HNC-8 System Operating Manual (Lathe)

V2.4 Series

Introduction

The manual may help you to quickly get familiar with the HNC-8 system, providing detailed information about commissioning, programming or application methods. Any updates or modification of the manual is not allowed without the written permission of Wuhan Huazhong Numerical Control Co., LTD (hereafter referred to as "HCNC"). Without HCNC's authorization or written permission, any units or individuals are not allowed to modify or correct the manual. HCNC will not be responsible for any losses thus incurred to customers.

In this manual we have tried as much as possible to describe all the various matters concerning of the system. However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities. Therefore, matters which are not especially described as possible in this manual should be regarded as “impossible” or “not allowed”.

Please favor me your instruction for shortages and inadequacies of the manual.

Note

- As to notes such as "Limitations" and "Usable functions", the specification provided by the machine tool manufacturer is superior to the manual. Please conduct dryrun before actual machining and confirm machining program, tool compensation volume and workpiece offset, and so on.
- Please explain matters which are not described in the manual as "Infeasible".
- The manual is prepared on the condition that all functions are configured. Please make a confirmation according to the specification provided by the machine tool manufacturer in use.
- For relevant instructions for machine tools, please refer to the specification provided by the machine tool manufacturer.
- Usable screens and functions differ with different NC systems (or versions). Please be sure to confirm specifications before use.

Contents

1 Overview

HNC-8 CNC lathe system includes HNC-808Di-TU, HNC-8Ai-TU and HNC-8Bi-TU. This manual is based on HNC-808Di-TU. In case of discrepancies, please refer to the specification provided by the machine tool manufacturer.

1.1 Basic Operation

HNC-808Di-TU system is the control system for CNC lathe There are 6 working mode keys on MCP panel "jog, auto, single block, MDI, incremental/MPG and reference position return". Function description of these 6 working modes are shown below.

Working mode	Function description	Function application
Jog	Control continuous movement of machine tool axis and auxiliary action.	Preparation for parts machining and simple parts machining.
Auto	The machine tool runs continuously and automatically based on the edited program.	Continuous and automatic machining and program verification of parts, and so on.
Single block	The machine tool runs automatically block by block based on the edited program.	Machining position check and program verification.
MDI*	The machine tool operates the manually input program.	Automatic machining and coordinate setup of simple parts, and so on.
Increment/handwheel	Accurately control axis movement of machine tool by key or handwheel.	Tool setting or manual machining of simple parts.
Reference position return	Control each axis of machine return to the reference point	Verify position of machine tool after power-up.

*For the non-Di series, MDI mode is set as the MDI function set

1.2 Basic Function

To complete different work under different working modes, corresponding application functions should be used. HNC-808Di-TU has 6 function keys on NC panel "Machining, set, program, diagnosis, maintenance and user-defined (MDI)". Each function key corresponds to a group of function sets from which corresponding function and interface can be selected (for soft key function menu and display interface, refer to chapter 3 "Display interface").

Function description and main content of function sets are shown below:

Function set	Function description	Function content
Machining	Functions required for automatic machining operations	Edit new program*, edit current loading program, edit selected program, machining program selection, program verification, tool setting, tool compensation*, graphics setup, display switch, user macro, machining information and parameter configuration (user)*
Set	Relevant functions of tool setting	Tool setting, tool compensation*, coordinate setup, tool life management, auto tool measurement and thread repair
Program	User program management function	Edit new program*, select, copy, paste and delete programs from system disk, USB flash disk and network disk, program rename and sort, set marker
Diagnosis	Fault diagnosis, performance commissioning, intelligent function	1. Fault diagnosis function: Alarm message, alarm history, ladder diagram, PLC status, macro-variable, log and other functions; 2. Performance commissioning function: Servo adjustment 3. Intelligent function: QR code, fault record and screw load check.
Maintain	Hardware setup, parameter setup, system upgrade, basic information, data management and relevant maintenance functions	1. System hardware configuration and configuration sequence setup function: Device configuration 2. Setup function of common parameters: Parameter setup 3. Setup function of user parameters: Parameter setup* 4. System upgrade and commissioning function: Batch commissioning, data management, system upgrade, permission management and user setup 5. Registration, basic information and other functions: Registration, machine tool information, system information, process package and time setup
User-defined ** (MDI)	Related functions of manual data input	Dwell, clear, save, input

Description:

*While configuring the function set of standard version, for ease of operation, some identical soft key functions are configured in different function sets (function set can be configured as user required)

** For the non-Di series, MDI mode is set as the MDI function set

1.3 Basic Display Interface

HNC-808Di-TU can realize different application functions through function keys and function soft keys and display corresponding interfaces. The display interface of this system mainly includes machining display interface, program selection and editing interface, machining setup interface, parameter setup interface and fault alarm display interface, etc.

The operator can know current status and information of system through interface or have a man-machine conversation in the conversation area to realize command input, parameter setup and other operations.

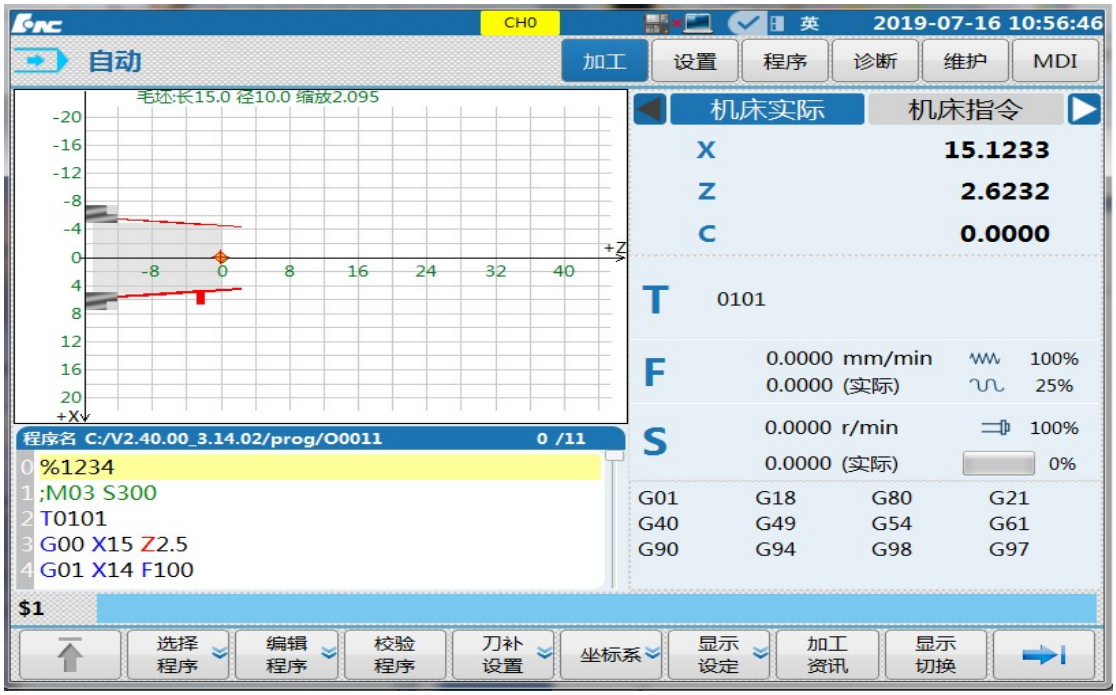
All interfaces are briefly introduced based on HNC-808Di-TU standard configuration.

