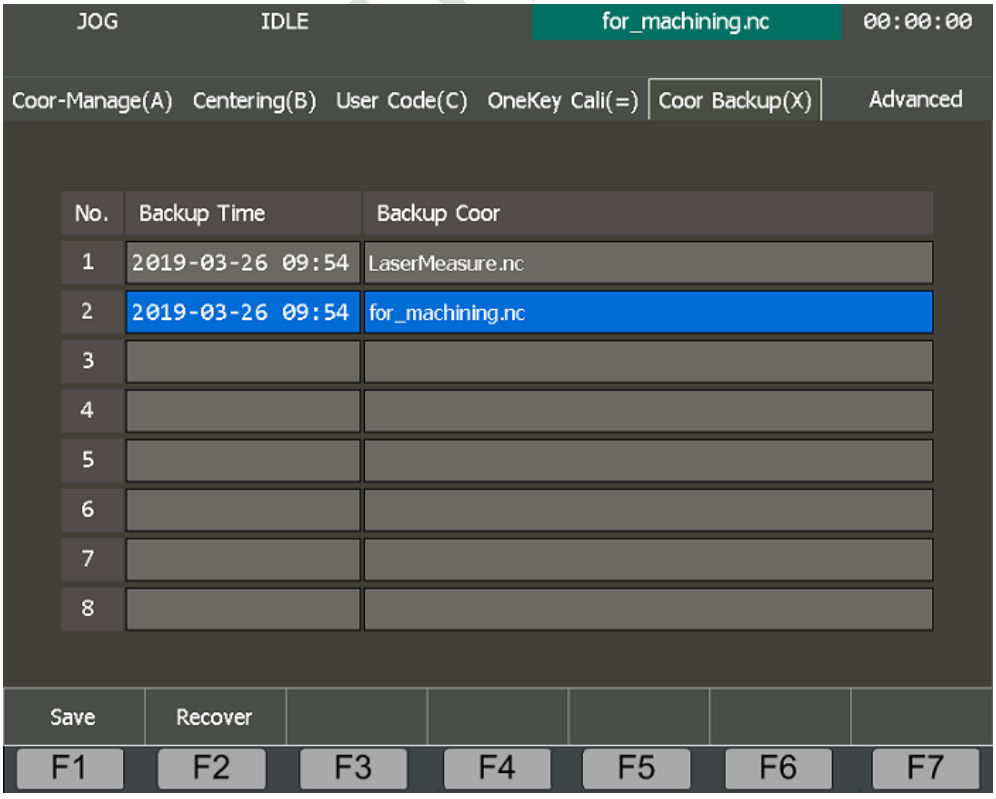
5.5 Save and Recover Coordinates

This operation is used to keep the workpiece offsets for later use and get back the previously saved workpiece offsets to save the trouble of repeatedly setting the workpiece offsets.

To save and recover coordinates, do the following:

1. To enter into **Coor Backup** interface, press 

→ **X**:



2. To save coordinates, do the following:

    1. Press ↑ / ↓ to select the target line No.
    2. Press **F1**. The system saves the workpiece offsets of the current program file into the target line No.

    If the workpiece offset already exists in the target line No., the operation will overwrite it.

3. To recover coordinates, do the following:

    1. To select the target line No, press ↑ / ↓.
    2. Press **F2**, and choose whether to recover the workpiece offset of Z-axis:

        - To recover the workpiece offsets of all axes, press **Y** in the two pop-up dialog boxes.
        - To recover the workpiece offsets except Z-axis, press **Y** only in the first pop-up dialog box.

