

# Preface

---

---

get familiar with the HNC-818 system (hereafter referred to as "system"), providing detailed information about the features, components, commands, usage, operation procedure, programming and beyond. Any updates or modification of the manual is not allowed without the authorization of Wuhan Huazhong Numerical Control Co., LTD (hereafter referred to as "Huazhong NC") under any circumstances. Huazhong NC will not be responsible for any loss caused by pirated copies.

The documentation focuses on the main operations of the system. Limited by space as well as product conceptualization and development, it's impossible for us to explain anything unnecessary or impossible. Hence, what are not described in the manual can be regarded as "IMPOSSIBLE" or "NOT ALLOWED".

The documentation is protected by copyright and contains proprietary and confidential information. No part of the contents of the documentation may be disclosed, used or reproduced in any form, or by any means, without the prior written consent of the copyright holder.

The manual may help you to quickly

# Contents

Preface.....	i
Contents .....	ii
I overview .....	4
II Operation.....	6
1 Operation Device.....	7
1.1 Display Screen .....	8
1.2 NC Keyboard .....	9
1.3 Machine Control Panel .....	10
1.4 Hand-Held Unit.....	11
1.5 Operation Panels .....	12
2 Power On, Power Off and Emergency Stop .....	14
2.1 Power On.....	15
2.2 Reset .....	16
2.3 Reference Returning .....	17
2.4 Emergency Stop .....	19
2.5 Over-Travel Release.....	20
2.6 Power Off .....	21
3 Manual Operation.....	22
3.1 Manually Move the Coordinate Axis.....	23
3.2 Spindle Control .....	27
3.3 Machine Lock and Axis Z Lock .....	29
3.4 Manual Data Input (MDI).....	30
4 Settings .....	33
4.1 Tool Management.....	34
4.2 Coordinate System Settings .....	39
4.3 Clear RCS.....	41
4.4 Parameter Settings.....	42
5 Program Editing and Management .....	68
5.1 Select Programs.....	69
5.2 Edit Program .....	74
5.3 Program Management .....	78
5.4 Random Line .....	81
5.5 Verify Program .....	88
5.6 Stop Program.....	89
5.7 Restart Program.....	90
5.8 Craft File .....	91
5.9 Workpiece Measurement.....	93

6 Operation Control.....	97
6.1 Start, Pause and Stop.....	98
6.2 Dry Running.....	100
6.3 Over-Block Operation.....	101
6.4 Optional Stop.....	102
6.5 Single Block Operation.....	103
6.6 Breakpoint Related Operation .....	104
6.7 Intervention at runtime.....	107
7 Position Information.....	109
7.1 Coordinate Display .....	110
7.2 Text Display .....	111
7.3 Graph Display .....	112
7.4 Joint Display.....	114
8 Diagnosis .....	115
8.1 Import System Logo Interface .....	116
8.2 Alarm Display .....	118
8.3 Alarm History .....	119
8.4 Ladder Monitoring .....	122
8.5 Input and Output .....	128
8.6 Status Display.....	129
8.7 Macro Variables .....	130
8.8 Servo Adjustment.....	132
8.9 Machining Information.....	146
8.10 Version.....	148
9 User Operation and Maintenance Information.....	150
9.1 Environmental Requirements .....	151
9.2 Grounding.....	152
9.3 Power Conditions .....	153
9.4 Clean Fan Filter.....	154
9.5 Operations After Being Left Unused.....	155

# I overview

The following models are described in the manual:

Name	Model
HNC-818A Milling NC Unit	HNC-818A-MU
HNC-818B Milling NC Unit	HNC-818B-MU

## **II Operation**

## 1 Operation Device

---

---

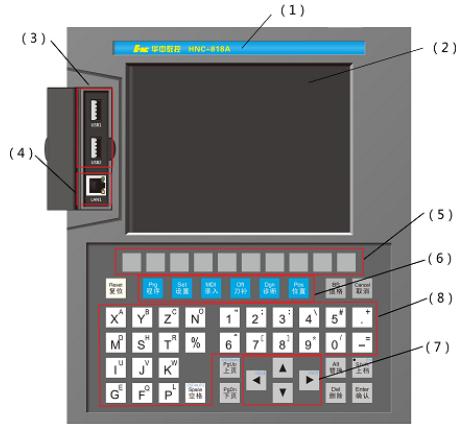
This chapter describes the operator area and system operation interface of HNC-818.

- **Display and Operation Panel**
- **NC Keyboard**
- **Machine Control Panel**
- **Hand-Held Unit**
- **System Operation Interface**

## 1.1 Display Screen

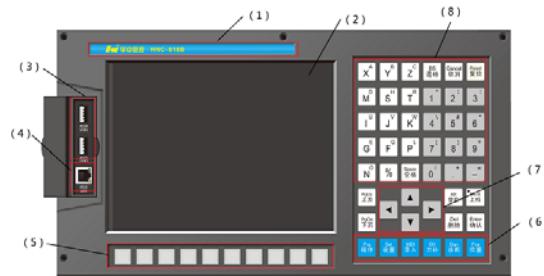
### HNC-818A-MU

- 8.4-inch color liquid crystal display (resolution: 800×600)



### HNC-818B-MU

- 10.4-inch color liquid crystal display (resolution: 800×600)



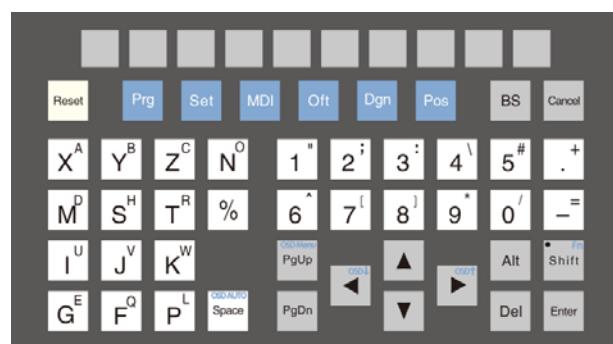
1. Company Logo
2. Display screen
3. USB interface
4. Ethernet interface
5. Soft keys
6. Functional keys
7. Cursor control
8. Characters and numbers

## 1.2 NC Keyboard

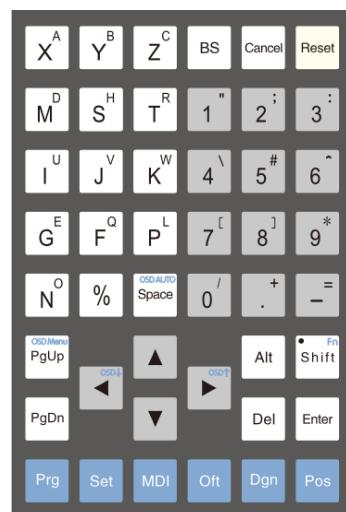
The NC keyboard includes a compact MDI keys, main menu keys and functional keys, which are mainly used for the programming, parameter input, MDI and system management and other operations

- MDI keys: Most of them have the function of the **Shift** key. For example, you may press the **Shift** key and a letter/number key simultaneously to input the characters in the upper corner.

HNC-818A



HNC-818B



### 1.3 Machine Control Panel

---

The machine control panel is used to control the motion and machining process.

**HNC-818A-MU**



**HNC-818B-MU**



## 1.4 Hand-Held Unit

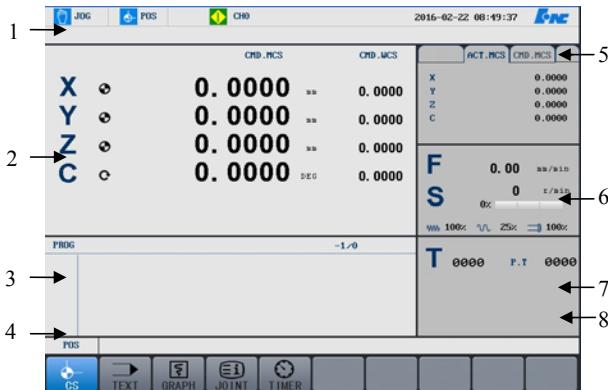
---

The hand-held unit, which is used to feed coordinate axis by hand, consists of a manual pulse generator (MPG) and coordinate axis selection switch. The structure of the handheld unit is shown below (the appearance as per the actual products):



## 1.5 Operation Panels

The operation interface of HNC-818 is the following:



1. Title bar

- Machining mode: Switch the machining mode among Auto, Single Block, Manual, Incremental, Reference, and Emergency Stop.
- Main menu: Display the currently activated main menu.
- Workstation: Display the current workstation.
- Channel information: Display the work status of each channel, such as "Normal", "Feed Hold", or "Error".
- System time: Display the current system time (it can be selected in the machine parameter).
- System alarm information.

2. Graphical display: The information displayed is based on the selected menus.

3. Code G display: Preview or display the G code of the machining program.

4. Menu command bar: run functional operation through corresponding functional key in the menu command bar.

5. Tabs: View different coordinate systems by switching the tabs.

6. Supplementary functions: Display the F/S information in automatic machining and adjustment information.

7. Tool information: Display the current tool.
8. Modal G and Machining time(in the main menu "Program":Display the G modal during machining and the machining time.

## 2 Power On, Power Off and Emergency Stop

---

---

The chapter describes the operations on power on/off, emergency stop, reset, reference returning and over-travel release.

## 2.1 Power On

---

### Operation procedure

1. Check and make sure the machine is normal to run.
2. Make sure the power supply and voltage to meet requirements and the wiring is correct.
3. Press the **Emergency Stop** button.
4. Electrify the machine.
5. Electrify the CNC system.
6. Make sure the indicators on the machine control panel are working.
7. After electrifying the CNC system, the system automatically runs with the machining mode being **Emergency Stop**.

## 2.2 Reset

---

### Operation procedure

When the system enters the operation interface in a power-on state, the initial operation mode is displayed as **Emergency Stop**. The **Emergency Stop** button should be pulled up in the clockwise direction to reset the system and enable the servo. Then the system could control the machining.

## 2.3 Reference Returning



The control of the Machine motion is based on the establishment of coordinate system . Therefore, after the system is reset, each machine axis must return to the reference point. Follow the instructions below:

1. If the currently operation mode is not **Home Return**, press the **Reference Return** button on the control panel to make sure that the system is in the **Home Return** mode.
2. Based on the X-axis machine parameter of **REF Direction**, press **X** and the direction key (the value of **REF Direction** is "+"). After the X- axis returns to the reference point, the indicator of the **X** key is on.
3. Return the Z axis to the reference point with the same method.
4. After return every axes to the reference point, the machine coordinate system is established.

### Attention

1. Return all axes to the reference point as soon as the power is on in order to ensure the axis coordinate is correct. After that, other operations can be performed.
2. Press the direction keys of X, Y, and Z simultaneously to return the three axes to the reference point at the same time.
3. Before returning the axis to the reference point, make sure that the axis is in the opposite direction of **REF Direction** (e.g. if the **REF Direction** of the X axis is "-", then make sure the X axis direction is "+" before returning it to the reference point); otherwise, manually move the axis until it meets the requirement.
4. When returning axis to the reference point, if over-travel occurs, press the over-travel release key on the control panel and manually move the axis in the opposite direction to exit the over-travel status.
5. After return each axis to the reference point, the reference return doesn't need to be performed again (including alarms and pressing the **Emergency Stop** button ) unless a servo drive device alarm is reported.
6. When returning axis to the reference point, if the **Reset** key is pressed before the **Reference Returning** key is pressed, the home returning will be canceled.

7. When returning axis to the reference point, if the **Reset** key is pressed after the **Reference Returning** key is pressed, the home returning will not be canceled.

## 2.4 Emergency Stop



During machining, press the **Emergency Stop** (ES) button to enter into the "Emergency Stop" mode in case of emergency. Once the ES button is pressed, the servo feed and the spindle rotating are disabled (the feed driver power supply is cut off in the control cabinet). Release the ES button ( turn the ES button clockwise ), then the system enters the reset status.

Make sure that the trouble has been solved before releasing the ES status. Return the axis to the reference point again after releasing the ES button to ensure correct coordinate position.

### Attention

Press the ES button before turning the power on or off to avoid electric shock.

## 2.5 Over-Travel Release

---



There is a limit switch at each end of the axis stroke to avoid servo collision. If the servo meets the limit switch, the over travel occurs. The situation that the axis travels over (the relevant indicators is on) is treated as an emergency stop by the system. Follow the instructions below to exit over travel:

1. Set the operation mode to JOG or HANDLE.
2. Press down and hold the **over travel release** key (the controller will temporarily ignore the emergency).
3. In the manual (hand-held) mode, move the axis to the opposite direction.
4. Release the **over travel release** key.
5. Then the status bar of the screen shows "Normal" instead of "Error", which means the system is back to normal and the operation can be allowed.

### Attention

Please pay attention to the motion direction and speed when moving the tool out of over-travel to avoid tool collision with the machine.

## 2.6 Power Off

---

### Operation Procedure

1. Press the Emergency Stop button to cut off the servo power.
2. Cut off the CNC power.
3. Cut off the machine power.



### 3 Manual Operation

---

---

The section of manual operation includes the hand-held units and machine control panel. This chapter describes the following content:

- **Manually Move the Coordinate Axis**
- **Manually Control Spindle**
- **Lock Machine**
- **Manual Data Input (MDI)**
- **Other manual operations**

### 3.1 Manually Move the Coordinate Axis

The manual operation related to machine axis movement is conducted on the hand-held units and control panel with relevant keys such as operation mode selection, axis manual, incremental rate, feed rate adjustment, and rapid adjustment , etc.

#### 3.1.1 Manual Feed



Press this key (the indicator is on) to enter the Manual mode. In this mode, the cutting tool can be jogged along the motion axis. The following describes the motion along the X axis:

1. Press the key X and the direction key (the corresponding indicator is on) to move along axis X continuously in the positive or negative direction.
2. Release the key X and the direction key (the corresponding indicator is off) to decrease and stop the motion along the axis X.

Move along axis Z continuously in the positive or negative direction with the same method.

In the Manual mode, press the axis X and Z keys simultaneously to manually control both axes motion at the same time.

#### 3.1.2 Rapid Traverse



key and one of the three axis keys simultaneously to rapidly move the tool in the positive or negative direction of relevant axis.

In the manual feed mode, press this

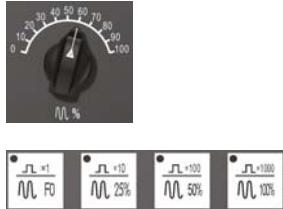
#### 3.1.3 Manual Feed Rate Adjustment



In the Auto or MDI mode, adjust the feed rate by rotating this button if the feed rate of the F-code programming is too high or too low. The adjustment range is **0% to 120%**.

In the manual mode, this button can be used to adjust the manual feed rate.

### 3.1.4 Rapid Traverse Speed Adjustment



The operations on rapid traverse speed adjustment are different based on different control panels.

1. Rapid traverse speed adjustment button: In the Auto or MDI mode, rotate this button to adjust the speed which is specified in the program. The range of adjustment is from **0%** to **100%**.
2. Override keys: In the Auto or MDI mode, press the corresponding override keys to adjust traverse speed.

### 3.1.5 Incremental Feed



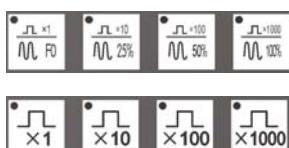
enter the incremental feed mode in which the tool can be moved incrementally along any axis. Take the axis X as an example:

1. Press the key X and the direction key (the corresponding indicator is on) to move the axis X with an incremental value in the positive or negative direction.
2. Press the X key and the direction key (the corresponding indicator is on) again to proceed the movement along the axis X with an incremental value in the positive or negative direction.
3. Perform similar operation to move the axis Z with an incremental value in the positive or negative direction.

Press the keys for axis X and Z simultaneously to manually move the axis X and Z with an incremental value at the same time.

Press this key (the indicator is on) to

### 3.1.6 Incremental Value Selection



There are different keys of incremental value based on different

### control panels

The incremental value is controlled by the four keys on the control panel: "**×1**", "**×10**", "**×100**", and "**×1000**".

The table below describes the relationship between the incremental value and override keys:

magnification:

Override	×1	×10	×100	×1000
Incremental value (mm)	0.001	0.01	0.1	1

**Note:** These keys can be locked by each other. When you press one of the keys (the corresponding indicator is on), then the other keys are invalid (the corresponding indicator is off).

### 3.1.7 Hand-wheel Feed

When the system is not in the emergency stop mode, press the incremental key (the indicator is on) to enter the hand-wheel feed mode. In this mode, the machine axis can be moved by the hand-held unit. Rotate the axis selection button to choose one axis, and then rotate the override button to the required override position. Through the rotation, the machine axis will move an incremental distance accordingly. In the hand-wheel feed mode, incremental feed for only one axis is supported each time.

## 3.2 Spindle Control

---

Manual spindle control is conducted with the spindle rotation keys on the control panel.

### 3.2.1 Clockwise Rotation of Spindle



In the manual mode, press this key (the corresponding indicator is on) to rotate the spindle in the clockwise direction at the specified speed until the counter clockwise rotation key or spindle stop key is pressed.

### 3.2.2 Counter Clockwise Rotation of Spindle



In the manual mode, press this key (the corresponding indicator is on) to rotate the spindle in the counter clockwise direction at the specified speed until the clockwise rotation key or spindle stop key is pressed.

### 3.2.3 Stop Spindle



In the manual mode, press this key (the corresponding indicator is on) to stop the spindle motor.

Note: The keys for spindle clockwise rotation, spindle counter clockwise rotation, and spindle stop can be locked by each other. When you press one of the keys (the corresponding indicator is on), then the other keys are invalid (the corresponding indicator is off).

### 3.2.4 Spindle Jog



In the manual mode, press and hold this key (the indicator is on), then the spindle will continuously rotate in the clockwise direction; Release this key (the indicator turns off), the spindle will stop running.

### 3.2.5 Spindle Speed Adjustment



The rotation speed of spindle can be adjusted by the spindle speed adjustment button.

Rotate the button to an override as required, which ranging from **50%** to **120%**. Do not adjust the spindle speed during the rate switching of mechanical gear .

### 3.2.6 Spindle Orientation



If the machine has a tool magazine, the spindle orientation function is usually required, because the tool on the spindle must be orientated during tool change; otherwise the tool or tool claw may be damaged.

In the manual mode, when the "Spindle Brake" is invalid (the indicator is off), press this key, and the spindle orientation function is immediately executed. After the orientation is completed, the indicator of the key is on, and the spindle accurately stops at a fixed position.

## 3.3 Machine Lock and Axis Z Lock

---

### 3.3.1 Machine Lock



The machine lock function is to lock all machine motion.

In the manual mode, press this key (the indicator is on) and then conduct manual operation, the information of axis position changes which is displayed on the screen , but the motion instructions from the servo axis is not output. Therefore, the machine stops moving.

**Note:** The machine lock function is valid only in the manual mode.

### 3.3.2 Axis Z Lock



feed. When just verify the movement path in the XY plane, this function is useful. In the manual mode, press this key (the indicator is on), and switch to the auto mode to run the machining programs. Then the information of Z-axis coordinate position is changed without actual Z-axis movement.