1.3.1 Machining Display Interface

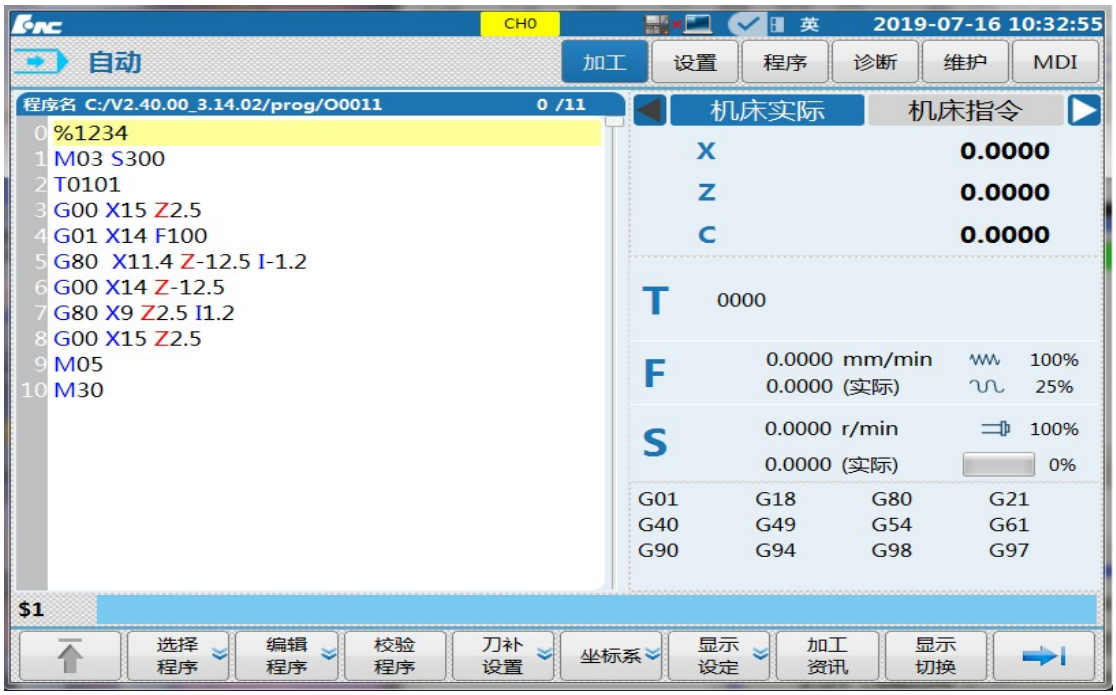
The machining display interface is convenient for the operator to observe the machining process. There are four display forms of large-character coordinate + program, joint coordinate, graphic path + program, and program. The 4 kinds of interfaces can be switched through the "Display Switching" function soft key to realize cyclic switching.

Big character coordinate + program display interface

Joint coordinate display interface



Graphics path + program display interface

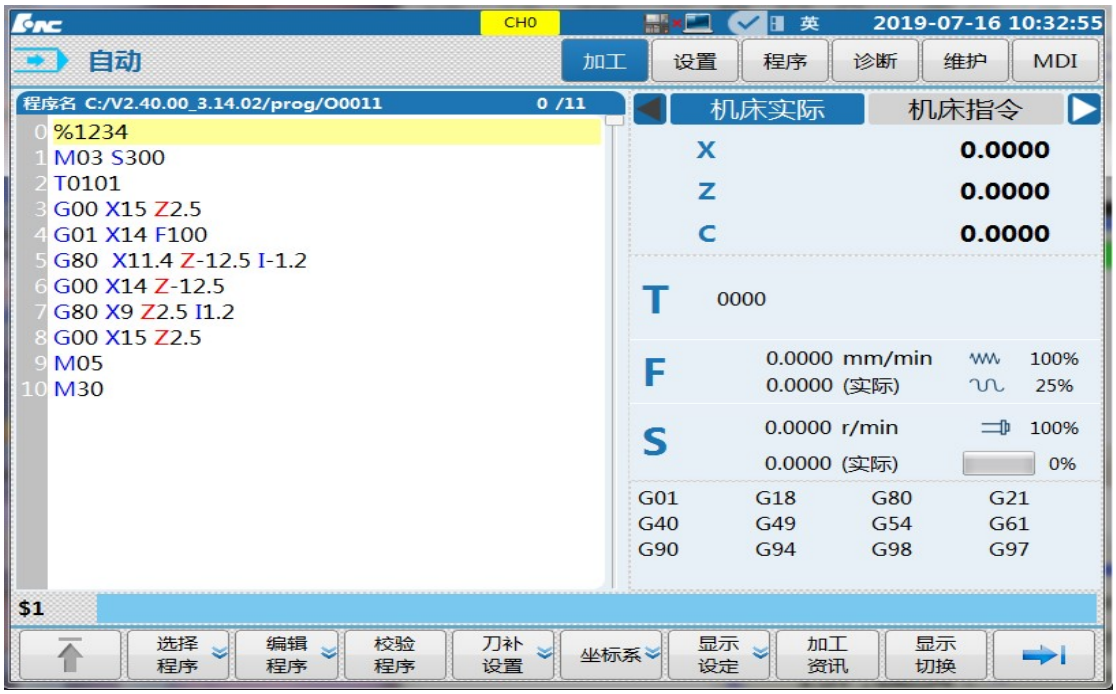


Program display interface

1.3.2 Program Selection and Editing Interface

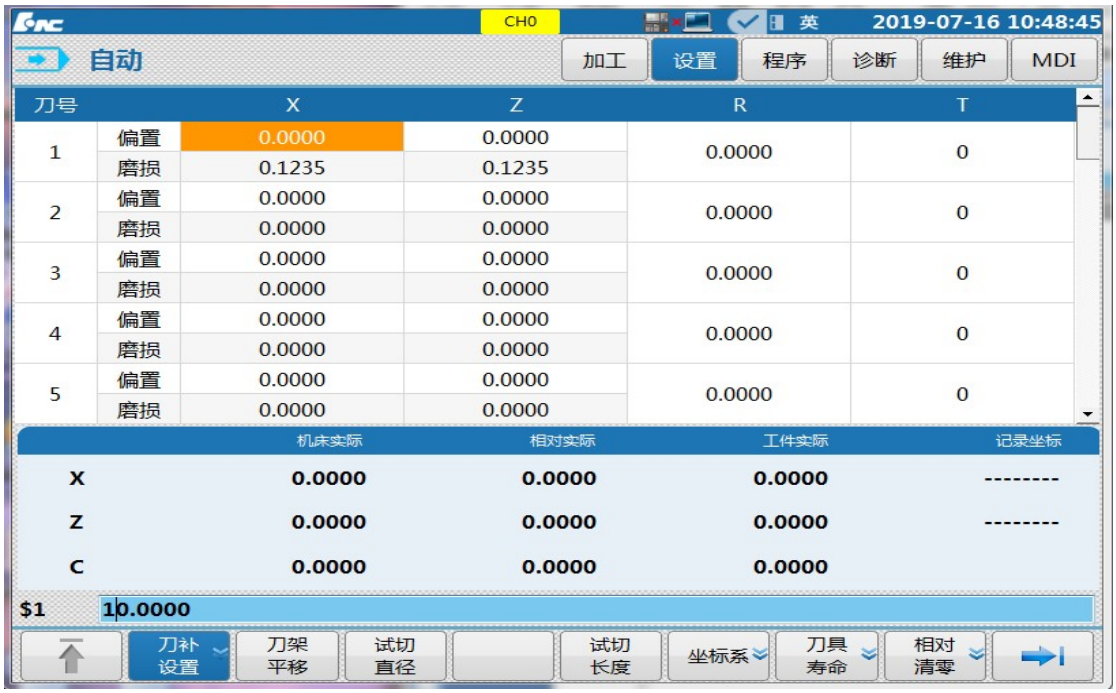
Users can select programs with cursor. When the cursor selects a program name in the list, the first blocks of the program will be displayed in the lower part of the screen in order to confirm the program searched.

Program selection interface



Program editing interface

1.3.3 Machining Setup Interface



Machining setup interface

1.3.4 Parameter Setup Interface

Parameter setup interface

2 Operating Equipment

2.1 System Host Panel (NC Panel)

2.1.1 System Host Panel Zoning

HNC-808Di-TU controller panel is 10.4 in. color LCD (resolution is 800×600). Panel zoning is shown below.

(1)---LOGO

(2)---USB interface

(3)---Alphabetical keyboard area

(4)---Number and character key area

(5)---Cursor key area

(6)---Function key area

(7)---Soft key area

(8)---Screen display interface area

2.1.2 Display Interface Zoning

The operation interface of HNC-808Di-TU is shown below:

(1) ---Title bar

- Machining mode: Working mode of the system can be switched among auto (run), single block (run), jog (run), increment (run), reset and emergency stop with corresponding keys on the control panel;
- System alarm message;
- Level 0 main menu name: Display currently activated main menu keys;
- USB flash disk connection and network connection;
- System sign, time.

(2) ---Graphics display window: Graphics displayed in this area differ with selected menu keys

(3) ---G code display area: Preview or display code of machining program.

(4) ---Input box: Enter information to be input in this column.

(5) ---Menu instruction bar: Operate system functions through function keys in the menu instruction bar.

(6) ---Axis status display area: Display coordinate position, pulse value, breakpoint position, compensation value and load current of axis

(7) ---Auxiliary function: T/F/S information area.