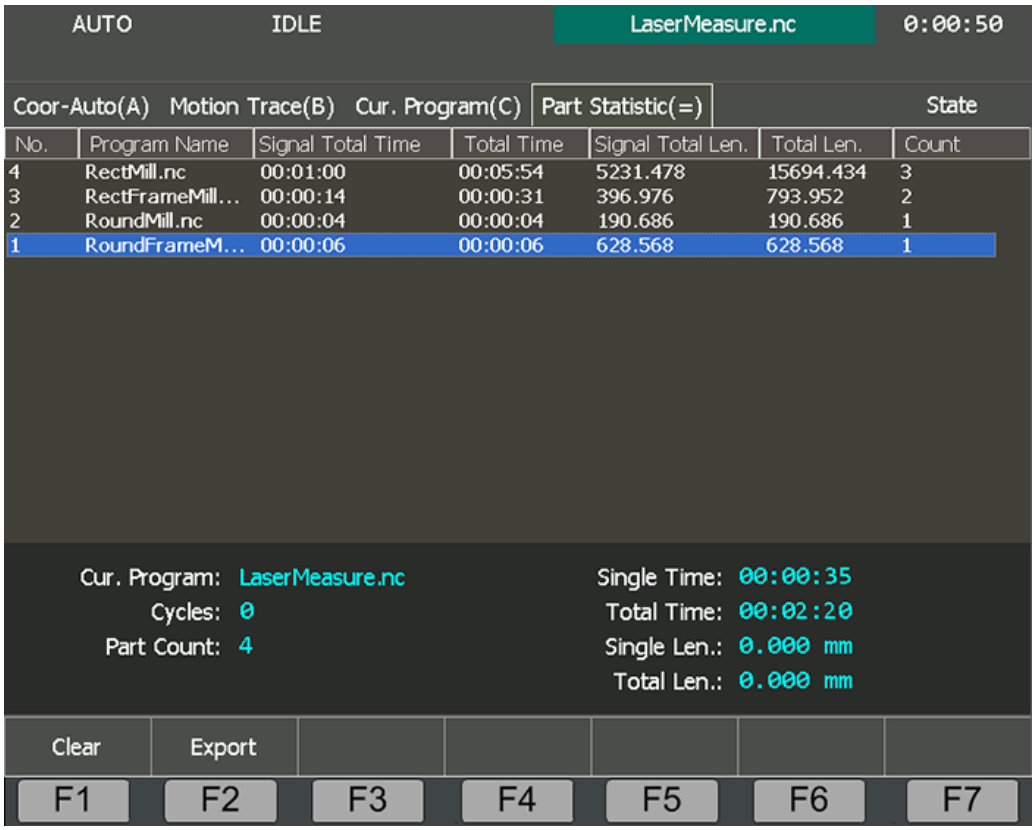
5.6Check Machining Statistics

This operation is used to check the collection of machining information shown in numbers of both current program file and program files in history.

To check machining statistics, do the following:

1. To enter into **Part Statistic** interface, press 

→ **=**:



2. Check the following machining information:

Single Time
The time for each machining of the program file.

Total Time
The total time for current machining task.

Single Length
The length for the program file.

Total Length
The total length for current machining task.

Count
The total machining times of the program file.
When cycle machining is disabled, the single total time is equal to total time, and single total length is equal to total length.
See Execute Cycle Machining for cycle machining.

In **Part Statistic** interface, you can also do the following:

- To clear all statistics, press **F1**.
- To export all statistics to a USB flash disk, press **F2**.

  The statistics will be saved as a TXT file.

5.7 Compensate the Tool

This operation is used 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.

With this operation, you can directly input the new tool parameter values in the **Compensation** interface, if the tool nose radius is altered due to tool wear, tool sharpening or tool change. It saves trouble of modifying the programmed program file.

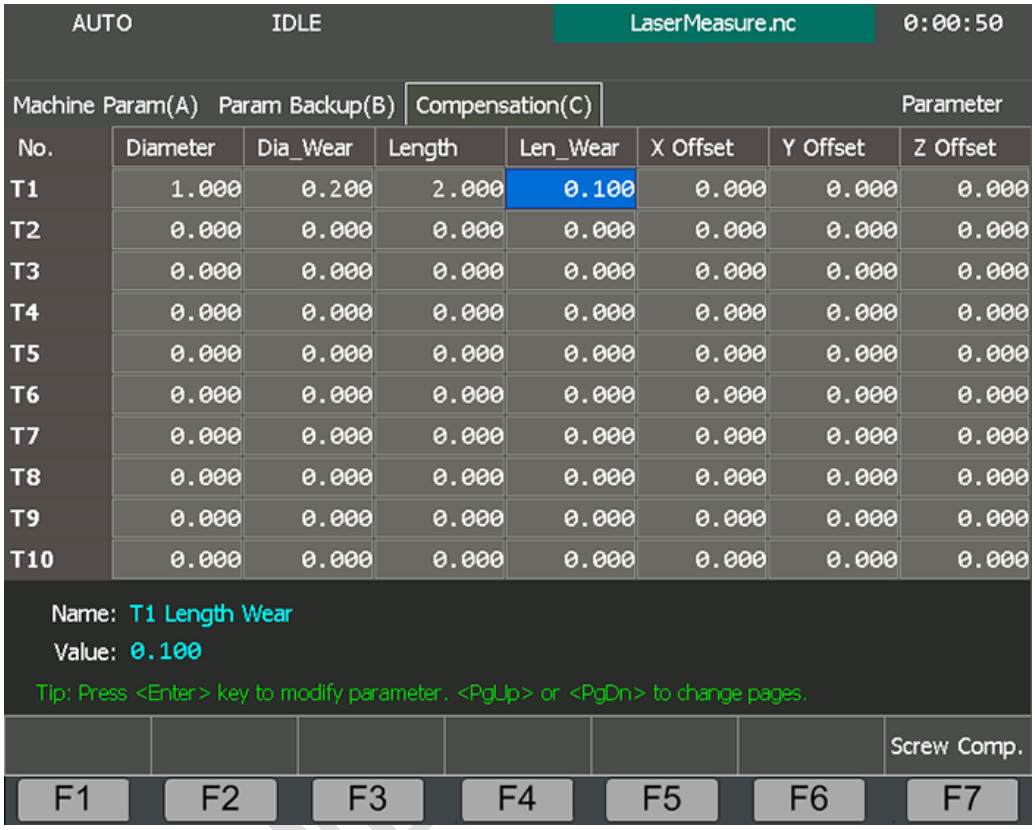
Before compensating the tool, do the following:

- Set parameter **Enable Cutter Compensation** to **Yes** to enable tool compensation.
- Set parameter **Tool Compensation Type** based on your own needs.
- Measure and record the following values:
  - Tool diameter
  - Tool diameter wear
  - Tool length
  - Tool length wear

To compensate the tool, do the following:

1. To enter into **Compensation** interface, press 

→ **C**:



2. To select the target tool number, press ↑ / ↓.

3. Press ← / → to select the target input box, press **Enter**, and modify the value of the parameter according to the measured values.

Tool compensation can be achieved by calling related commands. See NcStudio Programming Manual for details.

5.8Compensate Screw Error

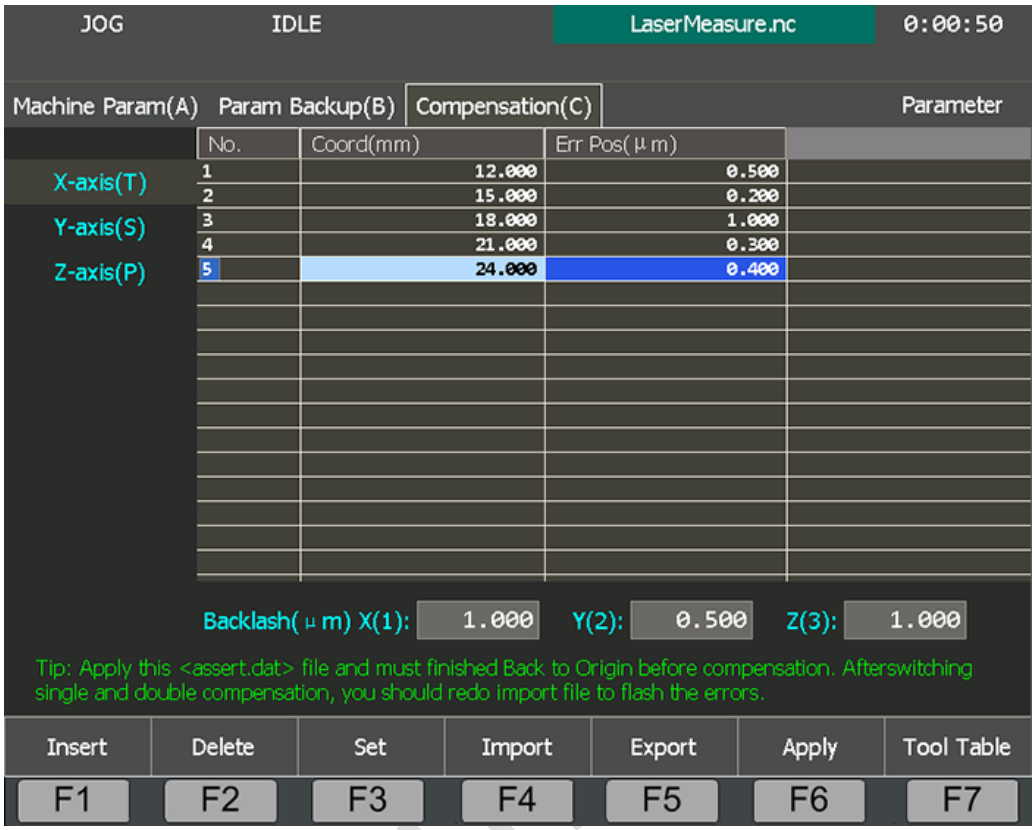
This operation is used to compensate screw pitch error and errors caused by backlash, to achieve high precision.