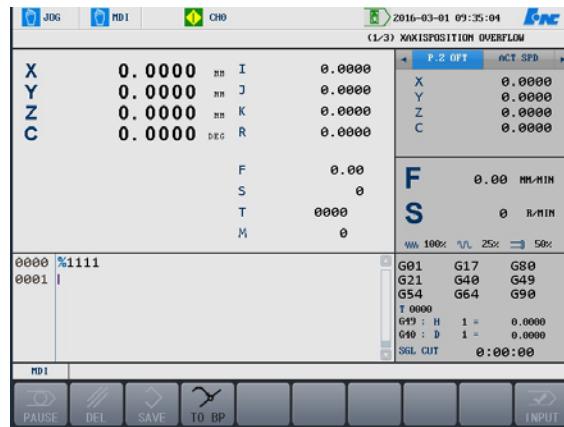
**Note:** The Axis Z Lock function is invalid in the auto mode.

This function is used to disable tool

### **3.4 Manual Data Input (MDI)**



Press the MDI key to enable the corresponding function. Input and execute one or more Code G blocks through the NC keyboard. See the figure below:



## Attention

1. After the system enters the MDI status, the MDI icon is displayed on the title bar.
  2. When the interface is switch to the non-program from MDI, the system is still in the MDI status.
  3. In the auto operation mode, the system cannot enter the MDI mode, but can do that after feed hold.
  4. In the MDI mode, press the **Reset** key to stop and clear the MDI program.



### 3.4.1 Input MDI Command Segment

## **Operation procedure**

The minimum unit for MDI is a valid command character. Therefore, there are two methods to input an MDI command segment:

1. Input once, that is, to input multiple command characters once.
  2. Input by several times, that is, to input only one character every time.

For example, if you need to input the "G00 X100 Z1000" command segments, you may:

1. Directly input "G00 X100 Z1000"

2. Press **Enter**. Then the values after X, Z on the screen change to **100**, and **1000** respectively.

The content will be shown on the screen when a command is input. If input wrong, press **BS**, **▶**, and **◀** keys to correct the content. If the system detects an error after **Enter**, a corresponding error message will be displayed. In this case, press **Clear** to clear the input information.

### 3.4.2 Execute MDI Commands

#### Operation procedure

After inputting an MDI command segment, press the Start button on the operator panel, and then the system will run the entered MDI command.

If the MDI command is incomplete or has wrong syntax, the system will display related error messages, and the MDI command stops.

### 3.4.3 Modify Commands

#### Operation procedure

Before running MDI command, directly input corresponding command characters and values in a command line if needed. For example, after entering "**X100**", you may input "**X109**" in the command line if you need to change the X-axis value to **109**.

### 3.4.4 Clear Current Commands

#### Operation procedure

After inputting MDI data, press the **Clear** key to clear all currently input data (other commands are still valid). All data behind X, Z, I, K, R is cleared. New data could be entered as required.

### 3.4.5 Stop Current MDI Commands

#### Operation procedure

When the system is running MDI commands, press the **Stop** key to stop running commands.

### 3.4.6 Save the Input MDI Command

#### Operation procedure

Press the save key to save the input  
G code as a program.

## 4 Settings

---

---

This chapter describes the data settings of the system, which includes:

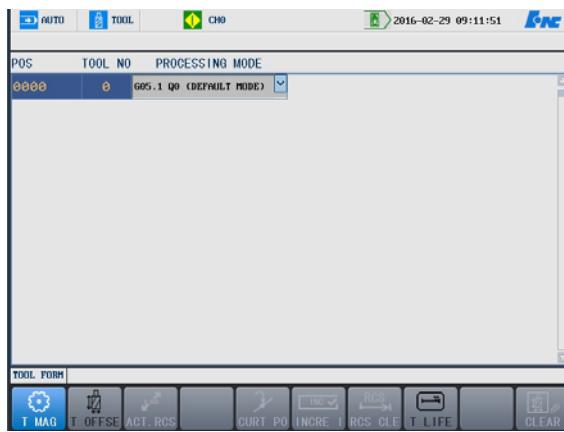
- **Tool Data (T series)**
- **Tool Data (M series)**
- **Coordinate System Settings**
- **RCS Clear**
- **Parameter Settings**

## 4.1 Tool Management

### 4.1.1 Tool Magazine

#### Operation procedure

1. Press **Oft** → **T MAG**. The tool magazine data table is displayed, where the data can be set. See the figure below:



2. Press **▲** and **▼** keys to move the cursor to select the item to be edited.
3. Press **Enter** to enter the editing status.
4. Press **Enter** again after modification.

### 4.1.2 Tool Compensation

#### Operation procedure

1. Press the **Oft** main menu. The tool compensation data table, which includes tool length, tool radius, length wear and radius wear, is displayed.
2. Press **▲** and **▼** keys to move the cursor to select the required tool number.
3. Press **▶** and **◀** keys to select the items to edit.
4. For example, select “length” option and press “current position” key, the system will automatically input the current position data.

5. Press **Enter** to enter into the editing status; Press the “incremental input” key so that the system automatically adds data.

**Note:** The valid range of data for editing is from -10000.0 to 10000.0.

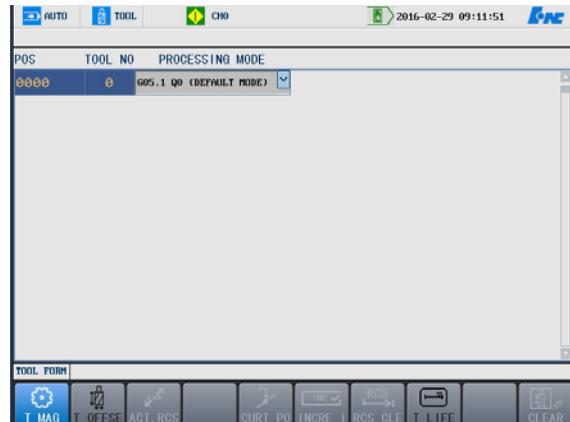
6. After editing, press **Enter** again.

### 4.1.3 Tool Life Management

See section 4.1.4.

### 4.1.4 Tool Machining Modes

Machining modes corresponding to tool number can be set on the tool interface. After calling a tool, the system automatically calls the corresponding tool machining mode, e.g. default mode (G05.1Q0), high-precision mode (G05.1Q1), high-speed high-precision mode (G05.1Q2), high-speed mode (G05.1Q3). See the figure below:



To use this function, it's necessary to call **G115L3** before all **M99** commands in the M06 subprogram of **USERDEF.CYC**, and enable the tool machining mode function through **010089 [T command control mode]**.

Disable the tool machining mode function through **010089 [T command control mode]** if you do not need to use this function.

### 4.1.5 Tool Management Based on RFID

Power on the reid reader and connect the RS232 communication interface to the computer.

Run the "SYGOL high-frequency digital reader Demo program.exe" to achieve the read-and-write data transmission of RFID and CNC tool data. See the figure below:



批注 [z1]: 中文图

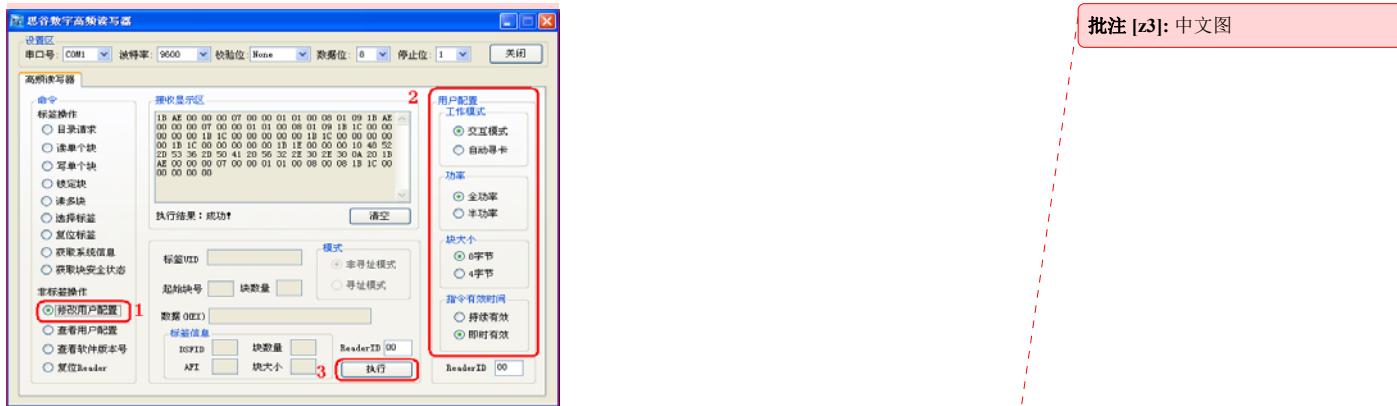
- As shown in the figure below, click **Open**.



批注 [z2]: 中文图

- As shown in the figure below: select **Change user configuration** (see 1), configure user information (see 2), and then select **Execute** (see 3) to complete the user configuration of the reader.

For detailed information, see the relevant commissioning user manual.



#### 4.1.6 RCS Clear

Write the Z-axis values in the relative actual coordinates into the corresponding tool length compensation where the cursor is.

TOOL NO	LEN	LEN FRAY	R	DIA FRAY
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000	0.0000
4	0.0000	0.0000	0.0000	0.0000
5	0.0000	0.0000	0.0000	0.0000
6	0.0000	0.0000	0.0000	0.0000
7	0.0000	0.0000	0.0000	0.0000
8	0.0000	0.0000	0.0000	0.0000
	A.CT.MCS	A.CT.RCS	A.CT.MCS	
X	0.7550	0.7550	0.7550	
Y	-2.4570	-2.4570	-2.4570	
Z	0.0000	0.0000	0.0000	
C	0.0000	0.0000	0.0000	



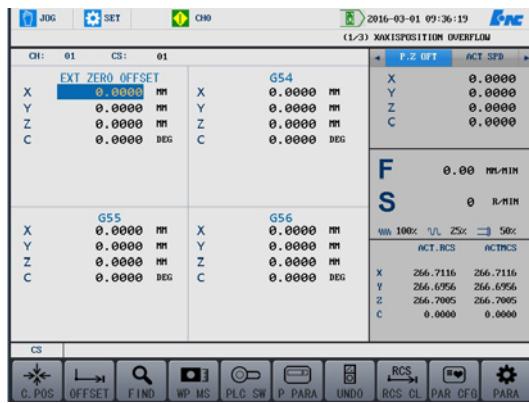
## 4.2 Coordinate System Settings

### 4.2.1 Setting Coordinate System



Follow the instructions below to set the coordinate system:

1. Press the **Set** menu and enter the manual coordinate system settings mode. See the figure below:



2. Press the "**PgDn**", "**PgUp**" keys to select the coordinate systems to be set. E.g. G54, G55, G56, G57, G58, G59, workpiece (the coordinate value of workpiece origin relative to the machine origin), relative (the origin of the current relative value), G54.1-G54.60.
3. Users may press **FIND** to search for specific workpiece coordinate system type. There are two input formats for coordinate system search:
  - "Px" indicates the extended coordinate system x  
E.g. P39 indicates to search the G54.39 extended workpiece coordinate system.
  - "x" indicates the coordinate system number  
E.g. If **2** is entered, then G55 will be searched.
4. Enter the position information of the selected coordinate system, then users may use any of the following methods:
  - Enter the required data in the editing box.
  - Press **C. POS**, **OFFSET**, and **UNDO** to enter data.

- b. [OFFSET]: If "+0.001" is entered, the coordinate system position of the selected axis is the current position plus the entered data. If "-0.001" is entered, the coordinate system position of the selected axis is the current position minus the entered data.
    - c. [UNDO]: Restore the last defined value.
      - The system reads the current tool position via pressing "WP MS →Record I" and "WP MS →Record II" ,then it calculates the midpoint of the two (Record I and Record II) through pressing "WP MS →Midpoit" and takes the point as the origin of the coordinate system.
  5. If the entered value is correct, the modified value will be displayed in the corresponding position; otherwise, the value will not be changed.
- a. [C. POS]: The system reads the current tool position.

## 4.3 Clear RCS

---

### Operation procedure

Press **Set** → **RCS CLE** to enter the interface as following:



Enter the axis name, e.g. **X**, the X axis values will be cleared. Meanwhile, The system coordinate system is changed to the relative coordinate system. The coordinate value is **0**. In this case, the coordinate value is the relative one based on the current position when the workpiece is manually moved. When you exit the interface, the coordinate system restores to the one before entering the relative coordinate system.

## 4.4 Parameter Settings

### 4.4.1 System Parameters

#### View parameters types

1. Press Set → PARA → NC PARA to enter the interface shown in the figure below:



2. Press ▲ and ▼ keys to select a parameter type as required.
3. Press ► key to enter the parameter list, select a parameter, and the detailed information will be displayed in the lower pane of the screen.

#### View Parameter Index

1. Press Set → PARA → NC PARA → Index.
2. Press ▲ and ▼ keys to select a parameter, and the detailed information will be displayed in the lower pane of the screen.

**Note:** See HNC-8 Parameters Description. for details about each parameter of HNC-818.

#### Enter password for parameter editing

1. Press Set → PARA → NC PARA → Password.
2. Enter the password for parameter editing.
3. Press Enter. If the password is correct, the parameters can be edited.

If users want to modify the system parameter values, they need to edit a password to obtain the permission.

**Edit Parameters**

1. Enter the correct password for parameter editing.
2. Select a parameter to be edited according to the index or type , press **Enter** to enter the editing status.
3. Enter a parameter value and press **Enter** to end the editing.

**Save parameters**

1. Press **Save** after editing all parameters.
2. Press **Y** if you want to save the modification.
3. Press **N** if you do not want to save the modification.

**Note:** Some parameters' modification takes effect only after power off and restarting the system.

**Restore the default value**

Press the corresponding **Default Restore** key if you want to restore the default value of a parameter.

**Restore the previous value**

After editing a parameter, press the corresponding **Previous Value Restore** key if you want to restore the value before editing, .

**Note:** The operation works only before the parameter value is saved.

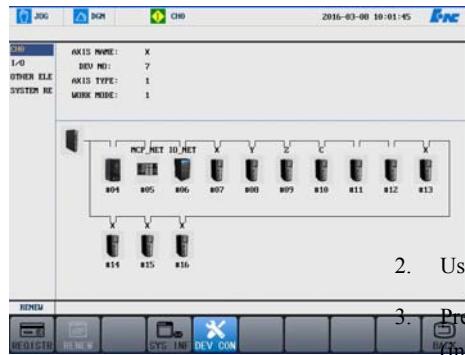
**Search parameters**

In the parameter index viewing mode, press **Search**, enter the parameter index number, press **Enter**, the system will locate the defined parameter.

**Device configuration**

Use the device configuration navigation function to set the device number.

1. Press **Set** →**PARA** → **NC PARA** → **DEV CON**. The topology of the hardware connection is as below:



2. Use the **▲** and **▼** keys to select a device type.
3. Press **Enter**. The configured information of the device type (including axis name, input, output, or other units) is displayed.
4. Press "Alt+n" and move the cursor to the area on the right of the screen.
5. Press **▲** and **▼** keys to select the data type to be edited.
  - Channel: axis name, device number, axis type, and operation mode.
  - Input and output: device name, device number, initial group number, and group count.
  - Other units: device name and device number.
6. Press **Enter** to edit the selected data type (except for the device number). If you need to edit the device number, use **▲** and **▼** keys to move the cursor to select the device in the device configuration navigation pane and press **Enter**. The system will automatically read the device number.

**Note:** See HNC-8 Parameters Description. for details about each device.

#### Automatic offset

1. Press **Set** → **PARA** → **NC PARA** → **A.OFT**.
2. Enter the logic axis number.
3. Press **Enter**, and then the system automatically calculates and enters the encoder feedback offset of the corresponding axis parameter.
4. Press the **Save** key to save the modification.

#### 4.4.2 Parameters Display

##### Operation procedure

Set the information displayed in the large character area and small character area.

1. Press **Set →PARA→ Display Parameters.**
2. Use **▲** and **▼** keys for selection.
- **Display column 1:** set the value of the first column in the large character area
- **Display column 2:** set the value of the second column in the large character area
- **Display column 3:** Set the value displayed on the tab
3. Use **▶** key to move the cursor to the parameter list.
4. Use **▲** and **▼** keys to select the type to be displayed.
5. Press **Enter.**

**Note:** You may use **▶** and **◀** keys to switch the value displayed on the tab.

#### 4.4.3 Graphic Settings of User P Parameters

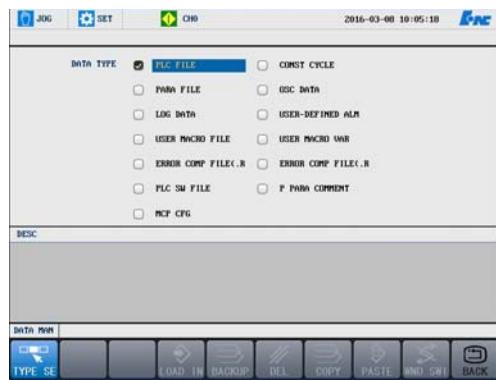
The settings of P parameters require the **USERP.STR** configuration file under the **parm** directory. The format of the file is as below:

```

USERP.STR - 记事本
文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)
0 主轴修调[50] 0
1 主轴修调[70] 0
2 主轴修调[80] 0
3 主轴修调[90] 0
4 主轴修调[99] 0
5 主轴修调[100] 0
6 主轴修调[110] 0
7 主轴修调[120] 0
8 进给修调[0] 0
9 进给修调[1] 0
10 进给修调[2] 0
11 进给修调[4] 0
12 进给修调[6] 0
13 进给修调[8] 0
14 进给修调[10] 0
15 进给修调[15] 0

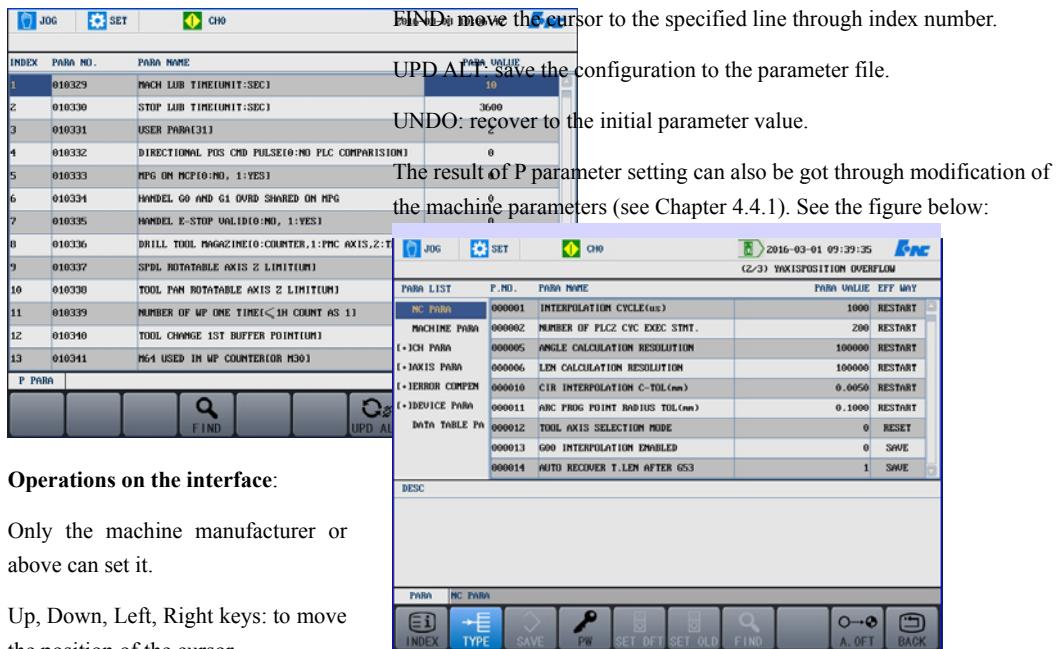
```

Press Set → PARA → DATA MAN and select PLC FILE to import the created USERP.STR configuration file into the CNC system. See the figure below:



This function is used to set the values of the user parameters

(010300 to 010499). See the figure below:



#### Operations on the interface:

Only the machine manufacturer or above can set it.