(8) ---G modal and machining information area: Display G modal and machining information during machining.

2.1.3 Definition of Host Panel Keys

The host panel includes

Simplified MDI keyboard area, function key area, soft key area.

MDI keyboard function

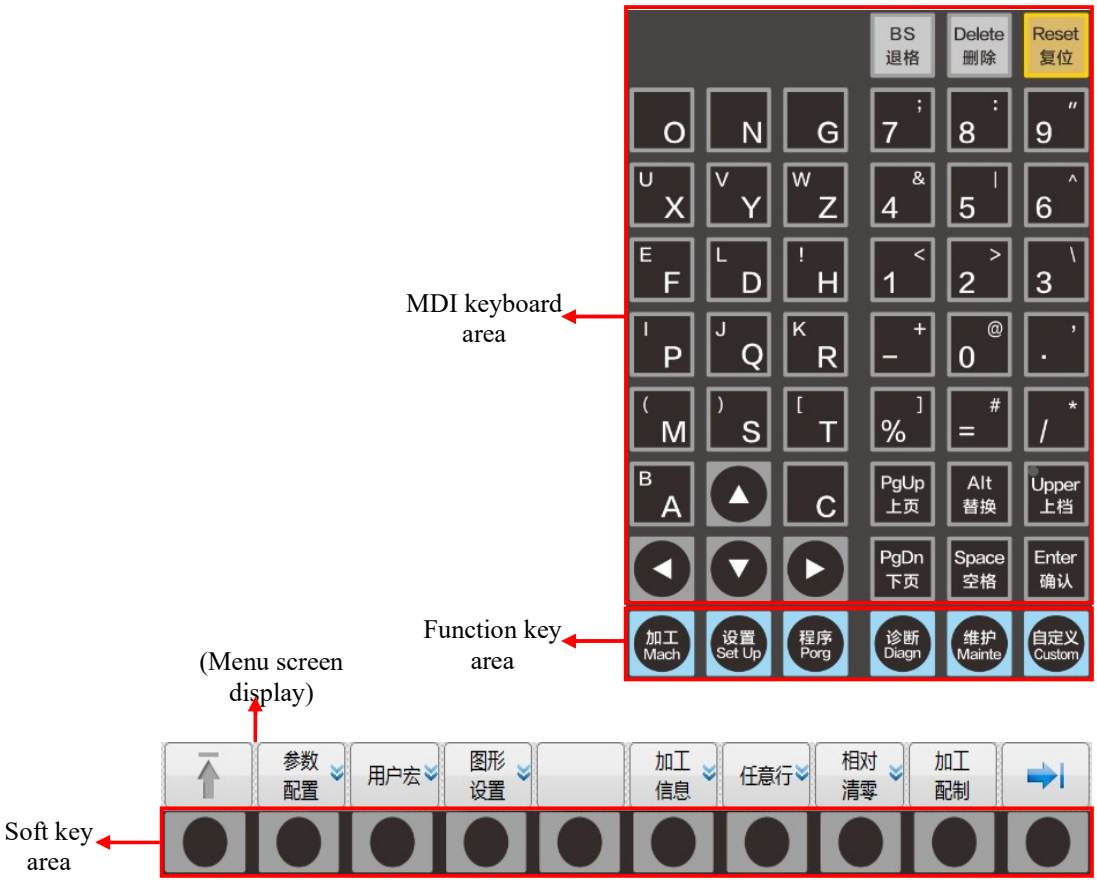
Input and edit instructions by this keyboard. Most keys have functions of upper shift key. Press "Upper shift" key and letter/number key simultaneously to input letter/number of upper gear key.

Function key function

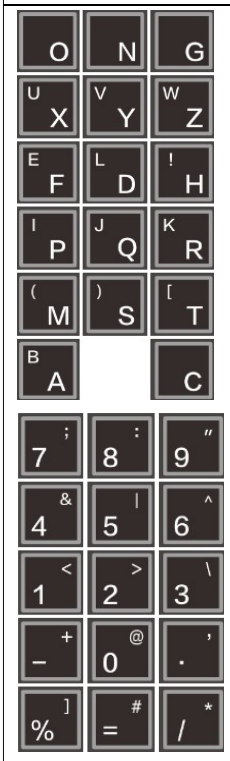
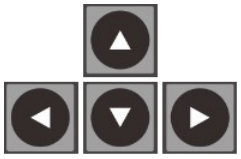
HNC-808Di-TU system has 6 function keys "Machining", "Setup", "Program", "Diagnosis", "Maintenance" and "User-defined", which correspond to different function sets and display interfaces (for specific functions, refer to chapter 3).

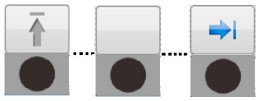
Soft key function

There are 10 soft keys below the screen of HNC-808Di-TU system, on which there are no fixed signs. The keys at the left and right ends are the ones to return to the previous menu and go to the next menu. All soft key functions correspond to menus displayed above them on the screen (for specific functions, refer to chapter 3).



Function of keys on MDI keyboard

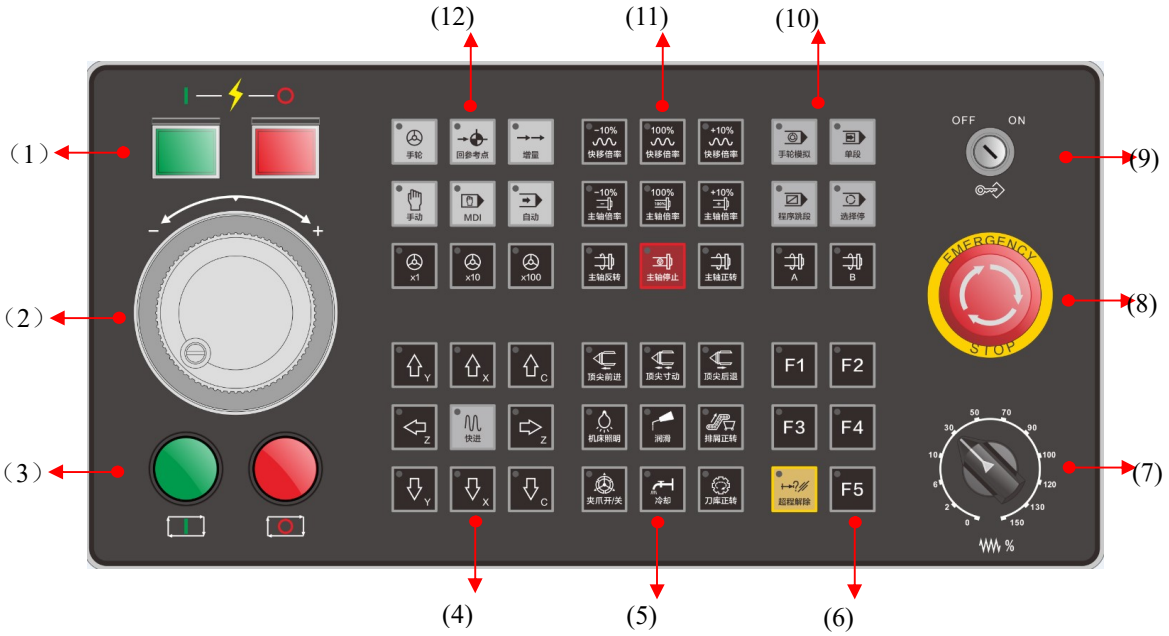
Key	Name/symbol	Functional description
	Character key (letter, number, symbol)/ 「 “Letter” 」 (such as「Y」)	Input letters, numbers and characters. Every character key has upper and lower characters. When the shift key and the character key are pressed simultaneously, the upper character is input; otherwise, the lower key is input.
	Cursor shift key/ 「Cursor」	Control the cursor to move horizontally and vertically.
	Program name symbol key/ 「%」	Based on the lower character, program name symbol of subprogram
	Backspace key/ 「Backspace」	Delete characters forward, and so on.
	Delete key/ 「Delete」	Delete current programs and characters, and so on.
	Reset key/ 「Reset」	CNC reset, feed, input stop, and so on.
	Alternate key/ 「Alt」	Press 「Alt」+「Cursor」 to switch content of the display frame (position, compensation and current, and so on) on the top right corner of the interface. (Detailed in 3.2.1.5); Press 「Alt」+「P」 for screenshot.