To compensate screw error, do the following:

1. To enter into **Compensation** interface, press 

→ **C**.

2. To enter into **Screw Comp.** page, press **F7**:



3. Do one of the following:

   - Compensate screw error by an external file.
   - Compensate screw error by related parameter settings.

5.8.1 Compensate Screw Error by an External File

This operation is used to automatically compensate screw error based on the error data in the file, after the backward error and forward error of the corresponding nominal coordinate of each coordinate axis are listed into the screw error compensation file.

Before compensating screw error by an external file, do the following:

1. Write a compensation file for screw error, name it `axeserr.dat`, and save it into the root directory of a USB flash disk.

2. Inset the USB flash disk into host.

To compensate screw error by an external file, do the following:

1. Press **F4** → **F1**. **Import Compensation Data** dialog box pops up.

2. Select the target compensation data.

Note: If the value of parameter **Screw Error Comp** is modified, you need to import the target compensation data again.

You can also export the imported compensation file to a USB flash disk for later use by pressing **F7** →**F5**.

5.8.2 Compensate Screw Error by Related Parameter Settings

This operation is used to compensate screw error by setting compensation parameters about screw error in the software directly.

To compensate screw error by related parameters, do the following:

1. To select the axis needed to compensate, press **T** / **S** / **P**.
2. Do one of the following:
   - Press **F3**, and set the start position, interval and amount for compensation.
   - Press **F1** to insert a blank line, and set the start position.

     Note: It is not allowed to insert several blank lines at the same time.
3. To set unidirectional compensation, press ↑ / ↓ / ← / → to select the target input box of **Err Pos**, press **Enter**, and input a value according to the actual situation.
4. **Optional:** If backlash is needed, press **1**, **2** or **3** to set backlash for each axis.
5. To save and validate the setting, press **F6**.

Return to the machine origin.

5.9 Execute Instant Calibration

This operation is used to automatically execute fixed calibration for several tools.

See Execute Fixed Calibration for details.

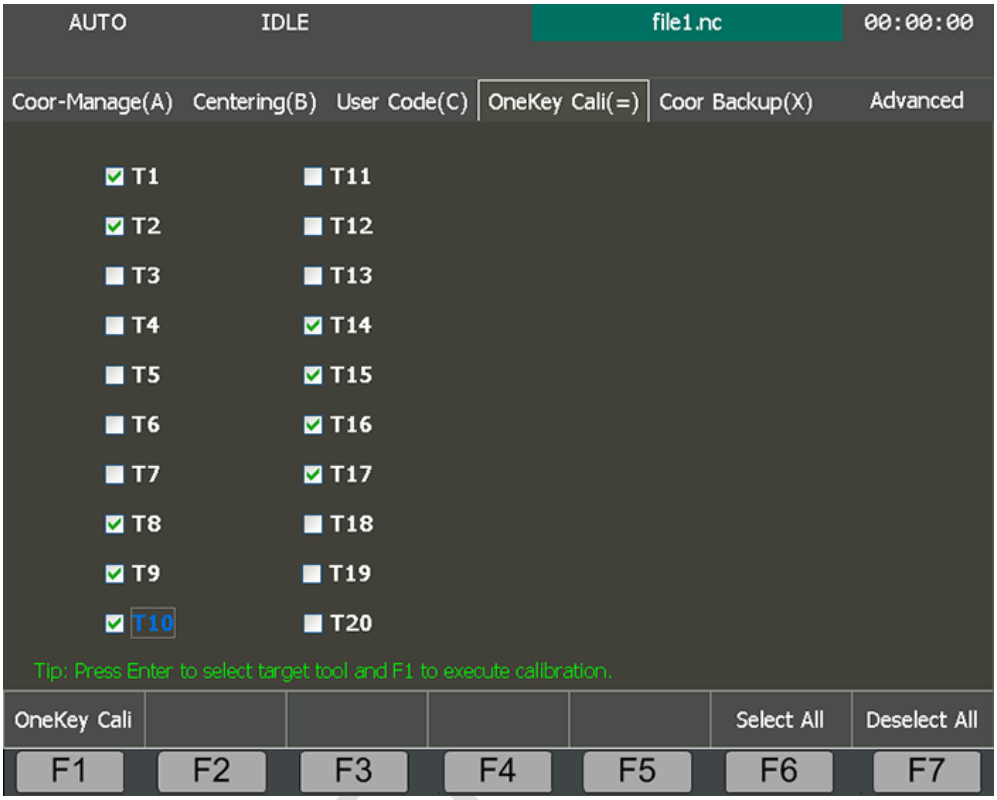
To execute instant calibration, do the following:

1. To enter into **Auto** mode, press 

.