Up, Down, Left, Right keys: to move the position of the cursor.

Enter: Set the parameter value where the cursor is.

#### Operations on the menu:

#### 4.4.4 Parameter Update after System Upgrade

parameter value to be input.

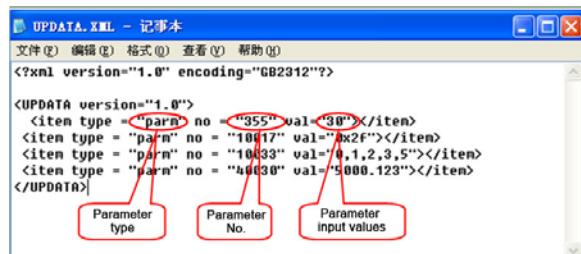
**Note:**

1. After the **UPDATA.XML** file is successfully read, it will be deleted. Therefore, this function takes effect only once.
2. After the parameters are updated from the UPDATA.XML file, the system will save and load the parameters again.
3. The **UPDATA.XML** file is released with the software upgrade package. Just place it into the **data** directory of the upgrade package.

If some new parameters are added after system version upgrade, the new parameters value should be manually defined. Hence, the function is used to automatically update the new parameters values.

Write the parameter values that need to be defined into the **UPDATA.XML** file, and then place it into the system upgrade package. During the system upgrade, the **UPDATA.XML** file will also be updated to the CNC system. When CNC starts, the system will detect and read the information in the **UPDATA.XML** file. The parameter values will be automatically reset.

UPDATA.XML file format:



```

<?xml version="1.0" encoding="GB2312"?>
<UPDATA version="1.0">
  <item type = "parm" no = "955" val = "30"></item>
  <item type = "parm" no = "10417" val = "0x2F"></item>
  <item type = "parm" no = "10433" val = "4,1,2,3,5"></item>
  <item type = "parm" no = "40030" val = "5000.123"></item>
</UPDATA>

```

**Parameter Type:** fixed as "parm".

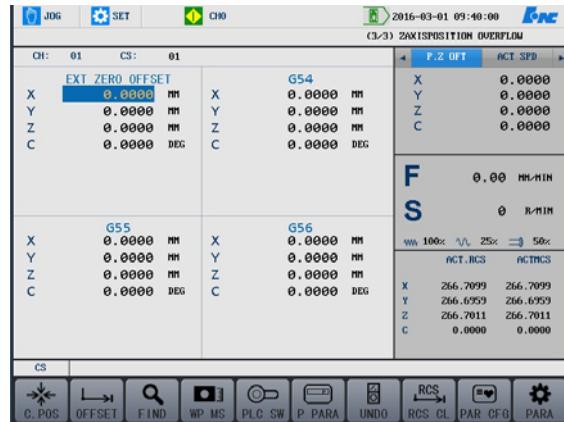
Items without it cannot be identified by the system.

**Parameter Number:** the number of the parameter requires automatic update.

**Parameter Input Value:** the

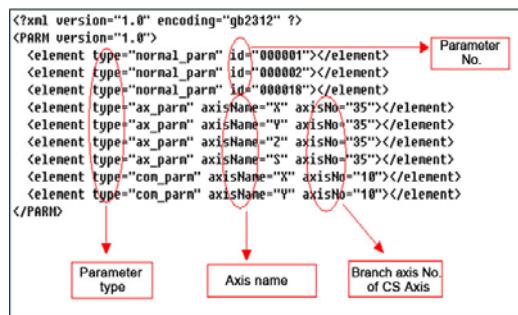
#### 4.4.5 Parameter Configuration Interface

This function is used to manage parameters as required. See the figure below:



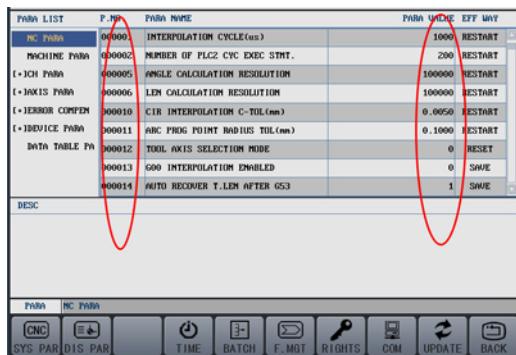
#### Operation description

1. XML file configuration: Users and machine debugging personnel configure the xml. parameter files which maintain their own parameter lists. The .xml file for users is named as **CFG\_PARM\_U.XML** which can save up to **50** parameters. The .xml file for debugging personnel is named as **CFG\_PARM\_D.XML** which can save up to **200** parameters. Both are placed in the **data** directory. See the figure below:



- a) For non-axis and non-compensation parameters (normal\_parm), directly enter the parameter ID (id).

- b) For axis parameters (ax\_parm), the axis name (axisName) and axis number (axisNo) should be provided. The axis name must be within {x, y, z, a, b, c, u, v, w, s} and is case insensitive. The system will find the corresponding logical axis number and branch axis number based on the given axis name, and then determine the axis parameter ID.
  - c) For the compensation parameters (com\_parm), the method to determine the parameter ID is similar as that of the axis parameters.
  - d) Device parameters and data table parameters cannot be written in the .xml file as configurable parameters.
2. The two types of configured .xml files can be displayed on the parameter configuration interface. The user .xml file is displayed by default. Debugging .xml file can be displayed after the debugging personnel enters correct password. After entering the parameter configuration interface, the parameters can be modified and saved. If the system prompts you that there is no permission to modify the file, you may enter the corresponding password and then modify the parameters. See the figure below:



#### 4.4.6 Time

##### Operation procedure

If you select to display the system time in the machine parameters, you may use the function to reset the system time.

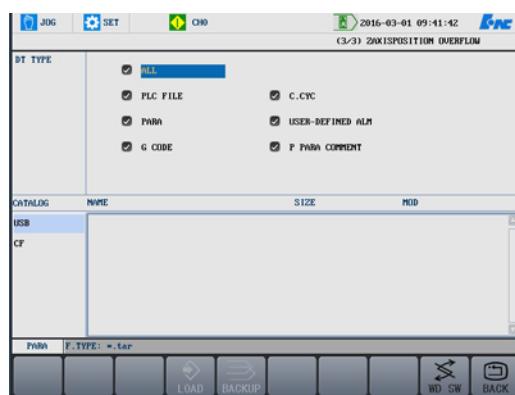
1. Press **Set** → **PARA** → **TIME** to enter the system time settings mode.
2. Use the cursor to select the time.

3. Press **Enter** to enter the editing status, where users may input data.
4. Press **Enter** again to save the settings.

#### 4.4.7 Batch Debugging

exported in batches.

1. Press **Set → PARA → ALL.**



2. By default, the system will select all data types. Press **Enter** to cancel some data types.
3. Press **WD SW**, then the cursor moves to the file directory area, where USB flask disk or CF card can be selected.
4. Press **BACKUP**, and the message whether you back up the file of the selected data type is displayed. Press **Y** and the system prompts successful backup.
5. Users may press **LOAD** to load files as required.

Debugging Files can be importd or

#### Attention

**The function is avialabel to machine users, CNC manufacturers and administrator.**

1. If you back up the file to the CF card, the system will automatically name the file as **fileman\_backup.tar**.
2. If you back up the file to the USB flask disk, the system will automatically name the file as **fileman\_backup\_YMMDDTIME.tar**.

3. Power off is not allowed during backup.

#### 4.4.8 Data Management

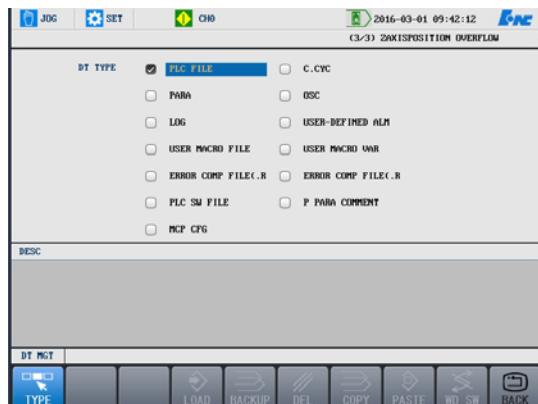
The function is used to load/back up parameters, PLC, fixed cycle, logs, compensation, and oscilloscope files.

**Note 1:** This section takes system parameter files to describe the loading/backing up operation procedure. Other files (except for error compensation files) follow the similar operations.

**Note 2:** Power off is not allowed during back up.

**Note 3: This function is available only to machine users, CNC users and administrator.**

1. Press Set → PARA → DT MGT.

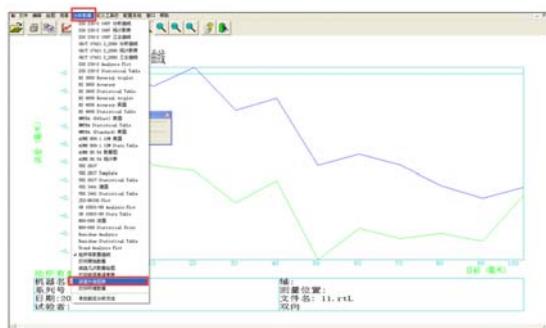


2. Move the cursor to select the data type to be imported or backed up, press **Enter**.
3. Move the cursor to select the file to import or back up. 批注 [A5]: 原文重复,删否
4. Press **WD SW**, and move the cursor to the file path where the files will be imported or backed up.
5. Press **Enter** or **BACKUP** again.
6. Press **COPY** and **PASTE** to copy and paste the files.

**Error compensation file import/backup** HNC-8 series supports automatic import of pitch error file generated by Renishaw laser interferometer.

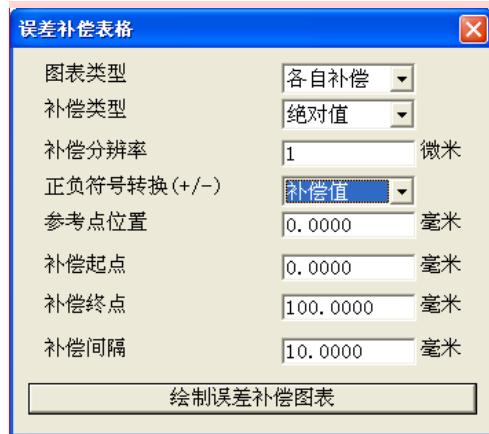
1. Generate pitch error file **LEC.REN**.

Open the \*.rtl file generated by Renishaw laser interferometer, select **Analyze Data** from the menus, select **Error Compensation Chart** from the drop-down list box, and set parameters in the dialog box that appears.



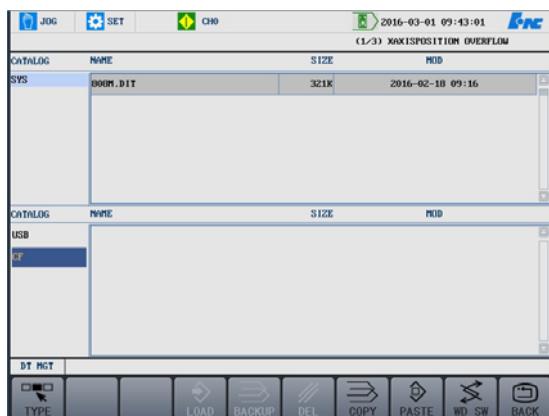
2. Conduct settings as following instructions:

- a. Set **Chart Type** as **Individual Compensation**.
- b. Set **Compensation Type** as **Absolute Value**.
- c. Set **Compensation Resolution** as 1 micrometer or **0.001** millimeter.
- d. Set **Positive and Negative Symbol Change (+/-)** as **Error Value** or **Compensation Value**.
- e. Set **Reference Position** as **0**.
- f. Set **Compensation Start Point** as the position of compensation start point.
- g. Set **Compensation End Point** as the position of compensation end point. The start point must be less than the end point.
- h. Set **Compensation Interval** as compensation interval, which must be a positive value.
- i. Click "Draw Error Compensation Chart" to generate the LEC.REN file. It is a temporary file which is saved in the installation directory of the Rainshaw software (generally C:\Program Files\Renishaw LaserXL). Copy the LEC.REN file to the USB flask disk.



批注 [z6]: 中文图

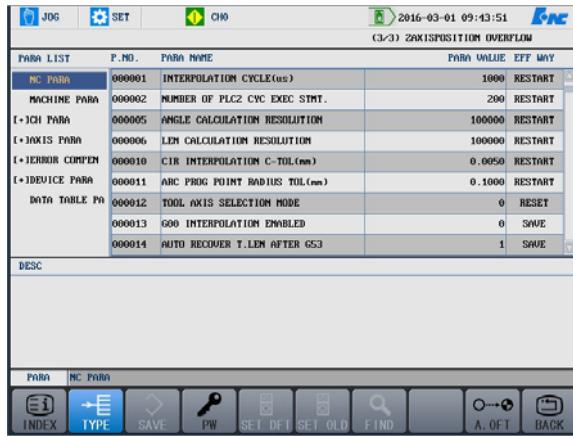
3. Press Set → PARA → DT MGT, select Error Compensation File, and press Enter.
4. Press WD SW, select USB flash disk or CF card, select the LEC.REN file, and press Import.



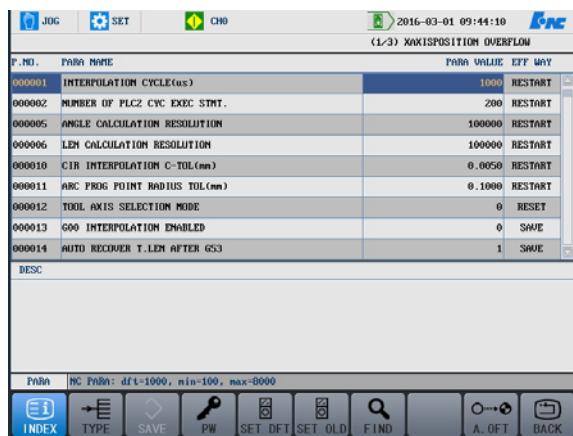
5. Enter the axis number where the pitch error data will be imported, and press Enter to import the data.



6. Press Set → PARA → NC PARA → ERROR COMPEN, press Enter to expand the compensation parameter table to view the corresponding pitch error compensation parameters. For example, Compensation axis 0—300020, 300021, 300022, 300023, and 300026 are all configured.



7. Press Set → PARA→ NC PARA → DATA TABLE PARA to view corresponding parameters. (The compensation values in the pitch error compensation file .REN are written to the data table parameters of (710000+ axis number\*1000)).



- Rules for unidirectional pitch compensation data table:  
negative return to zero, then write the compensation value of "Positive Machine Movement" into the data table started with (710000+ axis number \* 1000) in the positive sequence;  
otherwise, write the compensation value of "Negative Machine Movement" into the data table started with (710000+ axis number \* 1000) in the positive sequence.
- Rules for bidirectional pitch compensation data table:  
negative return to zero, then write the compensation value of "Positive Machine Movement" into the data table started with (710000+ axis number \* 1000) in the positive sequence, and

then write the compensation value of "Negative Machine Movement" into the data table in the positive sequence.

Otherwise, write the compensation value of "Positive Machine Movement" into the data table started with (710000+ *axis number* \* 1000) in the positive sequence, and then write the compensation value of "Negative Machine Movement" into the data table in the positive sequence (For positive return to zero, all input data should be minus the last value in the "Negative Machine Movement" column).

#### 4.4.9 Permission Management

After installing and testing the system, it's unnecessary to modify the parameters. If you need to modify certain parameters, select the appropriate user level and enter the correct password for modification. The password itself can also be modified, provided that you enter the correct password.

##### User level

Parameter configuration has significant effects on the system performance, so the system parameter modification is strictly limited. User permissions of this system can be divided into four categories: operator, workshop administrator, machine manufacturer, CNC manufacturer, and system administrator.

"**OPERATOR**" permission indicates no permission. When other permissions are canceled, the system will automatically change the permissions to the **OPERATOR** permission. The **000359** parameter can be used to set the default permission after the system boots. If the parameter is set to **0**, the default permission is **WS ADMINS**; if the parameter is set to **1**, the default permission is **OPERATOR**.

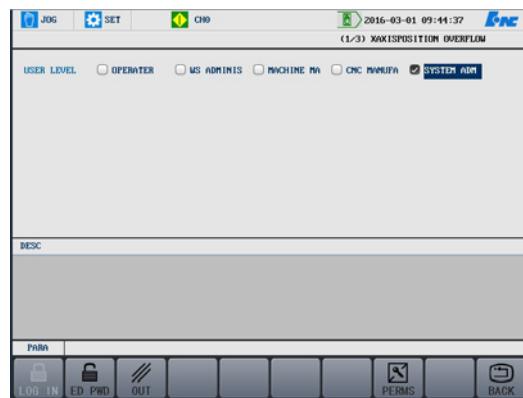
##### User Logoff

Press **Set** → **PARA** → **PERM MGT** → **OUT**, a user type can be reselected.

##### Enter password

1. Press **Set** → **PARA** → **PERM MGT**.

2. Select a corresponding user's type, and press **LOG IN**.
3. Enter the password, and press **Enter**.
4. If the permission password is correct, the parameters and password of the corresponding level can be modified. Otherwise, the system will prompt that the password is incorrect.



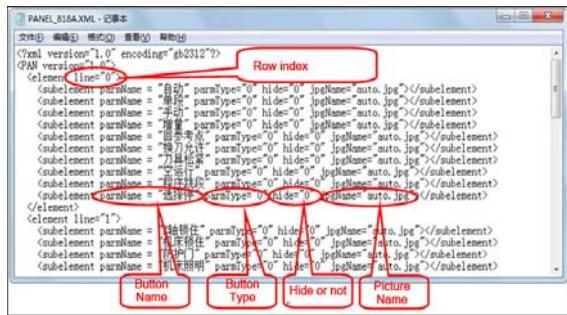
#### Modify password

1. After entering correct permission password, press the **ED PWD** key.
2. Enter the new password in the editing box, and press **Enter**.
3. Enter the same password again, and press **Enter** again.
4. After that, the password is successfully changed.

#### 4.4.10 Permission Configuration

##### Permission configuration file

The default permission configuration file is the **PANEL\_818A.XML** file in the **bin** directory, which can configure the permission of all user levels. Its format is as below:



**Line:** Line index (**0-6**), which can be discontinuous. It's unchangeable that “Line= **6**” is used for interface menu configuration.

**paraName:** Key name, up to 4 Chinese characters.

**ParaType:** Key type. The value **0** indicates text while **1** indicates picture.

**Hide:** Set whether to hide or display. The value **0** indicates to display while **1** indicates to hide.

批注 [z7]: 中文图

**jpgName:** Key picture name, valid only when **ParaType** is **1**.

The quantity of sub-element in each element group cannot exceed **10**.

### Operations on permission configuration

With the permission configuration file, you may configure the operation permission for all levels through the **PERMS** menu.



The general principle is that the user with higher permission can configure the permission for the users with relatively lower permission.

### Operations on interface

Up, Down, Left, Right keys: to move the position of the cursor.