	Shift key/ 「Shift」	When the shift key and the character key are pressed simultaneously, the upper character is input; otherwise, the lower key is input.
	Space key/ 「Space」	Move one blank space backward.
	Confirmation key/ 「Enter」	Open and confirm input.
	Page up or page down key/ 「Page up or page down」	Switch the previous and the next pages in the same display interface.
	Function key/ 〖Machining〗 〖Setup〗 〖Program〗 〖Diagnosis〗 〖Maintain〗 〖User-defined〗	Machining: Select function set required for automatic machining and corresponding interface. Setup: Select function set relating to tool setting and corresponding interface. Program: Select function set for user program management and corresponding interface. Diagnosis: Select function set for fault diagnosis, performance commissioning and intelligence and corresponding interface. Maintenance: Select relevant maintenance functions such as hardware setup, parameter setup, system upgrade, basic information and data management and corresponding interface. User-defined* (MDI): Select relevant functions of manual data input and corresponding interface.
	Soft key/ 『↑』 『→』 『"Function"』	There are 10 unidentified keys below HNC-808Di-TU display screen, namely soft keys. In different function sets or levels, the functions correspond to those displayed on the screen. Main functions of soft keys are as follows: 1) Switch sub-interfaces in current function set; 2) Input corresponding operations in current function set, such as edit, modify and data input, and so on. In 10 soft keys, the leftmost key is to return to the previous menu, arrow is valid when it is blue and it is gray under level 1 menu of function set. In 10 soft keys, the rightmost key is to go to the next menu. Arrow is valid when it is blue. Press this key for cyclic switch among interfaces in menus of the same level (menus of the same level of this system has no more than 2 pages).

Note: In the descriptions in the subsequent chapter, key name will be replaced with key symbol.

2.2 Operation Panel of Machine Tool (MCP Panel)

2.2.1 Operation Panel Zoning Of Machine Tool



(1)---Power supply switch

(2)---Handwheel pulse generator

(3)---Cycle start/feed hold

(4)---Feed axis movement control key area

(5)---Machine tool control key area

(6)---Machine control extension key area

(7)---Feedrate override band switch

(8)---Emergency stop button

(9)---Editing lock ON/OFF

(10)---Operation control key area

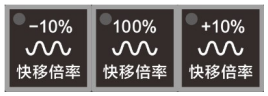
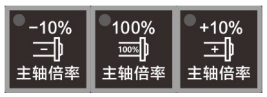
(11)---Speed magnification control key area

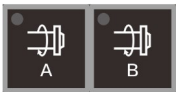
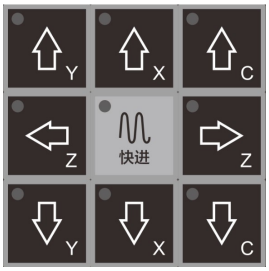
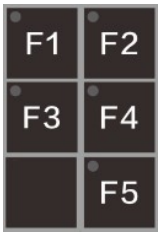
(12)---Working mode selection key area

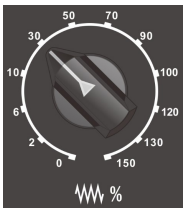
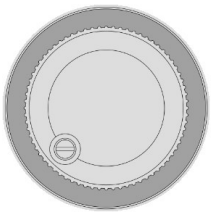
2.2.2 Definition of Operation Panel

This manual describes function and status of all keys based on standard PLC of HNC-808Di-TU system. In case of discrepancies, please refer to the specification provided by the machine tool manufacturer.

Key	Name/symbol	Functional description	Working mode at valid state
	Handwheel Working mode key /【Handwheel】	Select handwheel mode.	Handwheel
	Reset Working mode key /【Reset】	Select reset mode key.	Reference point return
	Incremental Working mode key /【Incremental】	Select incremental mode.	Incremental
	Jog Working mode key /【Jog】	Select jog mode.	Jog
	MDI Working mode key /【MDI】	Select MDI mode.	MDI
	Auto Working mode key /【Auto】	Select auto mode.	Auto
	Single block ON/OFF key /【Single block】	1) Switching of block-by-block operation or continuous operation program. 2) The indicator light lights up when the single block mode is valid.	Auto, MDI (Including single block)
	Handwheel ON/OFF key / [Handwheel]	1) To enable and disable the handwheel function. 2) When this function is enabled, the handwheel controls the tool to run as per the programed path. While rotating the handwheel clockwise, continue running the subsequent programs; while rotating the handwheel counterclockwise, backspace the run programs in the reverse direction.	Auto, MDI (Including single block)

	Block skip ON/OFF key /[block skip]	1) To specify whether the bock is skipped when a block is prefixed with "/"	Auto, MDI (Including single block)
	Optional stop ON/OFF key / [Optional stop]	1) To specify whether to stop the program When the program executes “M00”; 2) If this key has been pressed before the program is executed (indicator light lights up), when the program executes “M00”, the feedhold is performed, then press cycle start to continue running the subsequent programs. If this key is not pressed, the program is executed consistently.	Auto, MDI (Including single block)
	Overtravel release key / [Overtravel release]	1) Cancel machine tool limits; 2) Press and hold this key to release alarm and operate the machine tool.	Handwheel, jog, incremental
	Cycle start key / [Cycle start]	Start running program and MDI command.	Auto, MDI (Including single block)
	Feed hold key / [Feed hold]	Suspend the running of program and MDI command.	Auto, MDI (Including single block)
	Incremental/handwheel magnification key / [Incremental magnification]	When the handwheel rotates for 1 graduation or "Manual control of axis feed key" is pressed once, the corresponding movement distance of the machine tool is 0.001mm/0.01mm/0.1mm.	Handwheel, incremental, jog, reset, auto, MDI (including single block and handwheel)
	Rapid traverse speed override key / [Rapid traverse override]	Override of rapid traverse speed.	
	Spindle magnification key / [Spindle magnification]	Override of spindle speed.	
	Spindle control key / [Spindle CW/CCW rotation]	Control CW rotation, CCW rotation and stop of spindle.	Handwheel, incremental, jog, MDI (including

	Power head control key / [Power head]	1) Control of CW/CCW rotation of power head; 2) Press this key to switch between rotation/stop of power head.		single block and handwheel)
	Manual control of axis feed key / [Axis feed]	1) Control movement and direction of all axes in jog or incremental mode; 2) Select handwheel control axis in handwheel mode; 3) When all axes are pressed under jog mode, the axes run as per feedrate. When "Rapid traverse" is also pressed at the same time, the axes run as per rapid traverse speed.		Handwheel, incremental, jog
	Machine tool control key / [Machine tool control]	Manual control of auxiliary actions of machine tool	Machine tool center feed, jog, retract. Clamping jaw ON/OFF. CW rotation of magazine.	Handwheel, incremental, jog (and spindle stop)
			Lighting of machine tool. Lubrication. Chip removal CW rotation. Cooling.	Handwheel, incremental, jog, reset, auto, MDI (including single block and handwheel)
	Machine tool control extension key / [Machine tool control]	Manual control of auxiliary actions of machine tool.		Machine tool manufacturer is set as needed
	Program protection switch / [Program protection]	Protect program from being modified arbitrarily.		Handwheel, incremental, jog, reset, auto, MDI (including single block and handwheel)
	Emergency stop button / [Emergency stop]	Have the system and the machine tool immediately enter the stop state and close all output.		

	Feedrate knob / [Feedrate]	Feedrate override.	
	Handwheel / [Handwheel]	Control movement of machine tool. (When the handwheel function is valid, it can control the machine tool to move according to programed path).	
	System power-on / [Power-on]	Control power-on of CNC device.	
	System power-off / [Power-off]	Control power-off of CNC device.	

Note:

To simplify editing, keys are divided into working mode key, function key, function soft key, NC key, MCP key, return to the previous menu key and continued menu key and are identified using symbols in the following table.

Key name	Working mode key	Function key	Function soft key	NC key	MCP key	Return to the previous menu key	Continued menu key
Key symbol	【】	〖〗	『』	「」	[ ]	『↑』	『→』

In the descriptions in the subsequent chapter, key name will be replaced with key symbol

3 Display Interface

3.1 Display Interface Selection and Menu Structure

3.1.1 Common Operation Of Interface and Menu Selection

1) There are 6 function keys on the NC panel, which can be used to select corresponding function sets and display interfaces.

2) There is a group of function menus in the lower part of display interface and function menu is selected by soft key.