2. To enter into **OneKey Cali** interface, press 

→ **=**:



3. To select target tools, do the following:
   - Press ↑ / ↓ → **Enter**.
   - Press **F6**, and select all tools.
   - Press **F6**, and remove all selected tools.
4. To start instant calibration, press **F1**.

The system automatically changes the tool and start to calibration.

6 System Management

Through this section, you can quickly know the following operations for system management:

- Register the system.
- Switch the system language.
- Maintain the system.
- Use NcCloud Assistant.
- Set the network.

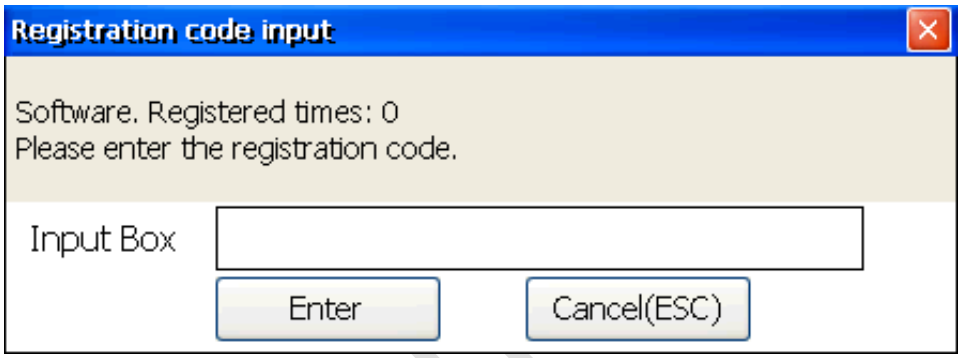
6.1 Register the System

This operation is used to get the legal usage time of the system.

To register the system, do the following:

1. Press 

→ **C** to enter into **System Info** interface, and check board card No. in **Hardware Info** area.
2. Send the number to your manufacturer or developer. Your manufacturer or developer sends you a registration code.
3. Press **F1**. **Registration code input** dialog box pops up:



4. Input the registration code in the input box.

6.2 Switch the System Language

This operation is used to change into another system language.

At present, simplified Chinese and English are available.

To switch the system language, do the following:

1. To enter into **System Info** interface, press 

→ **C**.
2. Press **F3**, and select the target language by pressing ↑ / ↓.
3. To confirm your selection, press **Enter**.

To validate the modification, restart the system.

6.3 Maintain the System

This operation is used to update the system and delete temporary files.

Before maintaining the system, ensure that a USB flash disk with an updated file has been inserted.

To maintain the system, do the following:

1. To enter into **System Info** interface, press 

→ **C**.
2. Press **F2**, and select a maintenance item:
   - **F1:** delete temporary files. Manufacturer password is required.
   - **F2:** update the system.

6.4 Use NcCloud Assistant

This operation is used to use **NcCloud Assistant** to provide data interface and communication protocols.

NcCloud Assistant is a built-in application in the system.

To use **NcCloud Assistant**, do the following:

1. Press 

→ **C** to enter into **System Info** interface.
2. Press **F5**, and follow steps in **NcCloud Assistant** dialog box.

   See NcCloud Assistant User Manual for details.

6.5 Set the Network

This operation is used to realize two-way transmission between the integrated CNC system and PC after establishing correct connection with PC.

See NK260 Networking for details.

7 Advanced Operations

Through this section, you can know the following customized operations:

- Load / unload material

  This operation is used to achieve one of the following actions:

  - Load material: the system automatically sends the machining workpiece to the machining position.
  - Unload material: the system automatically takes off the machined workpiece from the machining position after machining ends.
  - Load and unload material: the system automatically sends the machining workpiece to the machining position, and then automatically takes off the machined workpiece from the machining position after machining ends.
- Change tool by cylinders

  This operation is used to change tool for machine tools with multi-cylinders.
- Change tools for different types of tool magazines:

  This operation is used to change tool for the following types of tool magazines:

  - Linear tool magazines
  - Circular tool magazines
  - Double circular tool magazines
  - Servo tool magazines
  - Double servo tool magazines

  Note: Changing tool by the last two types of tool magazines is not supported in **NK260**.
- Enable gang drills

  This operation is used to change the gang drill or saw blades for machine tools with gang drills or saw blades.
- Start machining with double stations

  This operation is used to switch stations in turns for machining, so as to improve machining efficiency.

  At present, it is mainly used for labelling.

Please contact our technic sales engineers if you need to customize these operations.

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