**Enter:** define the value of the position where the cursor is. If the current status is **ON**, set it to **OFF**; if the current status is **OFF**, then set it to **ON**.

The settings will be saved in the NC parameters (**360** to **377**).

The configuration for users with different permissions are separately saved. What is modified is effective to the users with subordinate permissions, but ineffective to the users with the same permissions.

The configuration with different permission is saved as follows:

- NC parameter 360-362: current configuration
- NC parameter 363-365: users
- NC parameter 366-368: machine manufacturer
- NC parameter 369-371: CNC
- NC parameter 372-374: administrator
- NC parameter 375-377: reserved

**Change user permission with external PLC**

In addition to the permission management function, you may quickly change the user permission through external I/O signal, e.g. user permission configuration key.

The method is to write value into the G2979 register.

- Value 0: operator permission
- Value 1: workshop manager permission
- Value 2: machine manufacturer permission
- Value 3: CNC manufacturer permission
- Value 4: system administrator permission

After permission modification, a customized 121 event should be sent in order to inform the system that the permission has been changed.

**Switch permission between OPERATOR and WS ADMINIS by using the key lock mode**

In the actual production process, the workshop operator can only repeat the fixed cycle operation, and has no right to modify the program or feed override. The workshop administrator has right to modify the program or tool compensation. On site, use the key lock mode to switch between the two kinds of permission in order to ensure normal operation. The following is a PLC programming example: X486.6 is the key lock input point.

#### 4.4.11 Network Transmission

**Note:** The network connection is disabled by default. Set the parameter [000050] to enable it.

There are two network transmission modes:

1. Network shares: Access files through the Windows system network sharing.

Note: The prerequisite is that the administrator user exists in the Windows system and the password is blank.

- a) Set shared folder in the Windows

- Set shared folder in Windows XP operating system

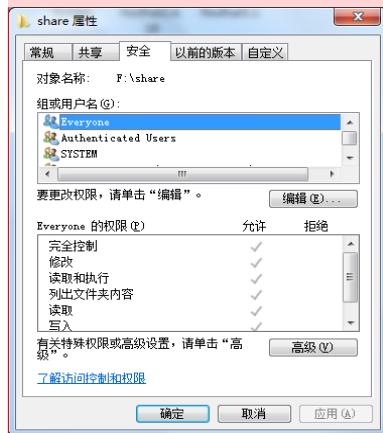
Right-click the folder, select **Properties**, click **Sharing**, Select **Share this folder**, set **Share name**, select **Allow** for **Change**, click **Apply**, and then click **OK**.



**批注 [z8]: 中文图**

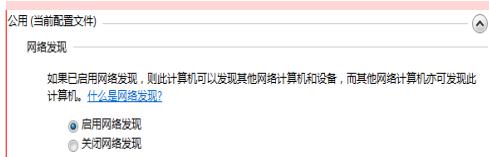
- Set shared folder in Windows 7 operating system

Right-click the folder, select **Properties**, click **Security**, click **Edit**, click **Add** to add the **Everyone** object type under **Group or user name**, and then select **Full Control** under **Allow for Permission for Everyone**.

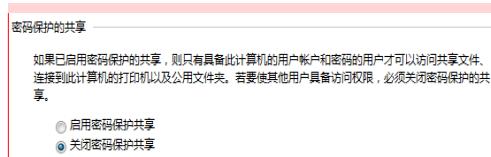


**批注 [z9]: 中文图**

Right click the folder, select **Properties**, click **Sharing**, and click **Network and Sharing Center**. On the following page, select **Turn on network discovery** and **Turn off password protected sharing**.



批注 [z10]: 中文图



批注 [z11]: 中文图

- b) Press **Set** → **PARA** → **COM**, and move the cursor to select a network disk mapping type.
- c) Press **Enter**, and select shared disk.

**Note:** It is not supported to copy and paste files to the shared disk through network sharing.

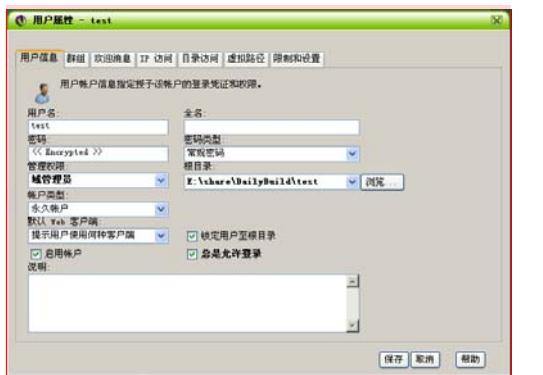


- d) Move the cursor to the server IP address, press **Enter**, and enter the IP address of the PC where the shared folder is.
- e) Move the cursor to the IP address of the local PC, press **Enter**, and then enter the IP address of the current CNC system.
- f) Move the cursor to the server root directory, press **Enter**, and enter the shared folder name on the PC.
- g) Click **Network Connection**, and the system prompts that the network has been connected.
- h) File download: **Program** → **Select** → **NET** → **Right arrow key** → **Copy** → **Left arrow key** → **CF card/USB flash disk** / **System disk** → **Paste**. Message: Paste success. The file is successfully downloaded.

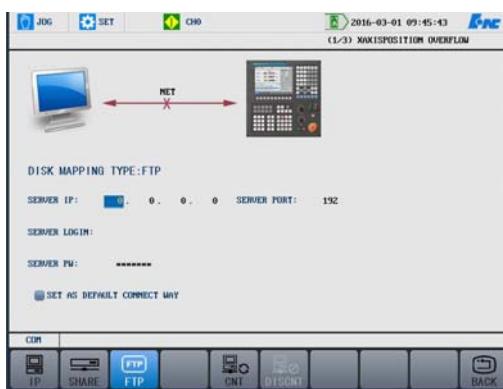
**Note: The sharing disk does not support file upload.**

2. FTP server (CNC system as a client): You need to install a third-party software as the FTP server.

- a) Install **SERVER-U (V7.3.0.0)** on the server. Refer to the official user manual of SERVER-U for detailed settings. As shown in the figure below, on the user information tab page, enter the user name and the password which must be the same with the login password of the server. Set the **Root Directory** to the path of the file to be uploaded or downloaded, and then click the **Directory Access** tab page.



- b) Press **Set** → **PARA** → **COM**, move the cursor to select a network disk mapping type.
- c) Press **Enter**, and the select **FTP**.



- d) Move the cursor to the server IP address, press **Enter**, and enter the IP address of the PC where serve-u is installed.
- e) Move the cursor to the server port, press **Enter**, and enter **21**.  
conflict with other IP addresses of the LAN.
- f) Move the cursor to the IP address of the local PC, press **Enter**, and enter the IP address of the CNC system, which cannot
  - g) The username, password, and root directory name of the server is the user information on serve-u.
  - h) Press **Network Connection**, and then the system prompts you that the network is connected.
  - i) File download: **Program** → **Select** → **NET** → **Right arrow key** → **Copy** → **Left arrow key** → **CF card/USB flash disk/System disk** → **Paste**. Message: Paste success. The file is successfully downloaded.
  - j) File upload: **CF card/USB flash disk** → **Right arrow key** → **Copy** → **NET** → **Paste**. Message: Paste success. The file is successfully uploaded.

#### 4.4.12 System Update

##### Attention

manufacturer and administrator.

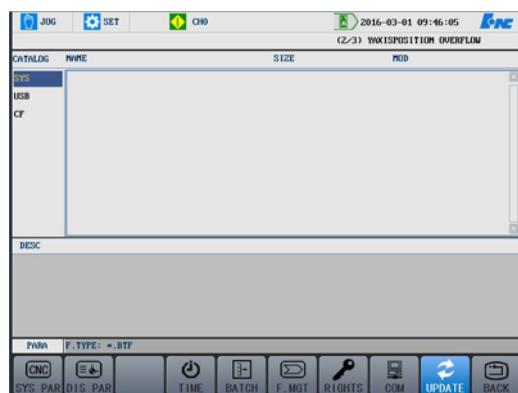
2. There may be risks in the update. Please back up all machine PLC files, parameter files, alarms, text, cycle programs and other documents before the system update.
  3. After the upgrade, the operations on breakpoint must follow the instructions below:
    - a) Do not use the breakpoint file which is generated before the update.
    - b) It is recommended to conduct update after the machining. If you use the breakpoint file generated before the update, it may cause problems such as system crash.
  4. Power off is not allowed during the update.
1. This function is limited to CNC

**Operation procedure**

1. Press **Set** → **PARA** → **UPDATE** to enter the system update interface.

**Note:**

- 1: Press **Alt+C** and **Alt+V** to copy and paste information.
  - 2: You can press **DEL** to delete **BTF** package only in the **CF card** directory.
2. Move the cursor to select the file to be updated, and press **Enter** to enter the interface shown in the figure below:



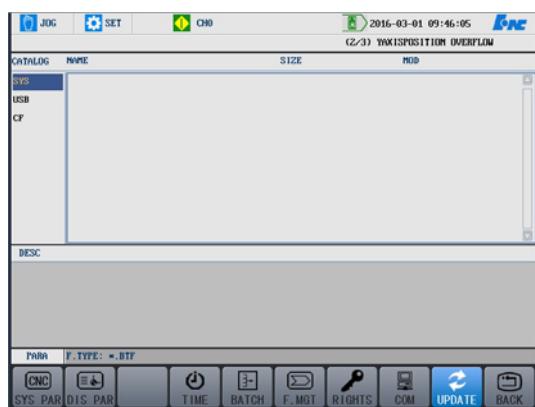
**Note:** Parameter update can be ONLY used by the debugging

personnel. After the parameter structure file is changed, the debugging personnel can conduct the parameter update.

3. Use the cursor to select an update type and conduct system backup.

**Note:** The default directory for backup is the **CF card** directory.

4. Successively press **Y** and **Enter** to confirm the update, or **N** and **Enter** to cancel the update.



## 5 Program Editing and Management

---

---

Under the program menu, you may edit, save programs and beyond. The chapter include the following sections:

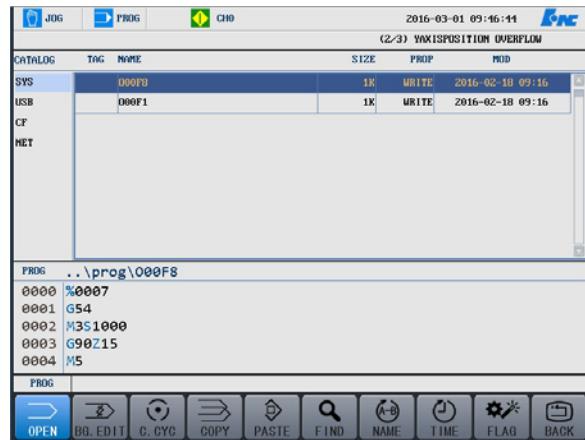
- **Select Programs**
- **Edit Programs**
- **Program Management**
- **Random line**
- **Verify Programs**
- **Stop Running**
- **Restart Program**
- **Craft File**
- **Workpiece Measurement**

## 5.1 Select Programs

### 5.1.1 Select a File



Under the **Prog** menu, press **OPEN** to enter the interface as shown in the figure below:



Follow the instruction below to select a program file:

1. Press **▲** and **▼** to select a storage type (system disk, USB flash disk, or CF card), and press **Enter** to view the sub directories under the storage.
2. Press **►** key to move to the program file list.
3. Press **▲** and **▼** keys to select a program file as required.
4. Press **Enter** to select the file and add it to the machining buffer.
5. If the selected program file is read-only G code file, the **[R]** label is displayed.

#### Attention

1. If users select no program file, the system selects the machining program that is saved in the buffer last time.
2. The program file name generally starts with **O** (by default), followed by four (or more) numbers or letters.
3. The program file name supported by NNC-818 system is in the "8 + 3" format: the name consists of one to eight letters or numbers, plus the extension of 0-3 letters or numbers, e.g. "MyPart.001", "O1234".

- Loading and unloading of USB flash disk**
1. Use the cursor to select “USB” in “Catalog”..
  2. Press **Enter** to load the USB flash disk.
  3. Press **DEL** to unload the USB flash disk.

**Attention** **Unplug the USB after unload it.**

### 5.1.2 Background Editing

- Background editing** Background editing indicates that other program files can be edited while the system is conducting machining.
1. Select a program by following the instructions in section 5.1.1.
  2. Press “**BG EDIT**” to enter the editing status. For detailed information about editing a program, see section 5.2.

- Background Creation** Background creation indicates that a new file can be created while the system is conducting machining.
1. Press **Prg** → **OPEN** → **BG EDIT** → **NEW**.
  2. Enter the file name.
  3. Press **Enter** to create a file.

### 5.1.3 Fixed Cycle

- Operation procedure**
1. Press **Prg** → **OPEN** → **C.CYC**, and the system displays the fixed cycle files.
  2. Use the cursor keys to select a file as required.
  3. Press **Enter** to load the file.

**Attention** **This function is available only to the machine manufacturer, CNC manufacturer, and administrator.**

### 5.1.4 Copy and Paste Files

You may copy and paste a file to a specified path.

1. Press **Prog** → **OPEN**, and select a file to copy.
2. You may press **FLAG** to select multiple files.
3. Press the **PASTE** key.
4. Select a target folder (This cannot be the same folder where the file is selected.)
5. Press **PASTE** to paste the file.

### 5.1.5 Search Files

#### Operation procedure

You may search files based on the entered file name.

1. Press **Prog** → **OPEN** → **FIND**.
2. Enter the file name, and press **Enter**. The searched file will be highlighted.

### 5.1.6 Sort Files

#### Operation procedure

You may sort files by time/name.

1. Press **Prog** → **OPEN** → **NAME** to sort files by file name.
2. Press **Prog** → **OPEN** → **TIME** to sort files by time.

### 5.1.7 Encryption for G Codes

#### Operation procedure

1. Press **Set** → **PARA** → **NC PARA** → **TYPE** → **MACHINE PARA**.
2. Move the cursor to select parameter **010299**—G code file key, press **Enter**, and enter the G code file key.
3. Press **Enter** again to confirm the operation.
4. Press **Save** and restart the system.

5. Run the software of G code file key on the PC. See the figure below:



批注 [z13]: 中文图

6. Encryption: Click **Browse**, select a G code file, enter the file key above, and click **Encrypt**. The system generates an encrypted G code file.
  - Read-only: The encrypted file cannot be modified.
  - Writable: The encrypted file can be edited.
  - Not display: The system does not display the encrypted file content.
7. Decryption: Click **Browse**, select a G code file, enter the file key above, and click **Decrypt**.



批注 [z14]: 中文图

**G code encryption type**

<b>[G Code Encrypted File Key Mapped]</b>	<b>Display Function</b>	<b>Edit Function</b>	<b>Operation Function</b>
Not display	Blank	Not support	Allowed
Writable	Able to be Browsed	Editable	Allowed
Read-only	Able to be Browsed	Not support	Allowed

<b>[G Code Encrypted File Key Not Mapped]</b>	<b>Display Function</b>	<b>Edit Function</b>	<b>Operation Function</b>
Not display	Blank	Not support	Not allowed
Writable	Unreadable	Not support	Not allowed
Read-only	Unreadable	Not support	Not allowed

## 5.2 Edit Program

---

### 5.2.1 Edit Program

**Programs existed in the system buffer**

- Press **Prog → EDIT** to edit the currently loaded file.

**No program existed in the system buffer**

- Press **Prog → EDIT**. The system automatically creates a file. Press **Enter** to edit the created file.

**Shortcut key**

- **Del**: Delete the character right to the cursor and maintain the cursor position. The remaining characters right to the cursor will shift leftwards accordingly with one-character space.
- **PgUp**: Move the program to be edited one screen up to the program header and maintain the cursor position. If it has reached the program header, the cursor will move to the first character in the first row of the program.
- **PgDn**: Move the program to be edited one screen down to the program end and remain the cursor position. If it has reached the program end, the cursor will move to the first character in the last row of the program.
- **BS**: Delete the character left to the cursor and the cursor will shift leftwards with one-character space. The remaining characters right to the cursor will shift leftwards accordingly with one-character space.
- **◀**: The cursor shifts leftwards with one-character space.
- **▶**: The cursor shifts rightwards with one-character space.
- **▲**: The cursor shifts one row upwards.
- **▼**: The cursor shifts one row downwards.
- **ALT+B**: Define block header.

- **ALT+E:** Define block end.
- **ALT+D:** Delete a block.
- **ALT+X:** Cut
- **ALT+C:** Copy
- **ALT+V:** Paste
- **ALT+F:** Find
- **ALT+N:** Find next
- **ALT+R:** Replace
- **ALT+L:** Delete a line.
- **ALT+H:** Move to file header
- **ALT+T:** Move to file end

**Attention**

**You must execute the file again after editing.**

### 5.2.2 Create Program

**Operation procedure**

1. Press **Prog** → **EDIT** → **NEW**.
2. Enter a file name, and press **Enter** to create a file.

**Attention**

1. The default directory for created files is the **prog** directory in the system disk.
2. The created file name cannot conflict with an existing one.

### 5.2.3 Save Program

**Operation procedure**

Press **Prog** → **EDIT** → **SAVE** to save a specific file.

**Attention**

When the program is a read-only file, the system will prompt you that the file saving is failed when pressing the **SAVE** key. In this case, you may only use the **SAVE AS** function.

### 5.2.4 Save as

**Operation procedure**

1. Press **Prog** → **EDIT** → **SAVE AS**.
2. Use the cursor keys to select the target folder.
3. Press **►** to move to the input box, and enter a file name.
4. Press **Enter** to save the file as another file.

### 5.2.5 Block Operation

**Operation procedure**

1. Press **Prog** → **EDIT** → **BLOCK EDIT**.
2. The block operation function is the same as the shortcut function described in section 5.2.1.

### 5.2.6 Search a String of Characters

**Operation procedure**

Search files based on the entered keywords.

1. Press **Prog** → **EDIT** → **FIND**.
2. Enter the keywords, and press **Enter**. The system highlights the searched characters.
3. Press **CONT**. The system highlights the next searched characters.

### 5.2.7 Replace a String of Characters

**Operation procedure**

1. Press **Prog** → **EDIT** → **REPLACE**, enter the characters to be replaced.
2. Press **Enter**.
3. Enter the characters used to replace the previous ones.
4. Press **Enter**. The system will ask whether to replace all.

- Press **Y** to replace all searched characters.
  - a. Press **Y** to replace the currently searched characters.
  - b. Press **N** to cancel the current replacement.
- Press **N**. The system will ask whether to replace

## 5.2.8 Change File Properties

### Attention

**The function is available only for the machine manufacturer, CNC manufacturer and administrator.**

### Operation procedure

1. Load the file to the machining buffer (see section 5.1.1).
2. Press **Prg** → **EDIT** → **ALLOW/NOT ALLOW**.
  - Not allow editing: Allow to view the program code, but not allow to modify it.
  - Allow editing: Allow to edit the program.

## 5.3 Program Management

---

### 5.3.1 Delete Program File or File Folder

#### Operation procedure

1. Press **Prog** → **P.MGT**, and select a drive where the program file is.
2. Use the direction keys to move the cursor to the program file or file folder to be deleted.
3. Press **Del**, the system displays the message to prompt you whether to delete the current file or file folder. Press **Y** to delete it from the current storage, or press **N** to cancel the deletion.

#### Attention

The current running program cannot be directly deleted.

You may use the **FLAG** function to select multiple programs to delete.

The deleted file cannot be restored.