3) Every group of function menus consists of 10 soft keys (space key is often reserved), among which the leftmost key is to return to the previous menu (『↑』), the rightmost key is to go to the next menu (『→』) and arrow is valid when it is blue.

4) The interface displayed when function key is selected after startup for the first time is the default interface of the function set and the function menu below is level 1 main menu. Extension menu of this level can be searched by『→』.

5) Menus of all levels under function set has at most 1 main menu and 1 extension menu. Press 『→』 for cyclic switch, at this time, only menu changes and interface does not change.

6) Interface selection before switching function set will be memorized. That is, while switching back to this function set, the displayed function menu and interface are the menu and interface upon the previous exit.

7) Function sets of this system have at most a 4-level menu structure. Press the right function soft key marked with "≫" to search the sub-menu. Return to the previous menu by 『↑』.

8) For configuration of soft keys of all levels of menus, the standard version of this system has set personalized display interface and menu based on actual needs of users. The user can configure special needs.

9) Generally data input and other man-machine dialog boxes can be opened using corresponding soft keys, but for some data input with high safety requirements, activate the input box using "Enter" (「Enter」) and input data or parameters.

10) When the man-machine dialog box does not exit, function sets cannot be switched by function keys.

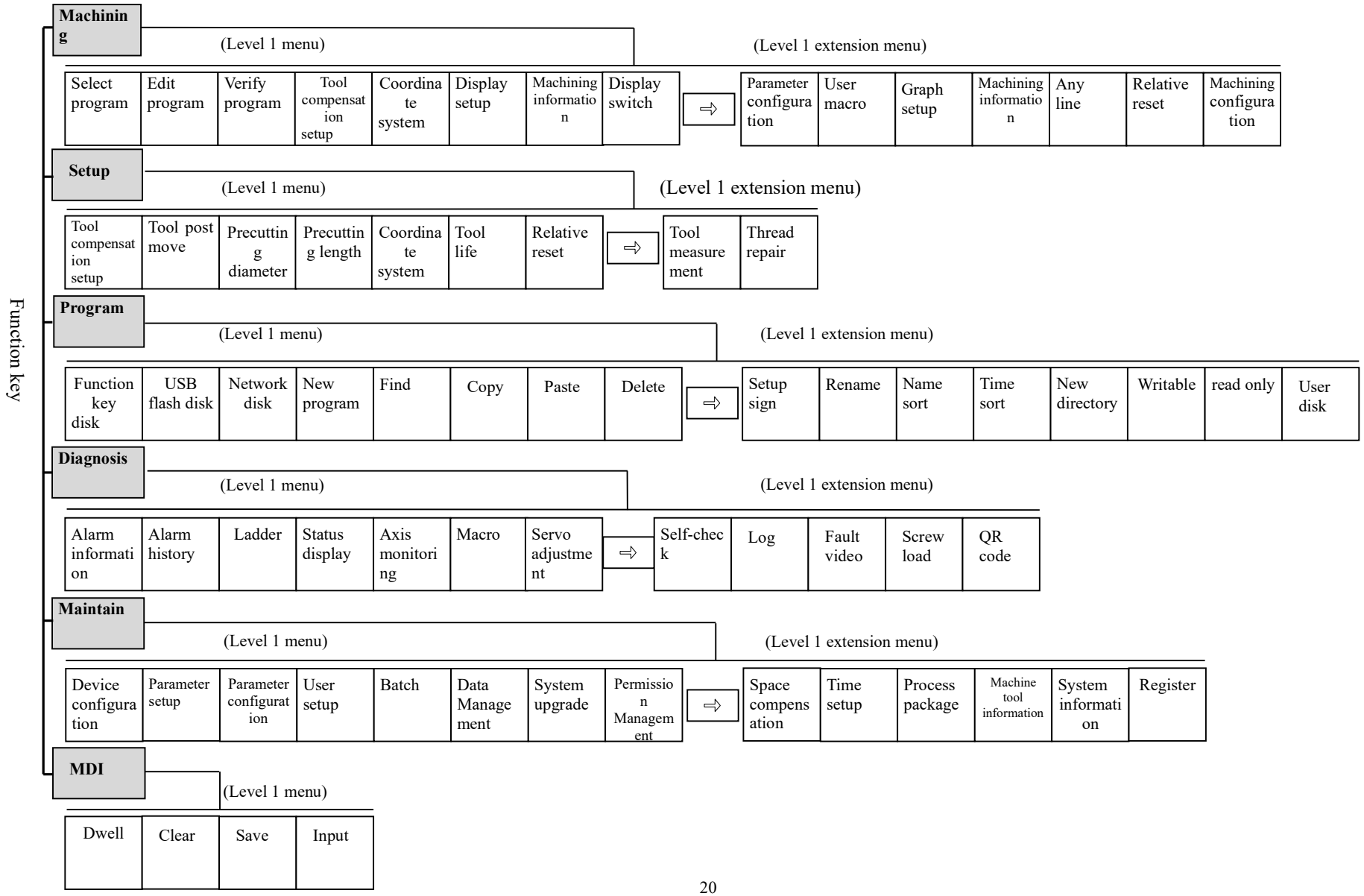
11) Exit mode of man-machine dialog box:

- Correctly input data and press "Enter" (「Enter」). After data is correctly entered., exit from the dialog box.
- If current input is activated improperly or abandoned, press "Reset" (「Reset」) to exit from the dialog box and the input data will not be recorded.

3.1.2 Function Menu Structure

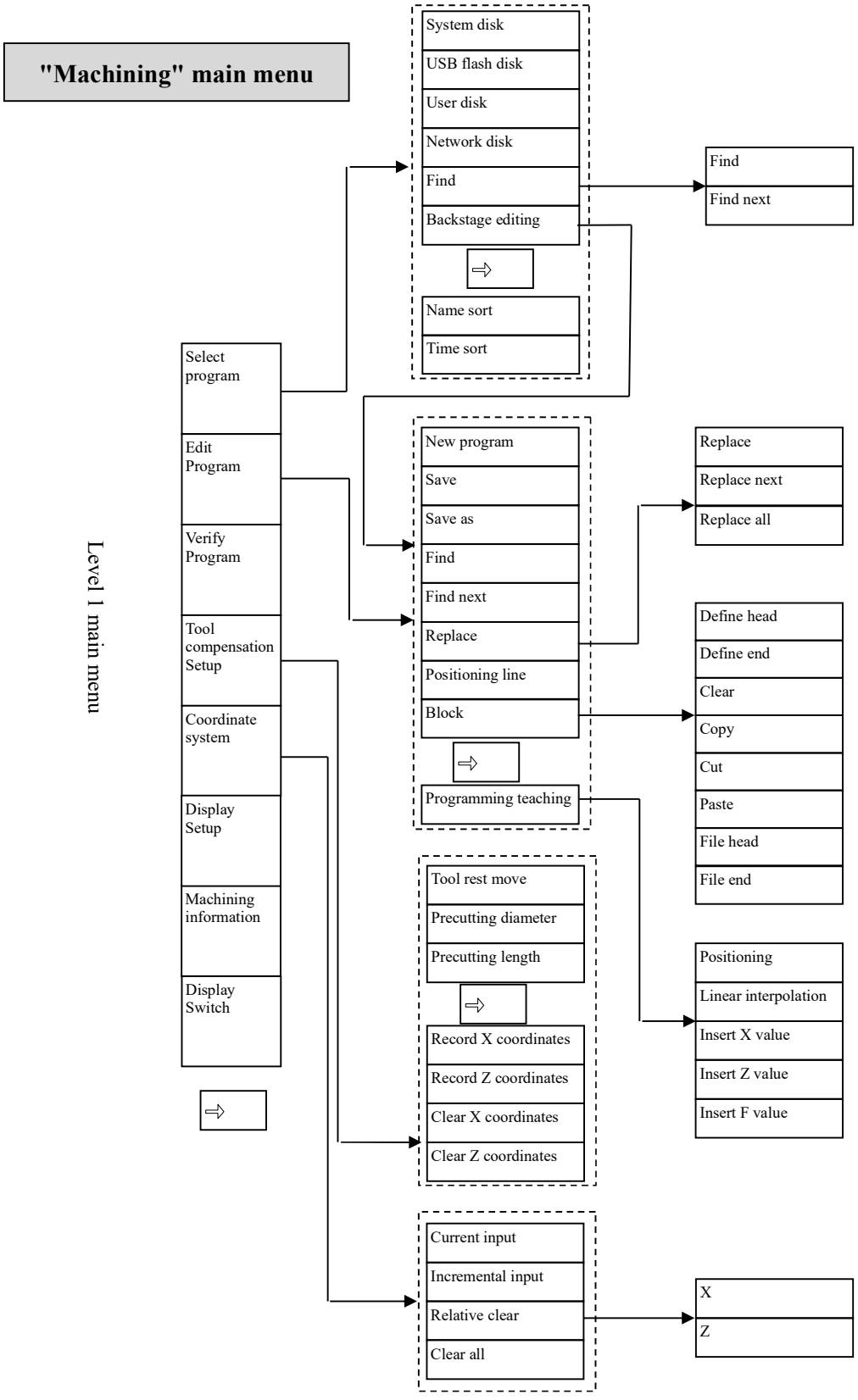
This menu tree is the basic structure chart of standard version of this system, and increase, decrease, sort or position of its function menus differs with configuration of user permission, parameter setup and machine tool manufacturer. For details, refer to the specification provided by the machine tool manufacturer.

