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

6.1 Start, Pause and Stop

6.2 Dry Running

6.3 Over-Block Operation

6.4 Optional Stop

6.5 Single Block Operation

6.6 Breakpoint Related Operation

6.7 Intervention at runtime

7 Position Information

7.1 Coordinate Display

7.2 Text Display

7.3 Graph Display

7.4 Joint Display

8 Diagnosis

8.1 Import System Logo Interface

8.2 Alarm Display

8.3 Alarm History

8.4 Ladder Monitoring

8.5 Input and Output

8.6 Status Display

8.7 Macro Variables

8.8 Servo Adjustment

8.9 Machining Information

8.10 Version

9 User Operation and Maintenance Information

9.1 Environmental Requirements

9.2 Grounding

9.3 Power Conditions

9.4 Clean Fan Filter

9.5 Operations After Being Left Unused
I overview
The following models are described in the manual:

<table>
<thead>
<tr>
<th>Name</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>HNC-818A Milling NC Unit</td>
<td>HNC-818A-MU</td>
</tr>
<tr>
<td>HNC-818B Milling NC Unit</td>
<td>HNC-818B-MU</td>
</tr>
</tbody>
</table>
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. Operation Device

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.
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. Power On/Off and 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.
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

In the manual feed mode, press this key and one of the three axis keys simultaneously to rapidly move the tool in the positive or negative direction of relevant axis.

### 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.

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:

<table>
<thead>
<tr>
<th>Override</th>
<th>×1</th>
<th>×10</th>
<th>×100</th>
<th>×1000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Incremental value (mm)</td>
<td>0.001</td>
<td>0.01</td>
<td>0.1</td>
<td>1</td>
</tr>
</tbody>
</table>

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. Manual Operation

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
3. Manual Operation

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

This function is used to disable tool 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**
3. Manual Operation

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. 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:

![Tool Magazine Data Table]

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.
4. Settings

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:

- As shown in the figure below, click Open.

- 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.
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:

![RCS CLE interface]

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:

![Parameter List Interface]

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:
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:
4. Settings

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**
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:

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 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.
4.4.5 Parameter Configuration Interface

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

![Parameter Configuration Interface](image)

**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:

![XML Configuration Example](image)

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:

![Diagram](image)

### 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

Debugging Files can be imported or exported in batches.

1. Press **Set → PARA → ALL.**

![Batch Debugging Interface](image)

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 exported in batches.

**Attention**

The function is available 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_YYMDDTIME.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.

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.
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 Renishaw software (generally C:\Program Files\Renishaw LaserXL). Copy the LEC.REN file to the USB flask disk.
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 \((710000 + \text{axis number} \times 1000)\) 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 \texttt{000359} parameter can be used to set the default permission after the system boots. If the parameter is set to \texttt{0}, the default permission is \texttt{WS ADMINS}; if the parameter is set to \texttt{1}, the default permission is \texttt{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.

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.

- 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.

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.
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.

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 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**.

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 conflict with other IP addresses of the LAN.

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**

1. This function is limited to CNC 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.
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:

![Parameter update interface]

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", "01234".
5. Program Editing and Management

### 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.

**Operation procedure**

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:

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**.
### G code encryption type

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Not display</td>
<td>Blank</td>
<td>Not support</td>
<td>Allowed</td>
</tr>
<tr>
<td>Writable</td>
<td>Able to be Browsed</td>
<td>Editable</td>
<td>Allowed</td>
</tr>
<tr>
<td>Read-only</td>
<td>Able to be Browsed</td>
<td>Not support</td>
<td>Allowed</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Not display</td>
<td>Blank</td>
<td>Not support</td>
<td>Not allowed</td>
</tr>
<tr>
<td>Writable</td>
<td>Unreadable</td>
<td>Not support</td>
<td>Not allowed</td>
</tr>
<tr>
<td>Read-only</td>
<td>Unreadable</td>
<td>Not support</td>
<td>Not allowed</td>
</tr>
</tbody>
</table>
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 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.
- Press N. The system will ask whether to replace
  b. Press N to cancel the current replacement.

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**

You may create sub directories in 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.

<table>
<thead>
<tr>
<th>Parameter No.</th>
<th>Description</th>
<th>Value</th>
<th>Effective Mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>040113</td>
<td>Random line scanning</td>
<td>1</td>
<td>Save</td>
</tr>
</tbody>
</table>

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:

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Y</td>
<td>Z</td>
<td>A</td>
<td>B</td>
<td>C</td>
<td>U</td>
<td>V</td>
<td>W</td>
</tr>
</tbody>
</table>

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**

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

![Feed Hold Mode](image)

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

**Attention**

- Press the **Feed Hold** key on the control panel 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

1. Press Prog → BLOCK → Non

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.

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

1. 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:

2. If there are multiple craft files, press the PREVIOUS or NEXT menu to switch files.

3. During program execution, you may view craft files.

Delete craft files

1. The associated craft files will be deleted when the main program is deleted.

2. Workshop manager or higher permission may delete the craft files displayed under CRAFT.

3. 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 feel, 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 \( \frac{\text{measured value of point A} + \text{measured value of point B}}{2} \) 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. 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.

Click the key on the machine control 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.
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 simulatively 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 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

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:

![Image of Text Display](image)

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

**Attention**

During program running, press **POS**

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.
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_hcnc.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

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.

![Alarm History Screen]

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

8.3.2 Machining Status Recorded during Alarms

**Overview**

Record the machining status information of the last 10 machine alarms after the machine is booted.

**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 STATRECORD.XML file under the data directory.

The format of the configuration file is as below:
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:
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

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.).

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 **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.).
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.

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. You may adjust parameters, conduct measurement for multiple times to meet the expected effects.

6. Press **APPLY** and input the compensation parameters automatically calculated by the system to the parameter list.

7. Press **Dgn → SV ADJ → ZM IN** to zoom in the graph.

8. Press **Dgn → SV ADJ → ZM OUT** to zoom out the graph.

9. 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.
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.

8.8.5 Trap

Function Description

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 loop gain and reduce speed fluctuation.

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.
**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.
Firstly, conduct debugging at low speed (e.g. 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.

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.
2. Decrease the speed loop gain a little.

### 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:
2. 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.

1. 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.

Renew

2. 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.

3. Press Enter again.

Set machine information

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

1. Press ▶ to move the cursor to the MACH MODEL part.
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:

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