### 5.3.2 Copy and Paste Programs

#### Operation procedure

You may use this function to copy and paste a file to a defined folder.

1. Press **Prog** → **P.MGT**, and select a file to be copied.
2. Press **COPY**.
3. You may press **FLAG** to select multiple files.
4. Select the target folder (The folder cannot be the same as the current one).
5. Press **PASTE** to paste the file to the defined folder.

### 5.3.3 Search Programs

**Operation procedure**

You may search files based on entered file name.

1. Press **Prog** → **P.MGT** → **FIND**
2. Enter the file name to be searched, and press **Enter**. The system highlights the searched results.

### 5.3.4 Change Program Name

**Operation procedure**

1. Press **Prog** → **P.MGT** → **RENAME**.

2. Enter the new file name in the editing box.
3. Press **Enter**.

**Attention**

The name of the program which is under machining cannot be modified.

### 5.3.5 Sort Programs

**Operation procedure**

Programs can be sorted by time/name.

1. Press **Prog** → **OPEN** → **NAME** to sort files by file name.
2. Press **Prog** → **OPEN** → **TIME** to sort files by file time.

### 5.3.6 Create a Program File or File Folder

**Operation procedure**

different disks.

1. Press **Prog** → **P.MGT**. The system displays the program file list.
2. Use the cursor keys to select a disk:"System Disk", "USB", or "CF Card".
3. Press **Enter** to create a folder in the selected directory.
4. Press **New Folder**, and enter a folder name.
5. Press **Enter** again.

You may create sub directories in



## 5.4 Random Line

### Scanning mode

Before running program, the system scans all modal information above a random line. The specified line inherits the coordinates, tool compensation and other information of the previous lines. There are two scanning modes: scanning without Z axis return, scanning with Z axis return. If the program is large, it may take a long time for scanning the modal information. Sub program scanning is supported.

### Non-scanning mode

The system does not compute the previous modal information of a random line. The mode is the default system status.

### Parameter settings

Parameter settings related to random line operations:

- 040113: The order of axis motion for random line.

Parameter No.	Description	Value	Effective Mode
040113	Random line scanning	1	Save

Parameter value: 0: Non-scanning mode (default); 1: Scanning without Z axis return; 2: Scanning with Z axis return

Example:

*G54*

....

*Z10*

*N1X10y10*

....

*M30*

When *040113=1*, and the random line is specified as **N1**; only X/Y axes are moved.

When *040113=2*, and the random line is specified as **N1**; X/Y/Z axes are moved.

- 040114: The order of axis motion for random line.

The relationship between the parameter value and axis is as below:

1	2	3	4	5	6	7	8	9
X	Y	Z	A	B	C	U	V	W

The value from **1** to **9** corresponds the axis **X/Y/Z/A/B/C/U/V/W** respectively. The **0** indicates that the axis is not configured.

For milling machines, when *040114=211*, firstly the X/Y axes move in place, and then the Z axis moves.

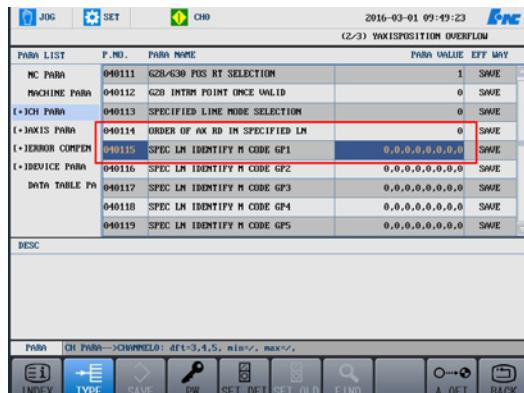
For lathe machines, when *040114=101*, the X/Z axes move in place simultaneously.

- For channel parameters 040115-040124, M code group can be identified by random line.

The type is BYTE[8], supporting up to eight numbers, which must be separate by the "," or "." symbol.

Only the M command in parameters can be restored. Therefore, a maximum of 10 M command groups is supported, and each group has eight M codes.

For example: Set M command group 1 to **3, 4, 5**, which indicates that M3/M4/M5 are in the same group.



### Attention

1. For the M commands of the same group obtained in the scanning mode, only the last M command is valid.
2. For the T commands obtained in the scanning mode, only the last T command is valid for lathe machines.

3. For the armless type ATC of milling machines, M6 should be in the same line with T codes.
4. For the S commands of the same group obtained in the scanning mode, only the last S command is valid. A maximum of four S commands is supported for the channel.

### 5.4.1 Scanning Mode

#### Operation procedure

1. Press **Prog** → **BLOCK** → **SCAN** to enter the scanning mode.



The system displays the scanned M/S/T value and its line number on the interface for users to view.

1. Scanned axis position
2. Scanned spindle speed, feed rate, and M command
3. Current G modal commands
4. Scanned tool compensation number, T command, and line number

#### 5.4.1.1 Specify Line Number

#### Operation procedure

1. Press the **Feed Hold** key on the control panel (the indicator is on) to enter the feed hold mode.
2. Press **Prog** → **BLOCK** → **LN NO.**, and enter the number of the line from which the execution starts. See the figure below:



3. Press **Enter**.
4. Press the **Start** button on the control panel to start the operation from the specified line.

#### 5.4.1.2 Specify N Number

**Operation procedure**

control panel (the indicator is on) to enter the feed hold mode.

2. Press **Prg** → **BLOCK** → **RED**, and enter the N number from which the execution starts. See the figure below:



3. Press **Enter**.
4. Press the **Start** button on the control panel to start the execution from the specified line.

1. Press the **Feed Hold** key on the

**Attention**

**when the following conditions are met:**

1. N number exists in the G code file.
2. The total G code lines can be displayed.



3. The program is less than 22M.

This function is available only

**5.4.1.3 Find****Operation procedure**

Search keywords, and start execution from the line where the keyword is.

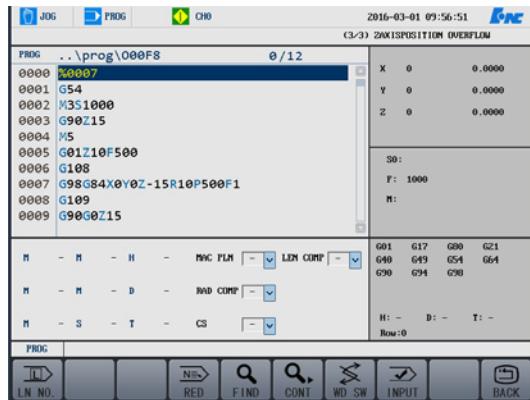
1. Press **Prog** → **BLOCK** → **FIND**.
2. Enter the keyword, and press **Enter**. The searched keyword is highlighted.
3. Press **CONT** to find the next keyword.
4. Press **Enter** again. The cursor points to the line where the keyword is.
5. Press the **Start** button on the control panel to start the execution from the specified line.

## 5.4.2

### 5.4.3 Non-scanning Mode

#### Operation procedure

SCAN to enter the non-scanning mode.



2. If you press the "WD SW" button, you can switch to the lower part of the window to enter the M/S/T code. You can enter five M commands and other commands. Press the **Enter** key to confirm the input, then press the **Start** button. The system will execute the M/S/T command first, and then execute the specified line.

1. Press **Prog** → **BLOCK** → **Non**

#### 5.4.3.1 Specify Line Number

#### Operation procedure

See section 5.4.1.1.

#### 5.4.3.2 Specify N Number

#### Operation procedure

See section 5.4.1.2.

#### 5.4.3.3 Find

#### Operation procedure

See section 5.4.1.3.

#### Attention

**Don't move the cursor over the subprogram during the random line operation.**

## 5.5 Verify Program

---

This function is used to verify the programs in the machining buffer and report errors if there is any.

New programs that have never been run on the machine should be verified before auto running.

Operation procedure

1. Call the program that need to be verified (**Prog → OPEN**).
2. Press the **Auto** or **Single Block** key on the control panel to enter the program operation mode.
3. Under the **Prog** menu, press **Verify**. The work mode displayed on the system interface changes to **Auto Verification**.
4. Press the **Start** button on the control panel to start the program verification.
5. If a program error occurs, the system displays the message where the error is.

### Attention

1. The machine doesn't run during program verification.
2. It is recommended to select difference graphs for verification to ensure correct program. For detailed information, see section 7 (Graph Display).
3. The verification speed depends on the feed adjustment override. It may help you to view the tool path.

## 5.6 Stop Program

---

### Operation procedure

This function is used to pause the running during program working.

1. Press **Prog** → **Stop**. The system prompts you that the operation is paused and whether to cancel the current program (Y/N).
2. Press **N** to pause the current operation, but save the current program modal information (In this case, you may press the **Start** button to proceed the program from the pause point).
3. Press **Y** to stop the program and unload the current program modal information (In this case, users must select the program and restart it from the beginning).

## 5.7 Restart Program

---

### Operation procedure

This function is used to restart a program from the beginning after the program is stopped.

1. Press **Prog** → **REWIND**. The system prompts you whether to restart the current program (Y/N).
2. Press **N** to cancel the restart.
3. Press **Y**. The cursor moves to the beginning of the program. Press the **Start** button to restart the program from the beginning.

## 5.8 Craft File

---

### Craft file display and management

Craft file is used to provide the operator with auxiliary information about processing. Craft file and G-code files are matched and bound via filename. After loading G code, you may browse the craft file which matches the G code to know the craft flow.

#### Design craft file

1. You should finish the craft file when creating G codes.
2. The craft file uses 24-bit bmp pictures with color 790\*440 resolution.
3. The craft file is associated with the program, so the craft file is named as the main program name (removing 'O' or 'o' from the name) plus the craft file number (0 to 9).

Example 1: If the main program name is "OA347F.nc", then the craft file name should be "A347F\_0.bmp" to "A347F\_9.bmp".

Example 2: If the main program name is "oWErt6", then the craft file name should be "WErt6\_0.bmp" to "WErt6\_9.bmp".

Note: The suffix of the .bmp file is lowercase of "bmp".

4. A program can have up to 10 craft files.

### Transmit craft files to the CNC system

After the G code file of the main program is transmitted from the host computer to the CNC device, the craft file is also transmitted from the host computer to the CNC device.

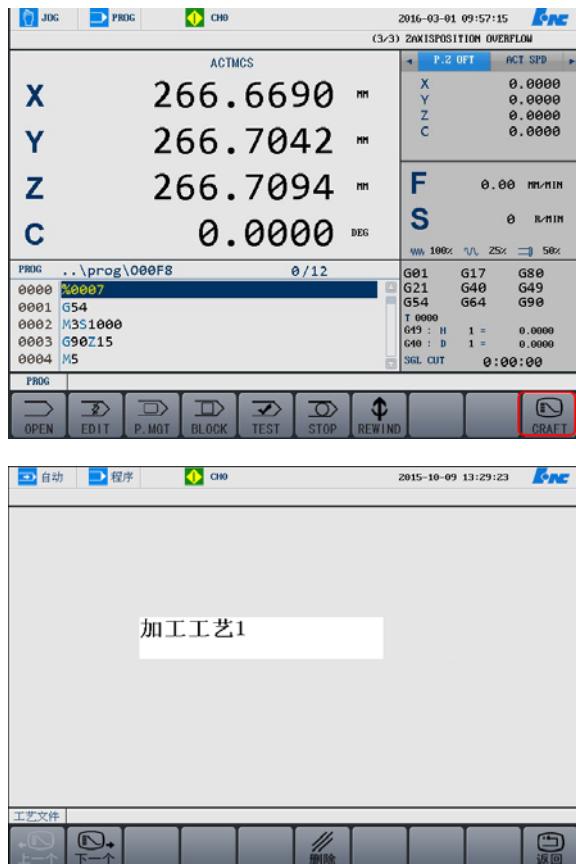
Attention: The craft file and main program file are in the same directory.

Example: The main program is transmitted to the **prog** directory, and the craft file is also transmitted to the **prog** directory.

The main program is transmitted to the **123** folder under the **prog** directory, and the craft file is also transmitted to the **123** folder under the **prog** directory.

**View craft file**

- After automatically or manually loading programs, you may press **CRAFT** to view the craft file corresponding to the loaded main program. See the figure below:



- If there are multiple craft files, press the **PREVIOUS** or **NEXT** menu to switch files.
- During program execution, you may view craft files.
- The associated craft files will be deleted when the main program is deleted.

**Delete craft files**

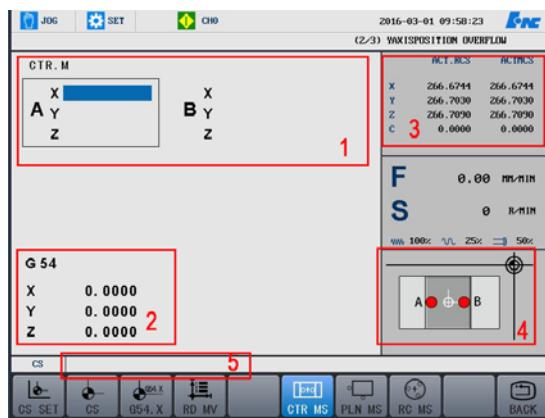
- Workshop manager or higher permission may delete the craft files displayed under **CRAFT**.
- You may view or manage craft files corresponding to multi-channel programs.

## 5.9 Workpiece Measurement

Move the tool on the spindle via manual or hand-wheel feed, to make it contact with the workpiece, so that the coordinate position can be measured. Specify the measured value to the defined workpiece coordinate system (G54 – 59 or G54.XX).

### Operation procedure

1. Select **Set** on the control panel to enter the **Set** menu, press **WC MS** to enter the interface shown as the figure below:



Options on the interface:

- Display area 1—Measure value display area: displays the current measurement position and measurement mode. Select **BACK** to clear the recorded coordinate values.
- Display area 2—Coordinate system setting area: displays the coordinate system and value to be set.
- Display area 3—Position recording area: display the current relative actual coordinate system and machine actual coordinate.
- Display area 4—Guide diagram: display the schematic diagram of the current measurement mode.
- Display area 5—Value input box: When selecting **G54.X**, “Enter the coordinate system number : P”will be displayed in the input box, then enter one value of 0~60.

2. Press the **CS SET** key, then the value which can be figured out in accordance with the measured values and the measurement mode, is set to the displayed workpiece coordinate system offset. The defined axis can only be the one that have values in the measurement position.
3. Press the **CS** key to select one of the coordinate systems from G54 to G59 to define.
4. Press **G54.X**. In the input box, enter X value to define the required coordinate system. Valid value: **0-60**.
5. Press **CTR MS**: Enter the center measurement mode (by default).
6. Press **PLN MS**: Enter the plane measurement mode.
7. Press **RC MS**: Enter the circle center measurement mode.

**Description**

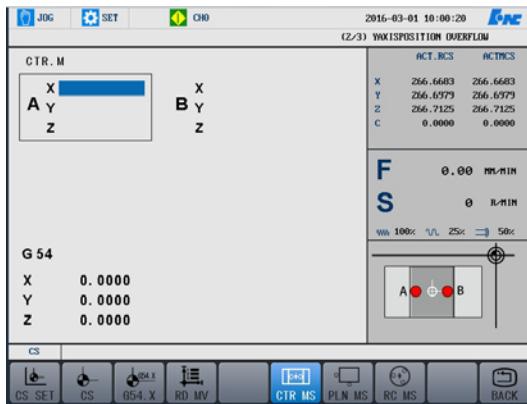
Operations on three measurement modes:

1. Center measurement

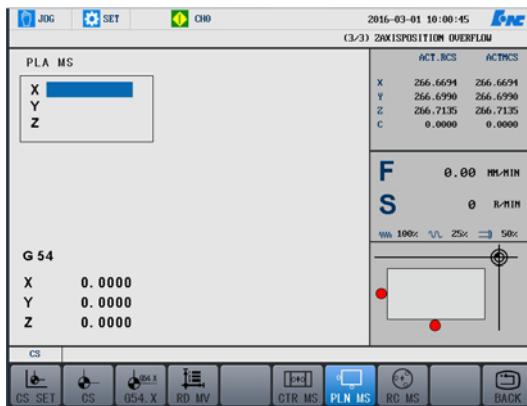
The measurement value display area is divided into Point A and Point B. The default selection box is Point A; The default axis is the X-axis; The background color is blue. Use the cursor keys to select the axis to be measured, and the left and right keys to select Point A or B.

Press the **RD MV** key, the system inputs the coordinate value corresponding to the machine actual position into the selected box with blue background, and automatically moves the selected box to the Point B.

Press the **CS SET** key. Then the value, which the system figures out via the formula of (*measured value of point A + measured value of point B*)/2 for the defined axis, is displayed the specified coordinate in area 2. If only the point A is measured while the point B is not measured, the system displays the message "reading measurement value failed".



## 2. Plane measurement



Only one point coordinate is displayed in the measurement value display area. The X axis is selected by default. The part to be edited is in blue background. You may use the Up and Down keys to move the cursor to the required axis.

Press the **RDMV** key, then the system inputs the corresponding coordinate value of the machine actual position into the coordinate of area 1. Press the **CS SET** key, then the axis whose value has been read is set to the specified coordinate of area 2.

## 3. Circle Center Measurement

Point A, B, C are displayed in the measurement value display area. The default plane is G17, default axes are X, Y axis. The X and Y coordinates of point A are in the selection box with blue background. You may move the cursor to the **Plane** and select the drop-down box, click **Enter**, and select G17, G18, and G19 planes.

The corresponding Coordinate are XY, XZ, and YZ.

Press the **RD MV** key, enter the corresponding coordinate value of the machine actual position into the selected box with blue background, and the selected box automatically moves to the next point.

Press the **CS SET** key, then the system calculates the circle center based on the three points, and input the calculated value to the specified coordinate system of the area 2.

If one of the three point coordinate is not measured, the system displays the message "Measurement failed"

## 6 Operation Control

---

---

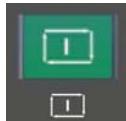
This chapter includes the following sections:

- **Start, Pause and Stop**
- **Dry Running**
- **Over-Block Operation**
- **Optional Stop**
- **Single Block Operation**
- **Breakpoint Related Operation**
- **Operation Control**

## 6.1 Start, Pause and Stop

---

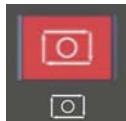
### 6.1.1 Start Auto-Operation



After a program file is loaded and verified, you may start the auto operation function.

1. Press the **Auto** key (the indicator is on) on the operation panel to enter the auto operation mode.
2. Press the **Start** button (the indicator is on) on the operation panel to automatically run the loaded program.

### 6.1.2 Pause Auto-Operation



During program operation, follow the instructions below to pause the on-going operation:

1. During program operation, press the **Feed Hold** key (the indicator is on) to enter the Feed Hold mode and pause the on-going operation.
2. Press the **Start** button (the indicator is on) to proceed with the auto operation.

### 6.1.3 Stop the Auto-Operation

#### Operation procedure

During auto operation, you may Follow the instructions below to stop the auto operation:

1. During program operation, press the feed hold key (the indicator is lit) to enter the Feed Hold mode and pause the on-going operation.
2. Press the **Manual** key to disable the M and S command functions.
3. If you want to exit the system, press the **Emergency Stop** key to stop the running program.
4. If you want to stop the running program but do not want to exit the

system, press **Prog** → **Reset**,  
and reload a program.

## 6.2 Dry Running

---

**Attention**

This function is invalid to thread cutting.