1) Level 1 menu of all function sets

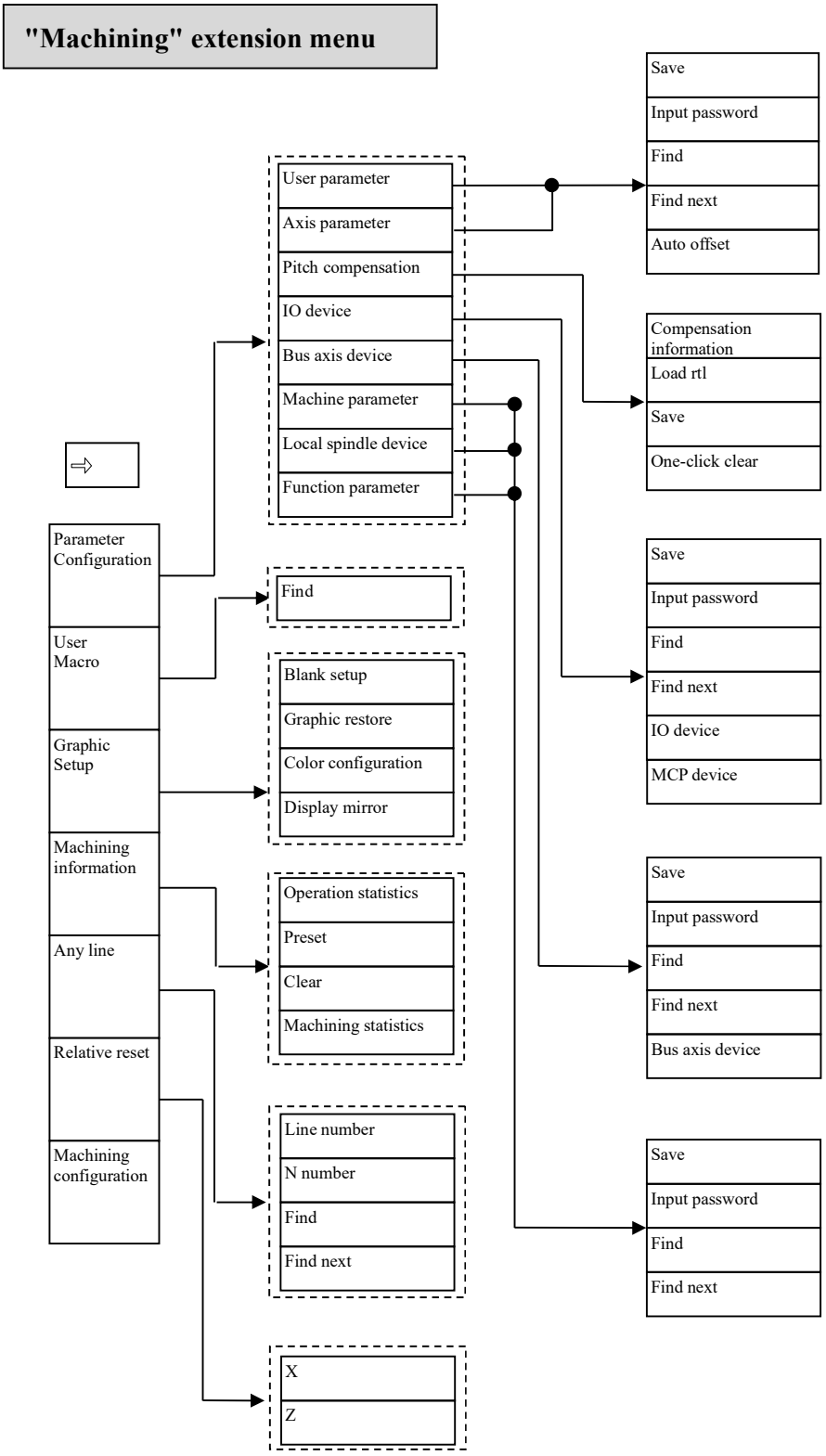

Function key
Machining
(Level 1 menu)
(Level 1 extension menu)
Select program
Edit program
Verify program
Tool compensation setup
Coordinate system
Display setup
Machining information
Display switch
Parameter configuration
User macro
Graph setup
Machining information
Any line
Relative reset
Machining configuration
Setup
(Level 1 menu)
(Level 1 extension menu)
Tool compensation setup
Tool post move
Precutting diameter
Precutting length
Coordinate system
Tool life
Relative reset
Tool measurement
Thread repair
Program
(Level 1 menu)
(Level 1 extension menu)
Function key disk
USB flash disk
Network disk
New program
Find
Copy
Paste
Delete
Setup sign
Rename
Name sort
Time sort
New directory
Writable
read only
User disk
Diagnosis
(Level 1 menu)
(Level 1 extension menu)
Alarm information
Alarm history
Ladder
Status display
Axis monitoring
Macro
Servo adjustment
Self-check
Log
Fault video
Screw load
QR code
Maintain
(Level 1 menu)
(Level 1 extension menu)
Device configuration
Parameter setup
Parameter configuration
User setup
Batch
Data Management
System upgrade
Permission Management
Space compensation
Time setup
Process package
Machine tool information
System information
Register
MDI
(Level 1 menu)
Dwell
Clear
Save
Input