Dry running can be activated only in the non-automatic or non-single block mode.



panel (the indicator is on), and the CNC system enters the dry running mode. The feed rate specified in the program is ignored, and the feed motion is at the dry running speed specified by the parameter. The feed rate is limited by the maximum machining speed and controlled by the feed override.

The Dry Running function does not conduct actual cutting, and is intended to confirm the cutting path and program.

During the actual cutting, you need to disable the function; otherwise, it may causes damages.

Click the key on the machine control

### 6.3 Over-Block Operation

---



If the over-block symbol “/” exists in the program. Press the key during program running, the system will automatically skip the block of the program beginning with the “/” symbol. Release the key to disable the over-block operation.

## 6.4 Optional Stop

---



If the M01 auxiliary command is used in the program, you may press the key to stop the program at the line of M01. Press the **Start** button to continue the program. Release this key to disable the M01 command function.

## 6.5 Single Block Operation

---



Press the key to enter the Single Block mode (the corresponding indicator is on). The program will be executed block by block.

1. Press the **Start** button to execute a block of the program. The motion slows to stop and the tool stops when the block is finished.
2. Press the **Start** button again to execute the next block of the program. After the block is completed, the operation stops.

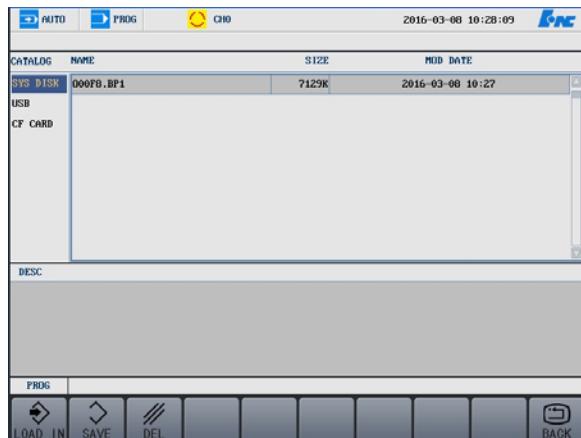
## 6.6 Breakpoint Related Operation

During machining, there are always some large workpieces which requires complicated operation and long duration. It is necessary to save and restore the breakpoint for further processing. In other words, you may save the breakpoint (let the system remember various status of the time point) before cut the power supply. After a period of time, you turn on the power supply and restore the breakpoint (let the system restore the status where the processing is broken) to proceed with the machining from the breakpoint.

### 6.6.1 Save the Breakpoint

#### Operation procedure

1. Press the feed hold key to enter the feed hold mode (the indicator is on).
2. Press **Prog → BP**.



3. Use the **▲** and **▼** keys to select a storage path (press **Enter** to view the folders in the selected storage path).
4. Press **Enter**, and the system will automatically create a file named with the current program name. You may modify the name as required.

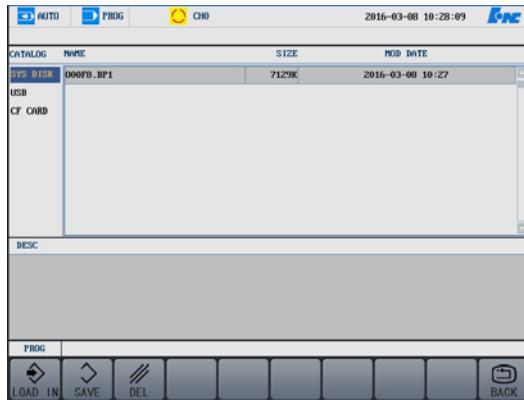


5. Press **Enter** to confirm the operation.

### 6.6.2 Resume Breakpoint

#### Operation procedure

1. If the power is off after you save the breakpoint, you need to return the tool back to the reference point after power on. Otherwise, directly press **Prog → BP**.



2. Use the cursor keys to select the directory where the target file is, switch to the file list, and select the breakpoint file to be loaded.
3. Press **LOAD IN**, the system will resume the breakpoint of the program based on the information saved in the breakpoint file.



### 6.6.3 Delete Breakpoint

#### Operation procedure

1. Press **Prog → BP**, and select a breakpoint file.
2. Press the **DEL** key, and the message as shown in the figure below is displayed:



3. Press **Y** or **Enter** to delete the breakpoint file, or press **N** to cancel the deletion.

**Note:** The deleted file cannot be restored.

### 6.6.4 Return to Breakpoint

If axis motion has been conducted after breakpoint saving, you need to return the axis to the position of breakpoint before proceeding the program from the breakpoint. Follow the instructions below:

### Operation procedure

1. Manually move the axis near to the breakpoint, and make sure no collision will occur when the tool returns to the breakpoint.
2. Under the MDI main menu, press **TO BP**. The system automatically displays the breakpoint file information.



3. Press **Start** to start running, and the system moves the tool to the breakpoint.
4. Press the **Start** button to proceed program from the breakpoint.

### Attention

Before returning to the breakpoint, you need to load the corresponding program; otherwise, the system may prompt you that the breakpoint cannot be resumed.

## 6.7 Intervention at runtime

### 6.7.1 Feed Rate Adjustment



In the Auto or MDI mode, adjust the feed rate by using the override keys when the feed rate of the F-code programming is too high or too low. The adjustment range is **0%** to **120%**.

In the manual continuous feed mode, this button can be used to adjust the manual feed rate.

### 6.7.2 Rapid Traverse Speed Adjustment



speed adjustment are different based on control panel.



1. In the auto or MDI mode, rotate the rapid traverse speed adjustment button to adjust the feed specified in the program. The range of adjustment is from **0%** to **100%**.
2. In the auto or MDI mode, press the corresponding override ratio keys to adjust the rapid traverse speed.

The operations on rapid traverse

### 6.7.3 Spindle Override Adjustment



The spindle clockwise/counter clockwise speed can be adjusted by spindle override.

Rotate the spindle speed adjustment button to adjust the spindle speed, which ranges between **50%** and **120%**. The spindle speed cannot be adjusted during mechanical gear change.

### 6.7.4 Machine Lock



This function is used to disable the machine coordinate axis motion.

In the manual mode, click the key (the indicator is on). The system simulative runs the program in the Auto mode without machine motion. The axis position information changes on the screen, but the movement

commands from servo axis are not output. Hence, the machine does not run. This function is used to verify programs.

**Attention**

1. The tool does not move to the reference point even the **G28**, **G29** functions are enabled.
2. The function is invalid during the auto operation.
3. During the simulative operation, the machine lock function can be canceled only when the simulation is completed.
4. You need to return the tool to the reference point every time after the function is executed.

## 7 Position Information

---

---

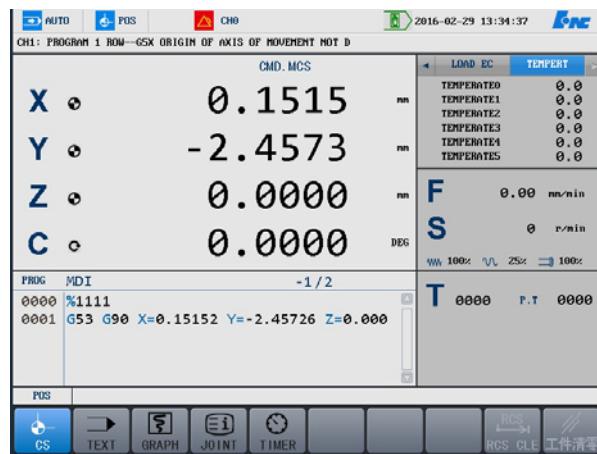
This chapter includes the following sections:

- **Coordinate Display**
- **Text Display**
- **Graph Display**
- **Joint Display**

## 7.1 Coordinate Display

Pos

During program running, press **POS → CS** to view the position information of the current program shown in different indication. See the figure below:



You may press **TIMER** to change the tool information interface at the right lower part (valid for milling machines).

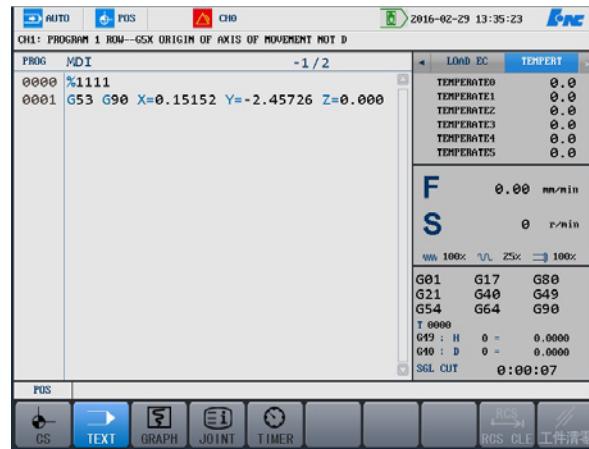
### Attention

Press **Set → PARA → DISPLAY PARAMETER** to select the displayed indication. See section 4.5.2.

## 7.2 Text Display

### Operation procedure

→ TEXT to view the G codes, coordinate system information, M commands, and feed rate F. See the figure below:



Press **TIMER** to change the machining information interface at the right lower part (valid for milling machines).

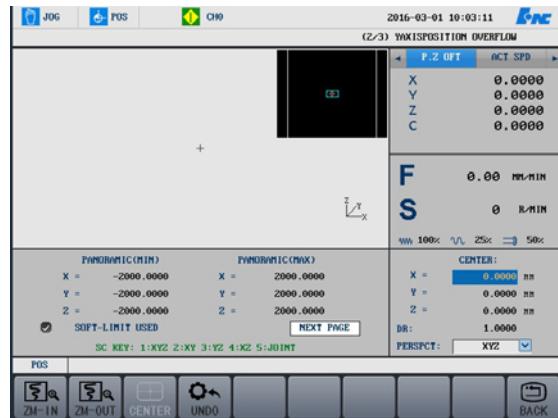
During program running, press **POS**

### Attention

Set the G code display mode to 3, and the line number displayed in the program is 9999 (for detailed information, see HNC-8 Parameters Description).

## 7.3 Graph Display

During program running, press **POS → GRAPH** to simulate the machining process.



### Coordinate System

1. Set the coordinate system (see section 4.3.2), e.g. G54 X10 Y20 Z30
2. Press **POS → GRAPH → CS** to select a coordinate system, and press G54.



3. The XYZ value of the graph center is displayed at the right lower part.

### Graph settings



1. Press **POS → GRAPH → GRAPH SET**. You may manually set the graph center, display scale, and display modes.
2. Move the cursor to select an item to set.
3. Press **Enter** to enter the editing mode.
4. Press **Enter** again to confirm the operation.

**Define graph center**

1. Press **POS → GRAPH → CENTER**.
2. Move the cursor to manually define the graph center.

**Attention:** The center can be defined in the XY, XZ, YZ modes.

**Restore settings**

1. Press **POS → GRAPH → UNDO**.
2. The system prompts you whether to restore the settings (Y/N).
3. Press **Y** to restore the settings or press **N** to cancel the restoring.

**Graph display mode**

You may use the shortcut keys to change the graph display modes.

- **[Switch view]:** Press 1, 2, 3, 4, 5 to switch among different display modes:
  - 1: XYZ
  - 2: XY
  - 3: YZ
  - 4: XZ
  - 5: the four modes above simultaneously
- **[Zoom in/out graph]:** Press PgUp or PgDn.
  - PgUp: Zoom in the graph
  - PgDn: Zoom out the graph
- **[Rotate graph]:** Press +, -, ▶, ◀, ▲, or ▼
  - +/-: Rotate the graph around the Y axis
  - ▶/◀: Rotate the graph around the Z axis
  - ▲/▼: Rotate the graph around the X axis

**Attention**

The graph cannot be set during program running.

## 7.4 Joint Display

### Operation procedure

During program running, press **POS → JOINT** to display eight kinds of position information.

CMCS		ACTMCS		D.T.G		TRK ERR	
X	0.0000	X	266.6513	X	0.0000	X	0.0000
Y	0.0000	Y	266.7013	Y	0.0000	Y	0.0000
Z	0.0000	Z	266.7060	Z	0.0000	Z	0.0000
C	0.0000	C	0.0000	C	0.0000	C	0.0000

LD CUR		CMD_P		HT POS		WCS_0	
X	0.000	X	0	X	266651337	X	0.0000
Y	0.000	Y	0	Y	266701258	Y	0.0000
Z	0.000	Z	0	Z	266706013	Z	0.0000
C	0.000	C	0	C	0	C	0.0000

## 8 Diagnosis

---

---

This chapter includes the following sections:

- **Alarm Display**
- **Alarm History**
- **Ladder Monitoring**
- **Oscilloscope**
- **Input and Output**
- **Status Display**
- **Macro Variables**
- **Servo Adjustment**
- **Machining Information**
- **Version**

## 8.1 Import System Logo Interface

### Function description

You may change the logo on the booting interface and application interface through the USB loading.

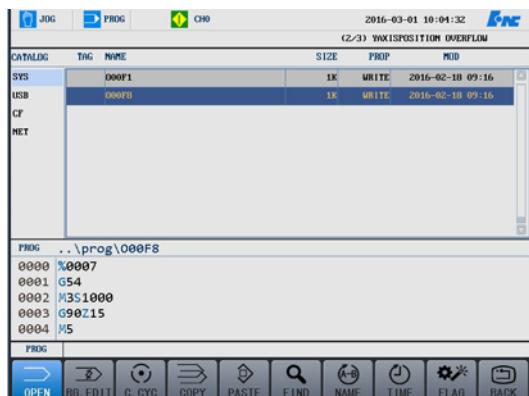
There are two logo pictures.

1. Booting logo



Picture standard:

- 1) BMP format
  - 2) 800\*600 pixel
  - 3) 24-bit true color
  - 4) File name: init\_hnc.bmp (Case sensitive)
2. Small system logo



Picture standard:

- 1) BMP format
- 2) 52\*24 pixel
- 3) 24-bit true color
- 4) File name: logo.bmp (Case sensitive)

#### Operation method

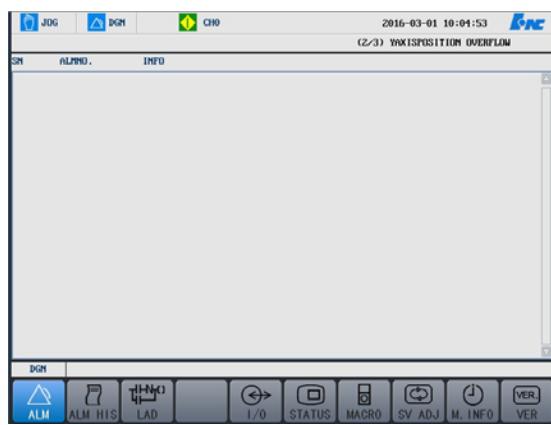
1. Create the LOGO picture based on the specified standard.
2. Enter the permission of Machine manufacturer or above, and the **LOAD LOGO** menu will be displayed under **Dgn → VER**.
3. Click **LOAD LOGO** to load the logo restart the system.

## 8.2 Alarm Display

Dgn

If an error occurs during system running or processing, the message "ALARM" will be displayed on the screen. You may press this key to check the detailed warning messages.

1. Press Dgn → ALM.

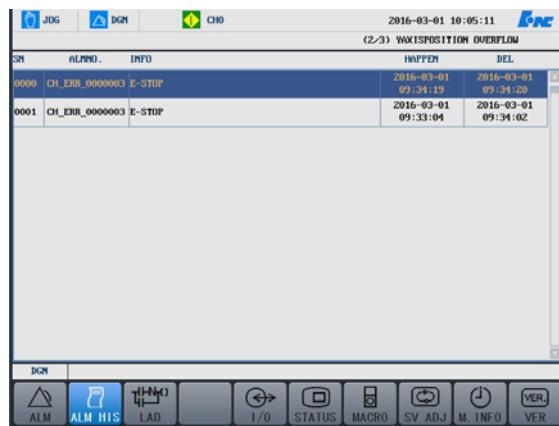


2. Use ▲, ▼, PgUp and PgDn to view alarm information.

## 8.3 Alarm History

### 8.3.1 Operation Procedure

1. Press **Dgn** → **ALM HIS** to display the alarm history.



2. Use **▲**, **▼**, **PgUp** and **PgDn** to view the alarm history.

### 8.3.2 Machining Status Recorded during Alarms

#### Overview

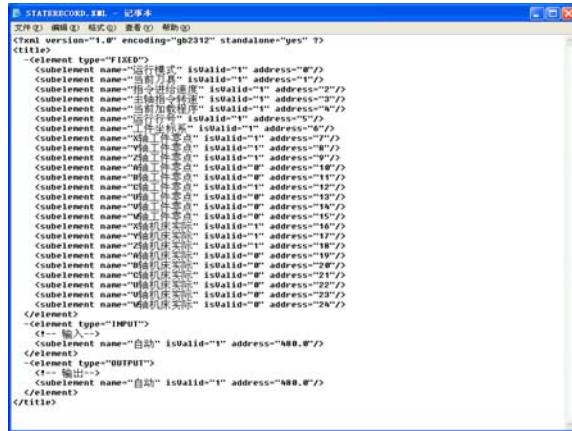
information of the last 10 machine alarms after the machine is booted.

Record the machining status

#### Function description

The recorded machining status information can be divided into fixed information and configuration information. All recorded information needs to be configured in the **STATERECORD.XML** file under the data directory.

The format of the configuration file is as below:



```

<?xml version="1.0" encoding="gb2312" standalone="yes" ?>
<title>
  <element type="FIXED">
    <subelement name="运行模式" isValid="1" address="8"/>
    <subelement name="当前刀具" isValid="1" address="1"/>
    <subelement name="主轴转速" isValid="1" address="2"/>
    <subelement name="进给速度" isValid="1" address="3"/>
    <subelement name="当前加减速率" isValid="1" address="4"/>
    <subelement name="冷却液流量" isValid="1" address="5"/>
    <subelement name="X轴工作空位" isValid="1" address="6"/>
    <subelement name="Y轴工作空位" isValid="1" address="7"/>
    <subelement name="Z轴工作空位" isValid="1" address="8"/>
    <subelement name="M轴工作空位" isValid="0" address="18"/>
    <subelement name="B轴工作空位" isValid="0" address="19"/>
    <subelement name="U轴工作空位" isValid="0" address="10"/>
    <subelement name="V轴工作空位" isValid="0" address="11"/>
    <subelement name="W轴机床空位" isValid="0" address="16"/>
    <subelement name="Y轴机床空位" isValid="1" address="12"/>
    <subelement name="Z轴机床空位" isValid="0" address="13"/>
    <subelement name="M轴机床空位" isValid="0" address="19"/>
    <subelement name="B轴机床空位" isValid="0" address="28"/>
    <subelement name="C轴机床空位" isValid="0" address="21"/>
    <subelement name="U轴机床空位" isValid="0" address="22"/>
    <subelement name="V轴机床空位" isValid="0" address="28"/>
  </element>
  <element type="INPUT">
    <!-- 输入 -->
    <subelement name="自动" isValid="1" address="480.0"/>
  </element>
  <element type="OUTPUT">
    <!-- 输出 -->
    <subelement name="自动" isValid="1" address="480.0"/>
  </element>
</title>

```

#### FIXED---Fixed status

It is not recommended that you modify the name and address properties in this status. You may modify the value of **isValid**: the value **0** indicates the status is valid while **1** indicates the status is valid.

#### INPUT--Input port status

Example: `<subelement name="automatic input" isValid="1" address="480.0"/>` input port X480.0: **name** indicates the name of the input port, **address** indicates the address of the input port, and **isValid** indicates whether the status is valid.

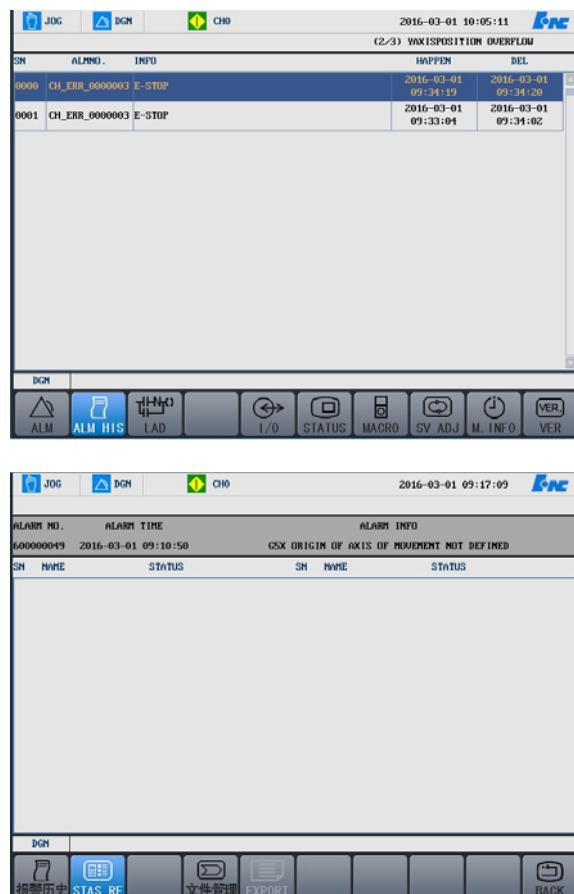
#### OUTPUT--Output port status

Example: `<subelement name="automatic output" isValid="1" address="480.0"/>` output Y480.0: **name** indicates the name of the output port, **address** indicates the address of the output port, and **isValid** indicates whether the status is valid.

A maximum of 24 records can be configured for one alarm.

The configured **STATE RECORD.XML** file in the **data** directory can be created into the BTF upgrade package, which can be imported to the CNC system by upgrading.

On the **ALM HIS** interface, select an alarm, and press **Enter** to view the status information corresponding to the alarm. See the figure below:



## 8.4 Ladder Monitoring

---

### 8.4.1 Ladder Diagnosis

#### Operation procedure

1. Press **Dgn** → **LAD** → **LAD Dgn** to view the value of each variable.
2. The system displays the value in decimal by default. You can press the hexadecimal function key, and the system displays the value in hexadecimal.
3. Use the cursor keys to select an element.
4. Press the **NOT ALLOW** or **ALLOW** key to shield or activate the element.
5. Press **UNDO** to cancel the shielding or activating operation.
6. You may enter an element name in the search box, press **Enter** to search the element. You may press **PgUp** or **PgDn** to search for the element with the same name up or down.

### 8.4.2 Ladder Modification

#### Attention

The function is available only to the machine manufacturer, CNC manufacturer and administrator.

#### Search

1. Press **Dgn** → **LAD** → **FIND**, enter the element name, and press **Enter** to search for the element.
2. Use **PgUp** or **PgDn** to search for the element with the same name up or down.

#### Modify

1. Press **Dgn** → **LAD** → **EDIT**.
2. Use the cursor keys to select an element, and press **Enter** to enter the editing status.
3. Enter the element value in the editing box.
4. Press **Enter** again to complete the editing.

5. Press the functional keys corresponding to the **EDIT** menu to create element.
  - Straight line: insert straight line
  - Vertical line: insert vertical line
  - Delete element: delete elements
  - Delete vertical line: delete vertical lines
  - Normally open: normally open contacts
  - Normally closed: normally closed contacts
  - Logical output
  - Inverse output
  - Function module (You may directly select elements based on the initials)

**Note:** For detailed information about the element, see HNC-8 PLC Programming User Manual.

**Commands**

1. Press **Dgn → LAD → COMMAND**.
2. Edit the ladder graph by using the following keys:
  - Select: select the row where the cursor is
  - Delete: deleted the row where the cursor is
  - Move: move the selected element
  - Copy: copy the selected element
  - Paste: paste the selected element
  - Insert row: insert a row prior to the row where the cursor is
  - Add row: insert a row behind the row where the cursor is

**Load**

Press **Dgn → LAD → LOAD** to load the current ladder graph information.

**Cancel**

Press **Dgn → LAD → UNDO** to cancel the editing on the ladder graph.

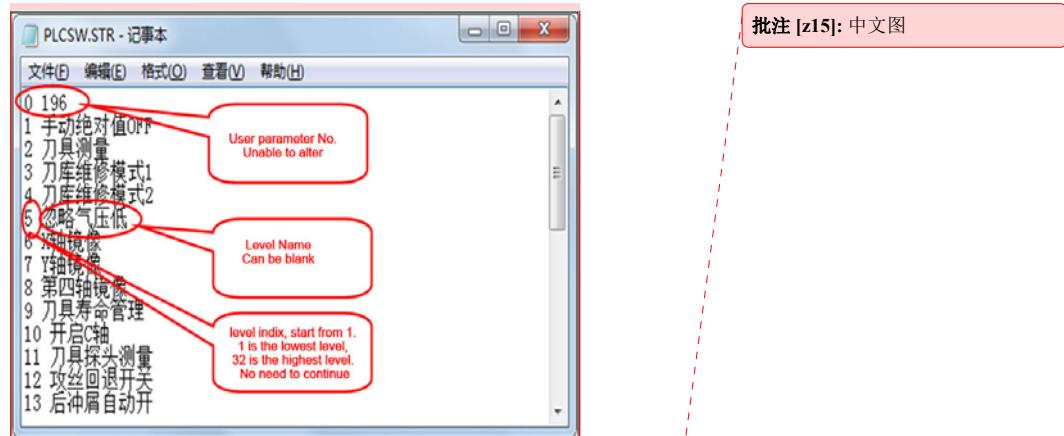
**Save**

Press **Dgn** → **LAD** → **SAVE** to save the editing on the ladder graph.

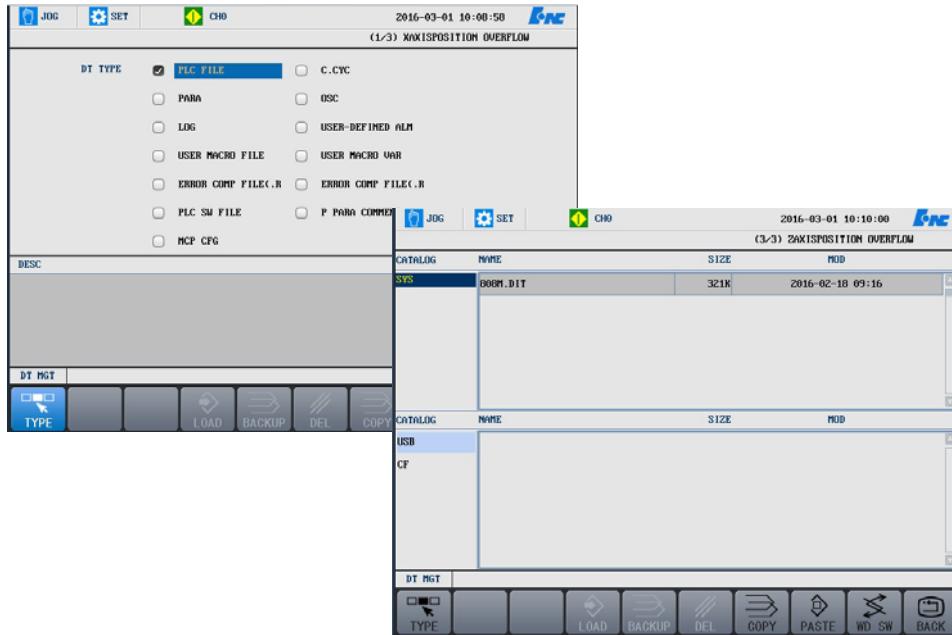
### 8.4.3 PLC Switch Settings

The PLC switch settings function is used to decode the specified user P parameter into 32 bits; each bit can be used as a configurable PLC switch.

You may use the **PLCSW.STR** configuration file under the **parm** directory to conduct the PLC switch settings. The **PLCSW.STR** file is in the following format:



After the **PLCSW.STR** file is configured, you may press **Set** → **PARA** → **DT MGT** to select **PLC FILE** and import the **PLCSW.STR** file. See the figure below:



Press Set → PLC SW to enter the PLC Switch function interface, where you may conduct operation based on bit for specified P parameters. See the figure below:



### Interface operation description

**Up, Down, Left, Right** keys: to move the position of the cursor.

**Enter:** define the value of the position where the cursor is. If the current status is **ON**, set it to **OFF**; if the current status is **OFF**, then set it to **ON**.

#### Menu operation description

**SET EFF:** This is used to protect the **ON** and **OFF** menus. Only after you press the **SET EFF** menu, can the operations on ON/OFF be valid.

**ON:** Set the position where the cursor is to **ON**.

**OFF:** Set the position where the cursor is to **OFF**.

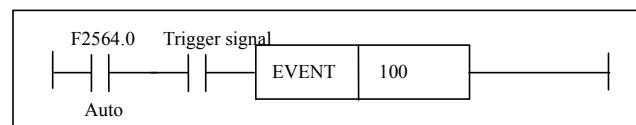
The settings are saved in the specified user P parameter. As shown in the figure below, the parameter **196** in this example is specified by the **PLCSW.STR** file.



#### 8.4.4 Manually Call Subprograms

##### Operation procedure

Create a ladder graph in the PLC as below:



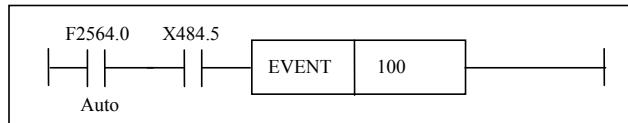
 : indicates the external input signal to call subprograms

100: indicates the program name **O100**

#### Attention

**In this function, the system is in the auto mode and requires a trigger signal.**

#### Example

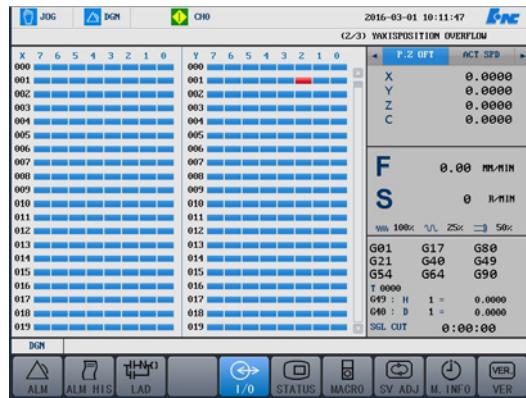


When the input signal X484.5 is valid, press the **Start** button to call the **O100** subprogram.

## 8.5 Input and Output

### Operation procedure

1. Press Dgn → I/O.



2. Use the PgUp and PgDn keys to view the X or Y register status.

## 8.6 Status Display

---

### Operation procedure

1. Press **Dgn** → **STATUS**.
2. Use **▲** and **▼** to select the register type to view.
  - X: input from the machine to PMC
  - Y: output from the PMC to machine
  - F: output from the CNC to PMC
  - G: input from the PMC to CNC
  - R: intermediate relay status display
  - B: breakpoint data display
3. Press **PgUp** and **PgDn** to browse screen up and down.
4. Press the "Binary", "Decimal" or "Hexadecimal" key to view the value of the register.
5. Press **FIND** to search for a register value.

### Attention

- You may press the functional keys or shortcut keys to view the F register and G register individually.
  - System (Alt+S)
  - Channel (Alt+C)
  - Axis (Alt+A)
- You may edit the B register.

## 8.7 Macro Variables

### Macro variables

HNC-818 CNC system provides you with the macro function similar to high-level language. You may use variables to conduct arithmetic operation, logical operation, and combined function operations. In addition, the macro program provides cycle statement, branch statement, and subprogram calling statements, which makes it easy to create complex machining programs and avoids complicated calculation that required in manual programming.

1. Press **Dgn** → **MACRO** to view system macro variables.
2. Press **FIND**, enter a macro variable number, and press **Enter** to search for the macro variable.

### Attention

1. For detailed information about each macro variable, see section 13 in this user manual.
2. The macro variables' value ranges from **-2147483648** to **2147483648**.

### User macro

1. Press **Set** → **PARA** → **DT MGT** → **TYPE** → **Customized Configuration File**.
2. Press **WD ST** → **USB**, select a customized configuration file, press **LOAD** to load the file.
3. Press **Dgn** → **MACRO** → **USER MA**. The system displays the macro variable numbers, names, and values.



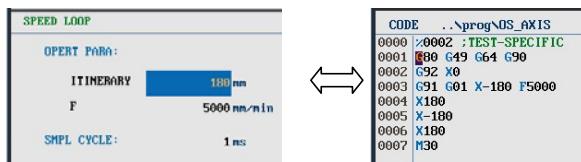
4. Move the cursor to select a macro variable value, press **Enter** to modify the value.
5. You may create macro programs as required.

## 8.8 Servo Adjustment

### 8.8.1 Speed Loop

#### Auto load of G codes

2. If you enter the **SV ADJ** interface in the auto mode, the system will automatically load the test G code. A message is displayed to prompt you that the test code is loaded and they may press the Start button to start sampling.
3. If you enter the **SV ADJ** interface in the manual mode, you need to change it to the auto mode, and press the **Reset** button to load the G code.
4. If you need to modify the G code, you may press **Set** to modify the G code, press **CODE** to browse the G code (The settings interface corresponds to the **CODE** interface.).



5. You may press **Alt+ direction keys (+/-)** to modify the test axis.

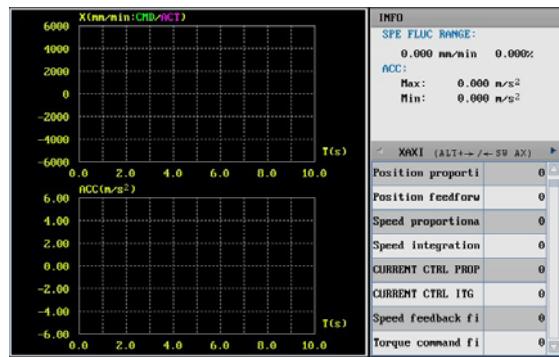
**Note:** You need to manually move the machine to a safe position before start sampling.

1. Press **Dgn** → **SV ADJ**.

#### Sampling

1. Press **Dgn** → **SV ADJ**, move the cursor to select **SPEED LOOP**.
2. In the manual mode, move the machine to a safe position, switch to the auto mode, press **Reset** to load the testing G code, and press **Start** to start measurement.
3. The measurement waveform of the speed, acceleration speed, and the speed fluctuation range is obtained after the measurement. See the figure below:

4. Press the cursor key "→" to adjust the parameter of each axis, e.g. "SPEED PROPORTIONAL GAIN" and "SPD FEEDBACK FILTER".
5. Press **Alt** + direction key (+/-) to select an axis for settings.
6. You may adjust parameters, conduct measurement for multiple times to meet the expected effects.



## 8.8.2 Position Loop

### Auto load of G codes

1. Press **Dgn** → **SV ADJ**.
2. If you enter the **SV ADJ** interface in the auto mode, the system will automatically load the test G code. A message is displayed to prompt you that the test code is loaded and they may press the Start button to start sampling.
3. If you enter the **SV ADJ** interface in the manual mode, you need to change it to the auto mode, and press the **Reset** button to load the G code.
4. If you need to modify the G code, you may press **Set** to modify the G code, press **CODE** to browse the G code (The settings interface corresponds to the **CODE** interface.).



CODE .SprogNGS\_AXIS  
 0000 G002 :TEST-SPECIFIC  
 0001 G0 G49 G64 G90  
 0002 G92 X0  
 0003 G91 G01 X-180 F5000  
 0004 X180  
 0005 X-180  
 0006 X180  
 0007 M30

y move the machine to a safe position  
 before start sampling.

5. You may press Alt + direction keys (+/-) to modify the test

### Sampling

cursor to select **POS LOOP**.

2. In the manual mode, move the machine to a safe position, switch to the auto mode, press **Reset** to load the testing G code, and press **Start** to start measurement.



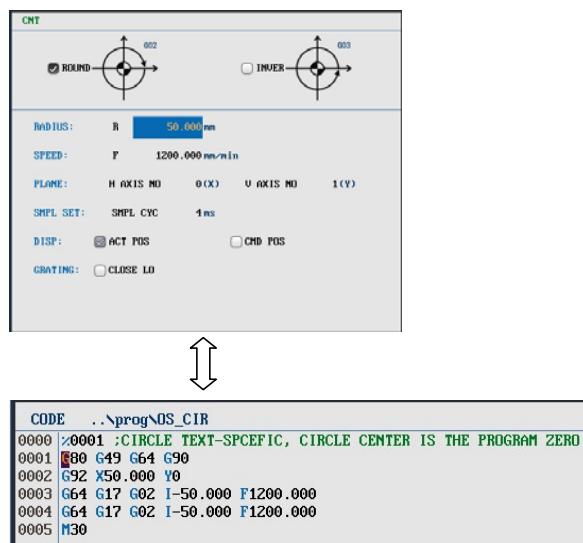
3. Press the cursor key "→" to adjust the parameter of each axis, e.g. "SPEED PROPORTIONAL GAIN" and "SPD FEEDBACK FILTER".
4. Press Alt + direction key (+/-) to select an axis for settings.
5. You may adjust parameters, conduct measurement for multiple times to meet the expected effects.

1. Press **Dgn** → **SV ADJ**, move the

### 8.8.3 Circularity Test

#### Auto load of G codes

1. Press **Dgn → SV ADJ.**
2. If you enter the **SV ADJ** interface in the auto mode, the system will automatically load the test G code. A message is displayed to prompt you that the test code is loaded and they may press the Start button to start sampling.
3. If you enter the **SV ADJ** interface in the manual mode, you need to change it to the auto mode, and press the **Reset** button to load the testing G code.
4. If you need to modify the G codes, you may modify the parameters on the interface to modify the G codes. You may view the modified G codes in the **CODE** interface.



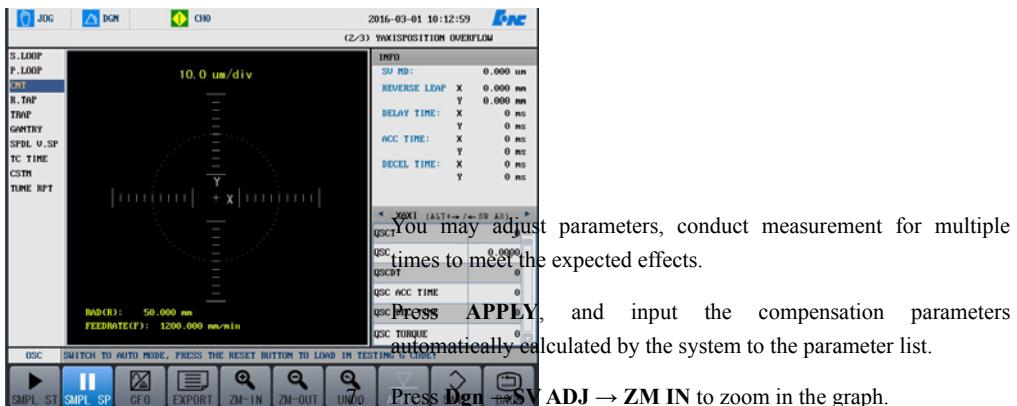
5. You may press **Alt+ direction keys (+/-)** to modify the test axis.