2) Menu structure of "Machining" function set

(1) "Machining" main menu

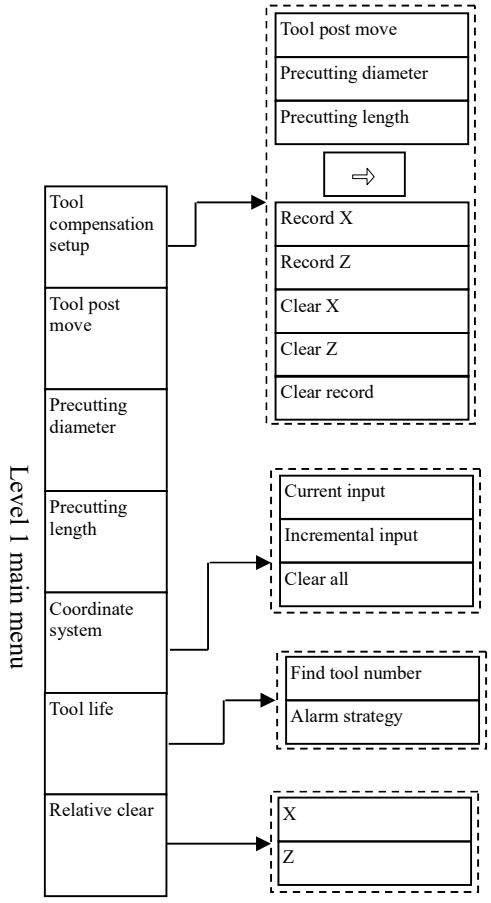


(2) "Machining" extension menu

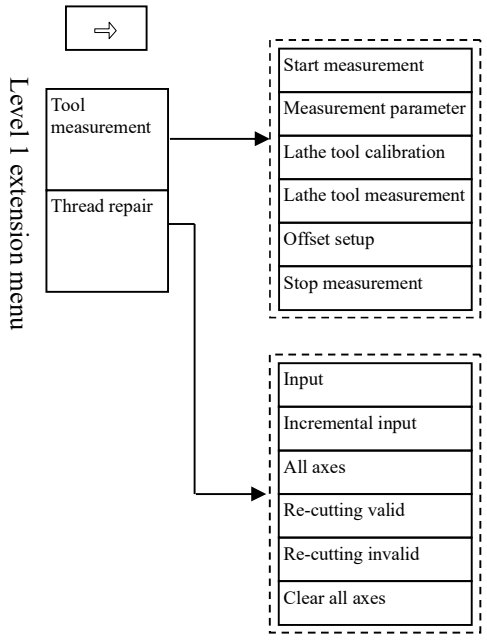


3) Menu structure of "Setup" function set

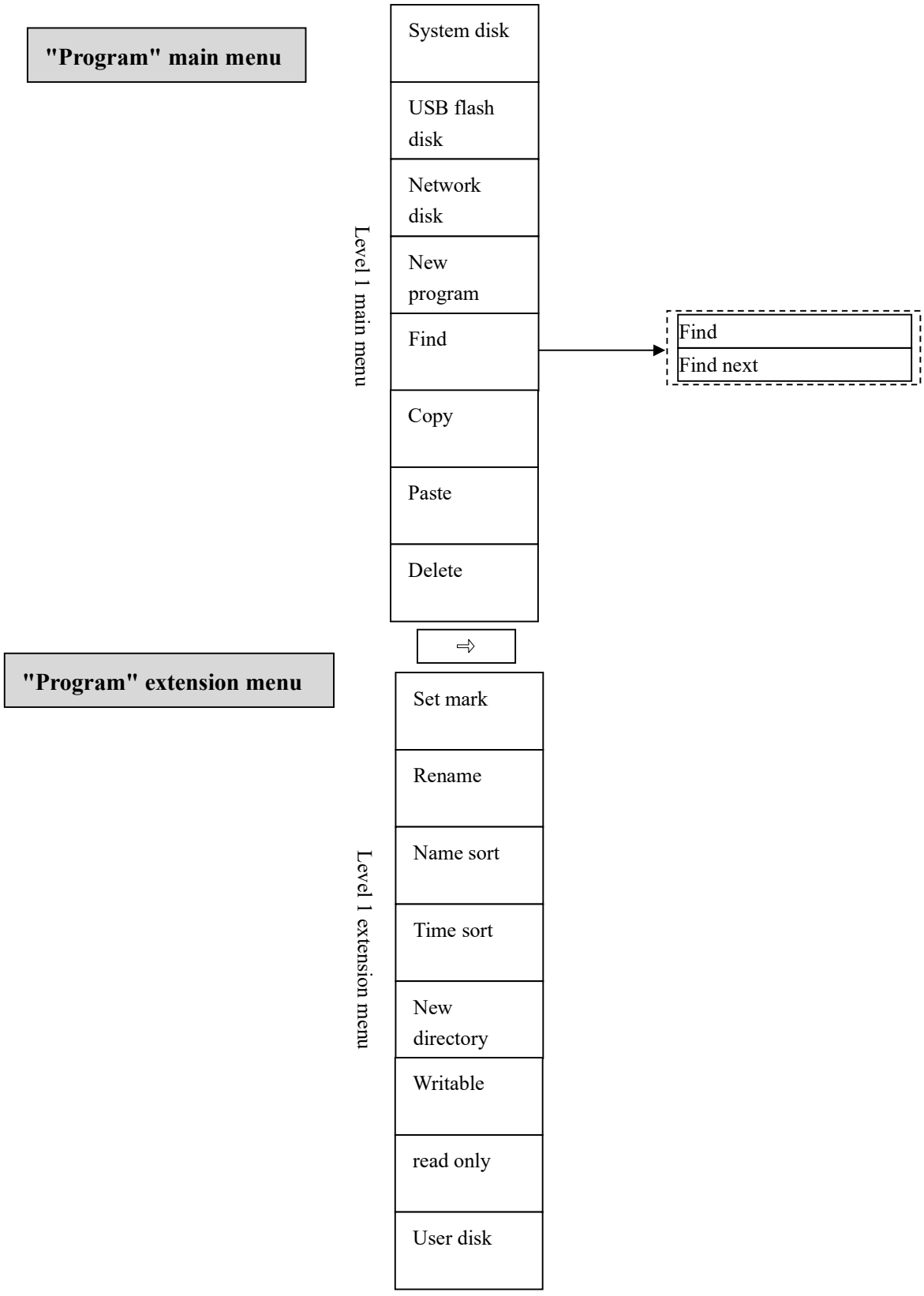
"Setup" main menu



"Setup" extension menu

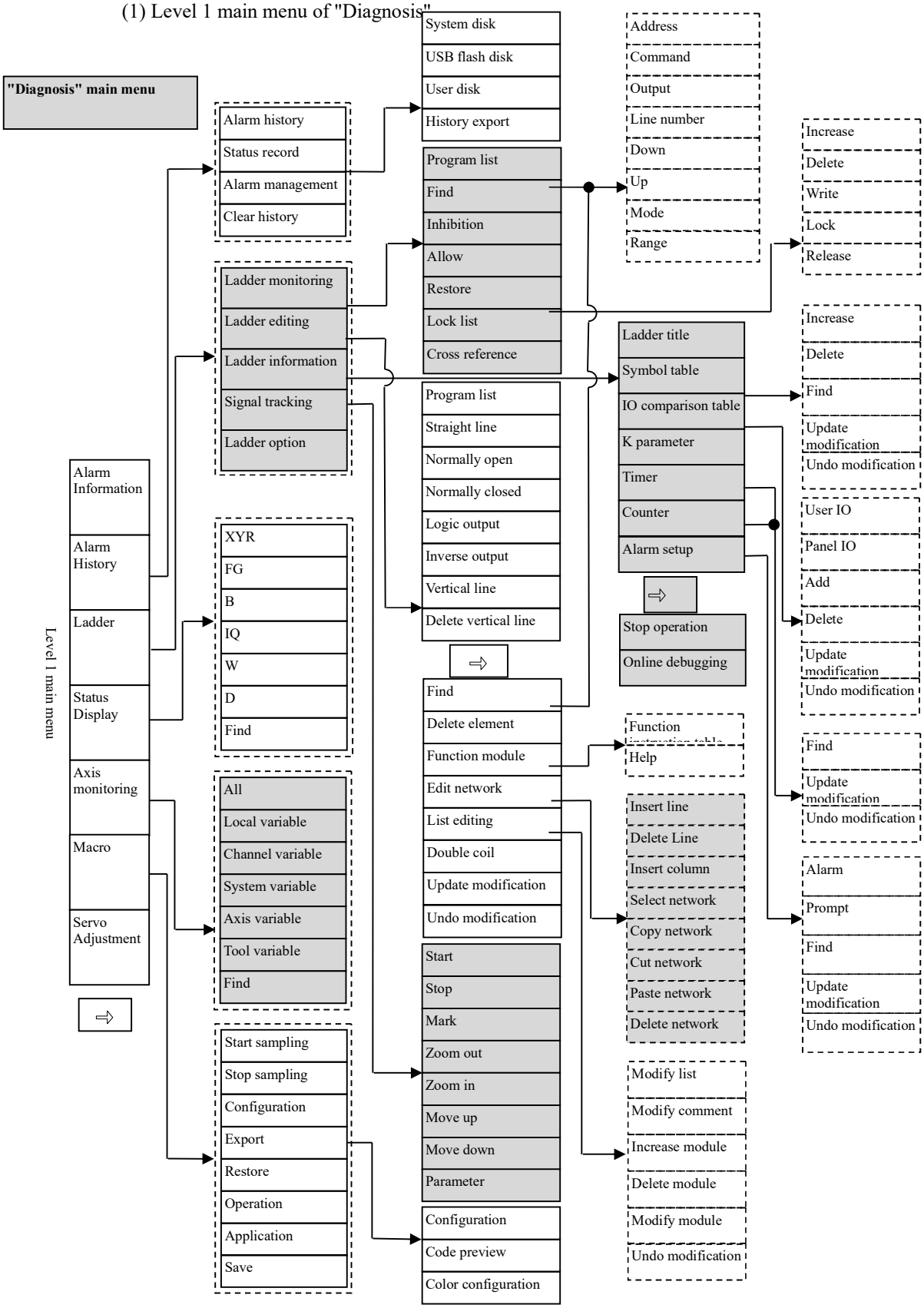