**Note:** You need to manually move the machine to a safe position before start sampling.

## Sampling

1. Press **Dgn → SV ADJ**, move the cursor to select **CNT**.

2. Press **SMPL ST** and **Start** to start the measurement. The result is shown in the figure below:



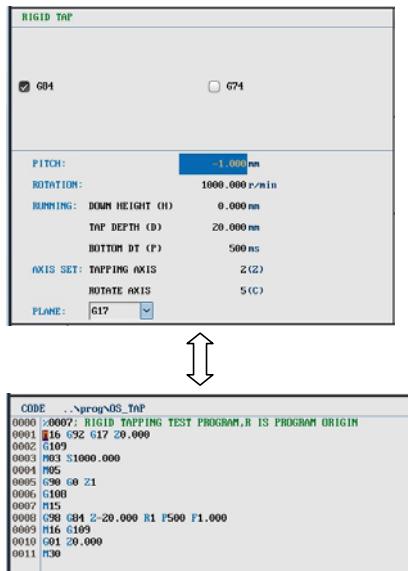
3. Press the cursor key "→" to adjust the parameter of each axis.
4. Press **Alt** + direction key (+/-) to select an axis for settings.
5. Press **Dgn** → **SV ADJ** → **ZM OUT** to zoom out the graph.
6. Press **Dgn** → **SV ADJ** → **UNDO** to restore the graph.

### 8.8.4 Rigid Tapping

#### Auto load of G codes

1. Press **Dgn** → **SV ADJ**.
2. If you enter the **SV ADJ** interface in the auto mode, the system will automatically load the test G code. A message is displayed to prompt you that the test code is loaded and you may press the Start button to start sampling.

3. If you enter the **SV ADJ** interface in the manual mode, you need to change it to the auto mode, and press the **Reset** button to load the testing G code.
4. Press **Set** to enter the G code settings interface. You may modify the parameters on the interface to modify the G codes. You may view the modified G codes in the **CODE** interface.

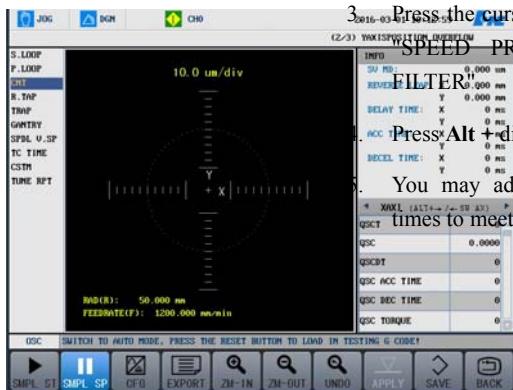


5. You may press **Alt+ direction keys (+/-)** to modify the test axis.

**Note:** You need to manually move the machine to a safe position before start sampling.

## Sampling

1. Press **Dgn → SV ADJ**, move the cursor to select **POS LOOP**.
2. In the manual mode, move the machine to a safe position, switch to the auto mode, press **Reset** to load the testing G code, and press **Start** to start measurement.



Press the cursor key "→" to adjust the parameter of each axis, e.g. "SPEED", "PROPORTIONAL GAIN" and "SPD FEEDBACK".  
 Press Alt + direction key (+/-) to select an axis for settings.  
 You may adjust parameters, conduct measurement for multiple times to meet the expected effects.

## 8.8.5 Trap

### Function Description

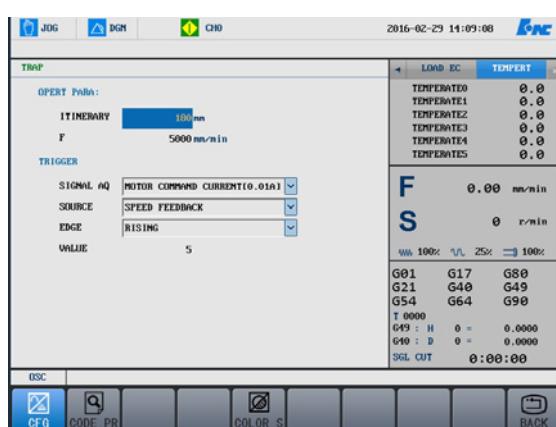
loop gain and reduce speed fluctuation.

The system provides trap function, which is mainly used in the following scenarios:

1. Increase the speed loop gain:  
 Select a speed to move the machine. After increasing the speed loop gain, if machine vibration occurs, then users may use the trap 1 to eliminate the vibration. If users continue to increase the gain, and the vibration occurs, then users may use the trap 2 to eliminate the vibration. In this mode, it is recommended to use the processing speed for debugging, in order to maximize the speed
2. High and low speed optimization: Firstly, conduct debugging at low speed (for example F2000), increase the gain and add trap 1. Thereafter do not increase the gain, and change to the high-speed (e.g. F40000) mode to conduct debugging, to check whether machine vibration or whistle occurs. If there is vibration or whistle, use trap 2 to eliminate it.



Settings interface



Usage	Description
Increase the speed loop gain	Select a speed to move the machine. After increasing the speed loop gain, if machine vibration occurs, then you may use the trap 1 to eliminate the vibration. If you continue to increase the gain, and the vibration occurs, then you may use the trap 2 to eliminate the vibration. In this mode, the speed loop can be increased to about 2000.

	whether machine vibration or whistle occurs. If there is vibration or whistle, use trap 2 to eliminate it.
--	--

If two traps are configured and there is still a little vibration or whistle, you may:

1. Increase the speed integral time constant properly, but the value should not exceed 3.

High and low speed optimization	Firstly, conduct debugging at low speed (for speed loop gain a little). F2000), increase the gain and add trap 1. Thereafter do not increase the gain, and change to the high-speed (e.g. F40000) mode to conduct debugging, to check
---------------------------------	---

### 8.8.6 Gantry Synchronization Axis

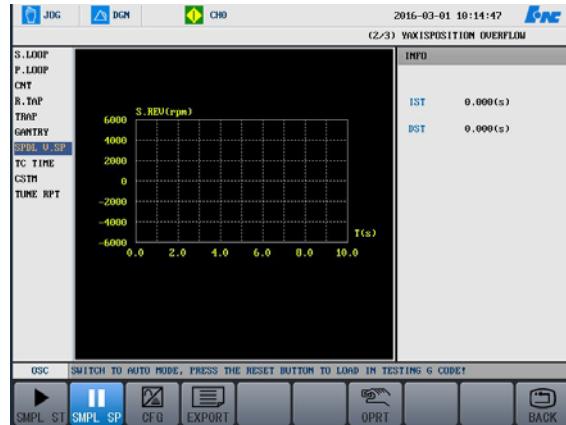
This function is mainly used for collecting the load currents of two synchronization axes, as well as the position deviation and the current deviation of the synchronization axes. See the figure below:



The system automatically generates G codes based on the data (synchronization axis number, travel, speed) defined in the settings interface. In the auto mode, run the G code to collect data. In the manual mode, press the **SMPL ST** button to manually move the axis to collect data. After the data collection is completed, the maximum positional deviation and current deviation values of the synchronization axis will be displayed in the right pane.

### 8.8.7 Spindle Acceleration and Deceleration Speed

This function is used to test the time that the spindle speed increases from **0** to the specified speed, and the time that the spindle speed decreases from the specified speed to **0**. See the figure below:



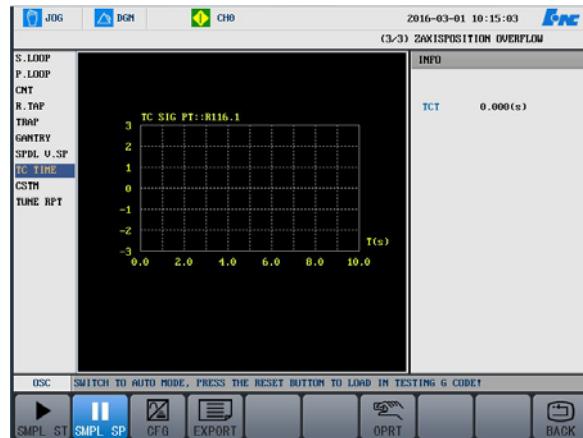
In the auto mode, enter the spindle acceleration and deceleration speed configuration interface. Set the spindle speed, and the system will automatically generate simple G-codes, e.g. "M3 S1000 M5 M30". After running the G code, the system automatically calculates the spindle acceleration and deceleration speed with specified rotation, and output the results in the right information pane.

#### Attention

1. Make sure the Spindle override adjustment button is in the 100% position; otherwise, the calculated time may be inaccurate.
2. Please set the value of the spindle parameter "**SPEED ARRIVAL RANGE**" to **0**; otherwise, the calculated result may be inaccurate.

### 8.8.8 Tool Change Time

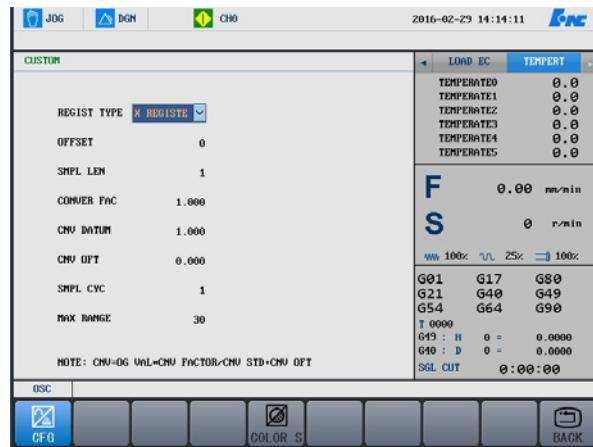
This function is used to collect tool change signal and calculate the time required for tool change. See the figure below:



### 8.8.9 Customized Collection

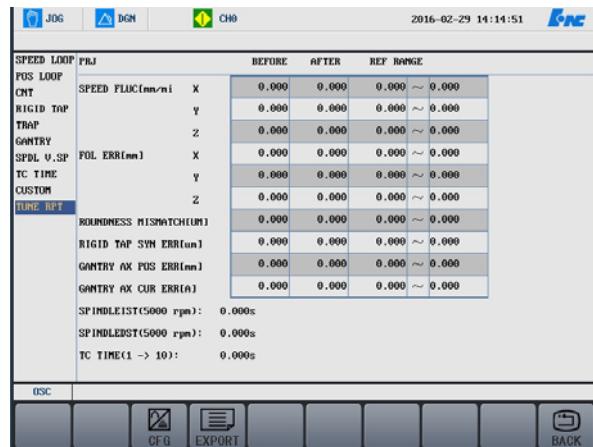
This function can be used to capture the value of any register. Users can customize the conversion coefficient, conversion standard and offset of the register value. According to the formula "conversion value = original value \* conversion coefficient/conversion standard + offset", users may convert the register value into real values, such as voltage, current, position, temperature, and so on. The value collection and configuration interface is as below:





## 8.8.10 Tuning Reports

The function is used to record the KPIs before and after tuning, establish tuning files, and export the file for archiving. See the figure below:



### Attention

When using the servo adjustment function for tuning, make sure the feed override and Spindle override adjustment button is in the 100% position; otherwise, the recorded data may be inaccurate.

### 8.8.11 Operation and Settings

You may zoom in, zoom out, change colors for the sampling diagram mentioned above.

#### Graph related operations

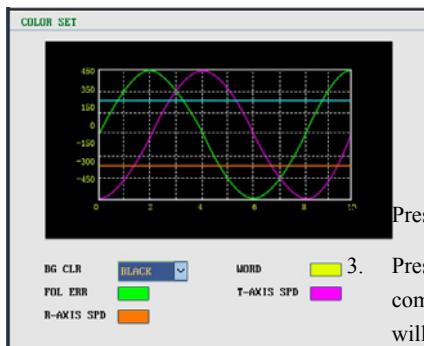
You may press **Dgn** → **SV ADJ** to view the graph.

- **PgUp** and **PgDn**: select a mode to browse the graph: JOINT, GRAPH 1, GRAPH 2
- **Alt+↑** or **Alt+↓**: In the combined mode, switch between GRAPH 1 and GRAPH 2
- **Zoom in horizontal axis**: Zoom in the horizontal axis
- **Zoom out horizontal axis**: Zoom out the horizontal axis
- **Zoom in vertical axis**: Zoom in the vertical axis
- **Zoom out vertical axis**: Zoom out the vertical axis
- **Zoom in windows**: Zoom in the selected area in the graph
- **Zoom out windows**: Zoom out the selected area in the graph
- **UNDO**: restore the original legend of the graph
- **Enter**: Zoom in the graph in the selected area

#### Color settings

You may change the color of the graph displayed in the system.

1. Press **Dgn** → **SV ADJ** → **Set** → **COLOR SET**. The figure below shows the color settings interface for the speed loop graph:



Press **Enter** to select a background color.

3. Press **Enter** to confirm the settings. The system text color, command speed color, actual speed color, acceleration speed color will be changed after the background color is changed.

#### Export

Press **Dgn** → **SV ADJ** → **EXPORT** to export the sampling data.

#### Save

Press **Dgn** → **SV ADJ** → **SAVE** to save the parameter settings.

## 8.9 Machining Information

### View

Press **Dgn** → **M. INFO** → **STATISTICS** to view the machining information.

### Pre-define

**This function is available only to the machine manufacturer, CNC manufacturer, and administrator.**

1. Press **Dgn** → **M. INFO** → **SET** to define the machining information.
2. Move the cursor set to select the item to set.
3. Press **Enter**.

### Clear

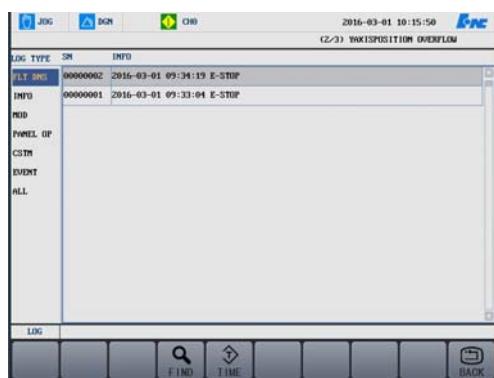
**This function is available only to the machine manufacturer, CNC manufacturer, and administrator.**

Press **Dgn** → **M. INFO** → **CLEAR** to clear all current machining statistics information.

**Note:** After modifying the time, users need to manually clear the machining statistics time related information; otherwise, wrong statistics data may be displayed.

### Log

1. Press **Dgn** → **M. INFO** → **LOG** to display system debugging information.

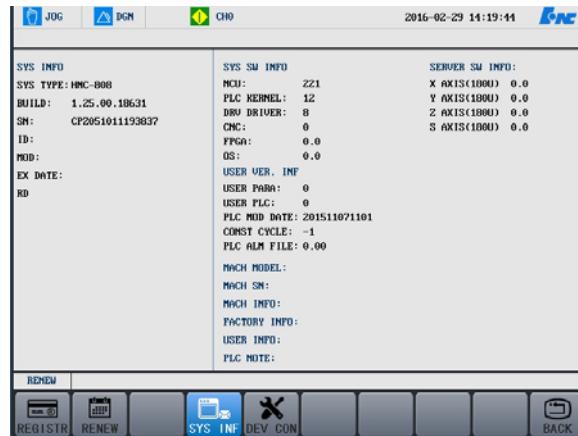


2. Move the cursor to select a log type.
3. Press **EXPORT** to export corresponding log files.
4. You may press the chronological order key to display the log files in the chronological order.

## 8.10 Version

### Query version

You may Press **Dgn →VER** to view the system version information.



### Register

Note: The registration expiration alarm is reported 12 hours after the real expiration.

### Renew

- As shown in the figure above, the message "REGISTR" is highlighted on the interface. Users may press **Enter**, select **USB**, load the registration file, cut off the power, and restart the system.
- As shown in the figure above, the message "RENEW" is highlighted on the interface. Users may press **Enter**, and enter the renew code.

Note: Pay attention to the capitalization and hyphens.



- Press **Enter** again.

### Set machine information

**Note:** This function is available only to the machine manufacturer, CNC manufacturer, and administrator.

- Press **►** to move the cursor to the **MACH MODEL** part.

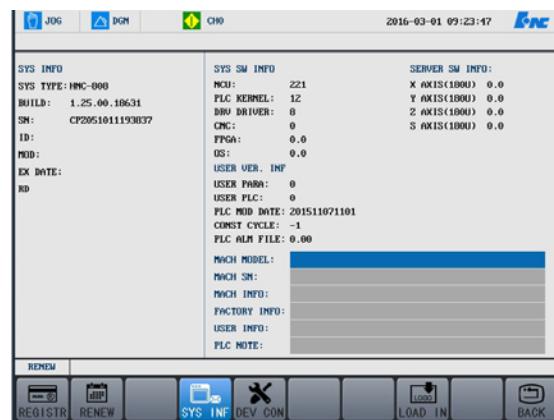
MACH MODEL:	
MACH SN:	
MACH INFO:	
FACTORY INFO:	
USER INFO:	
PLC NOTE:	

2. Press **Enter** to input information about machine model, manufacturer, etc.

#### PLC version information

This function is used to record the PLC modification time and information.

After debugging personnel modify the PLC, the PLC modification time will be automatically recorded, shown in the figure below. In addition, the PLC modification information can be input in the **PLC NOTE**, e.g. Modified by. A maximum of eight characters can be input.



## **9    User Operation and Maintenance Information**

---

---

This chapter includes the following sections:

- Environmental Requirements
- Grounding
- Power Conditions
- Clean Fan Filter
- Operation After Being Left Unused

## 9.1 Environmental Requirements

---

The table below describes the environmental requirements of HNC-818:

Items	Description
Working temperature (°C)	<b>0</b> to <b>+45</b> , non-freezing
Temperature change	< <b>1.1</b> °C/min
Relative humidity	<b>90%</b> RH or lower (non-condensable) Normal: <b>75%</b> or lower Short period (within a month): Max. <b>95%</b>
Storage temperature (°C)	- <b>20</b> to <b>+60</b> , non-freezing
Storage humidity	Non-condensable
Surrounding environment	Indoor (non-exposed to sun), anti-sepsis, anti-burning, anti-fog, anti-dust
Height	A maximum of <b>1000 (2000)</b> meters above sea level
Vibration (m/s)	<b>10</b> to <b>60</b> Hz: <b>5.9 (0.6G)</b> or lower

## 9.2 Grounding

---

Correct grounding is critical for the numerical control unit and other electrical devices. Correct grounding may:

- Protect operators from electric shock or injury caused by no grounding or incorrect grounding.
- Protect the electrical devices from inductive interference which may lead to errors or unexpected results.

When installing machine, reliable grounding must be provided. The neutral line in the power grid cannot be used as the grounding line, otherwise, it may cause device damage or abnormal operation, or even casualties.

### 9.3 Power Conditions

---

The power supply of HNC-818 turning system is provided by the electric cabinet. For more information about machine power supply, see the installation guide of the machine.

## 9.4 Clean Fan Filter

---

Filters are used on cooling fans to prevent dust from entering into devices, which are designed at the inlet and outlet.

However, it may prevent adequate cooling if the filters become clogged, and thus cause improper device running. It is recommended to clean the filters every three months. In dusty environments, clean the filters more often.

## 9.5 Operations After Being Left Unused

---

After a long period of being left unused, numerical control devices should be cleaned and dried, so should the wiring and ground connections. Once power is resumed after being left unused, observe the operation for several hours to make sure there is no problem.