4) Menu structure of "Program" function set

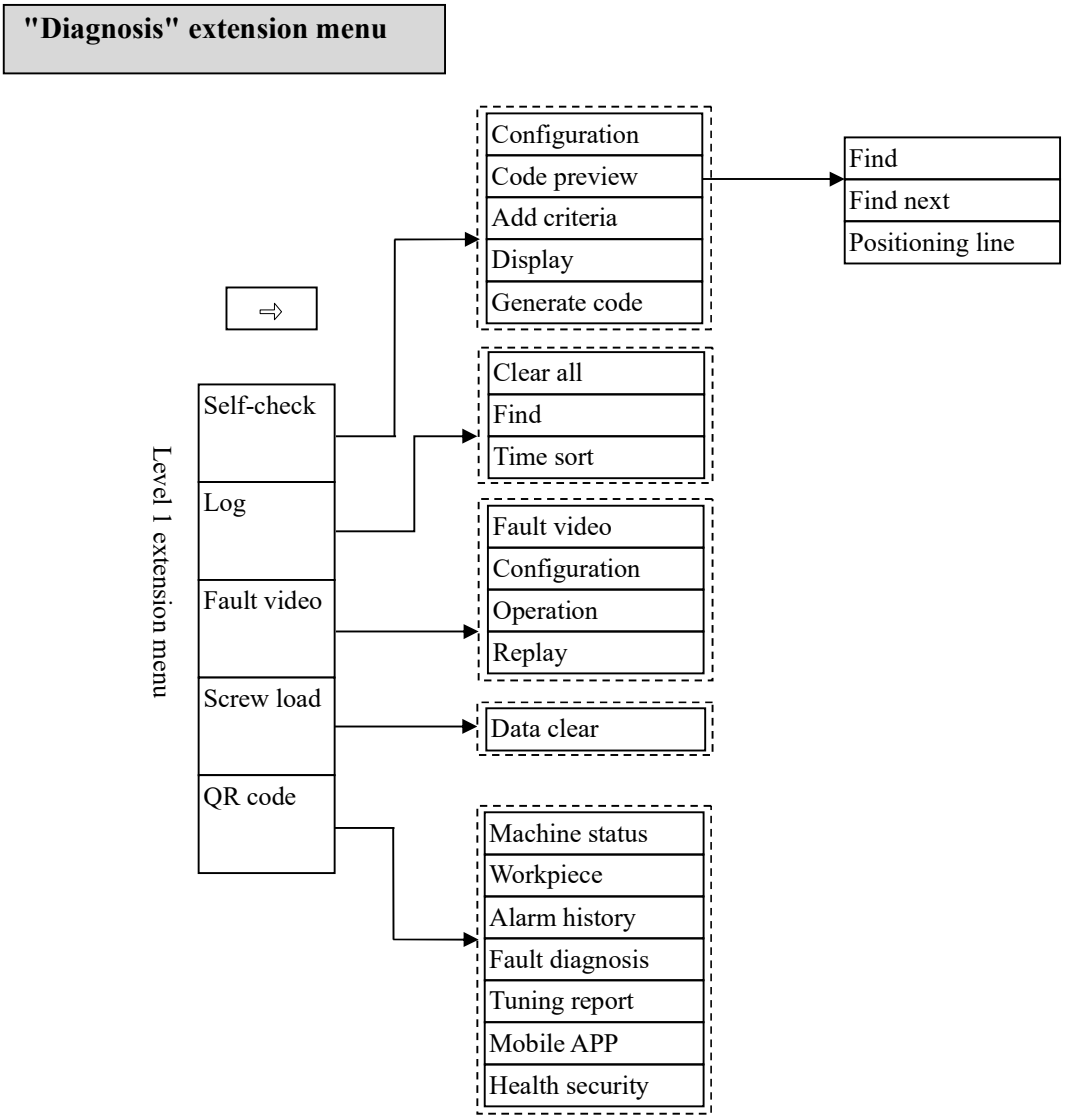


5) Menu structure of "Diagnosis" function set

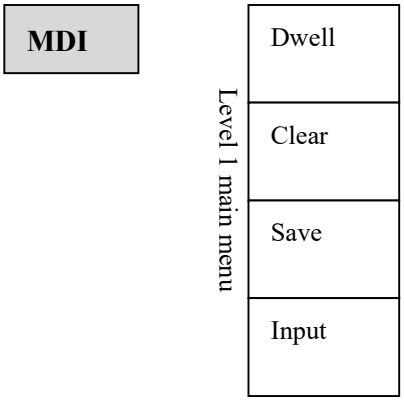
(1) Level 1 main menu of "Diagnosis"



(2) Level 1 extension menu of "Diagnosis"

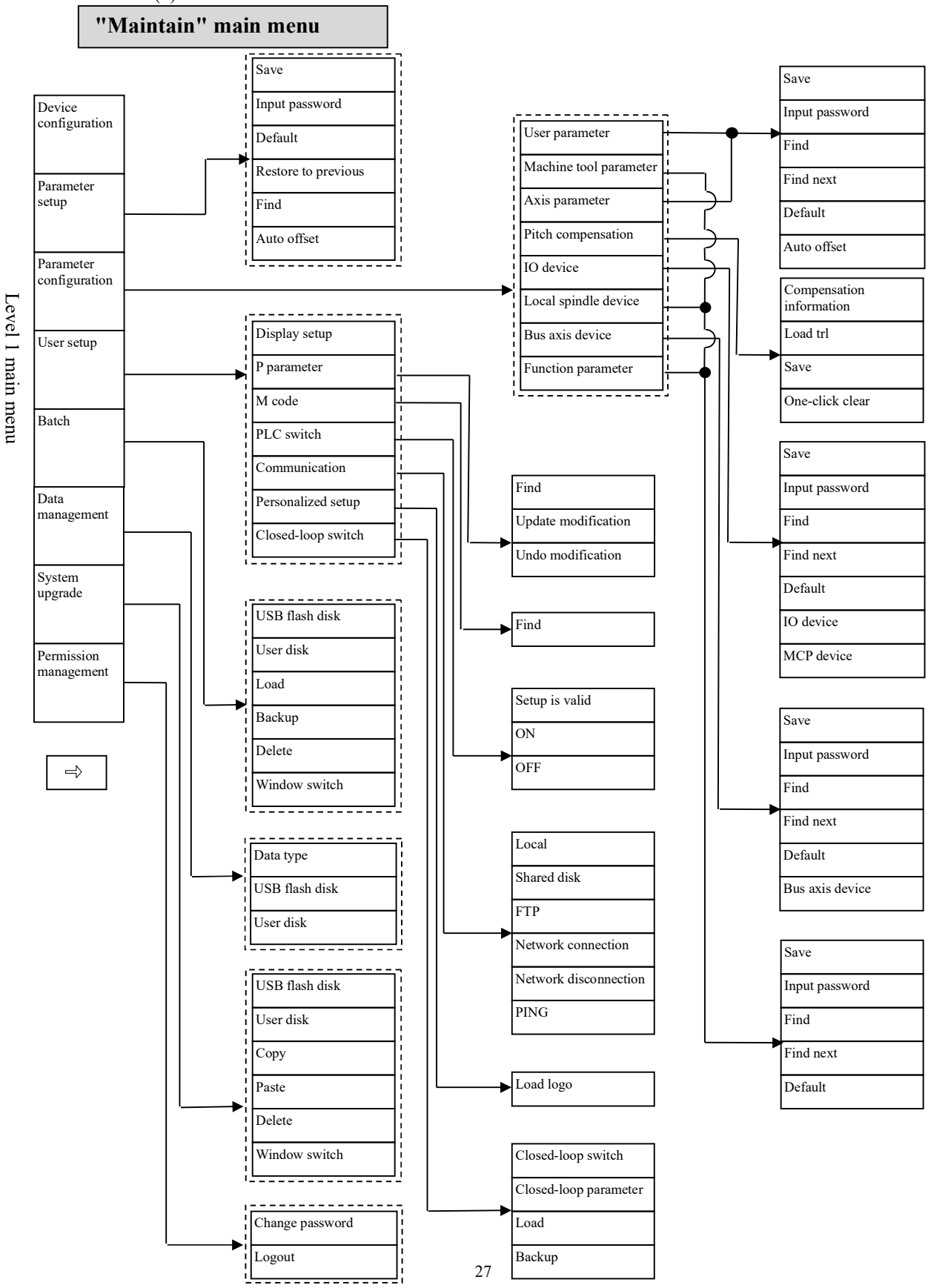


6) Menu structure of "User-defined (MDI)" function set

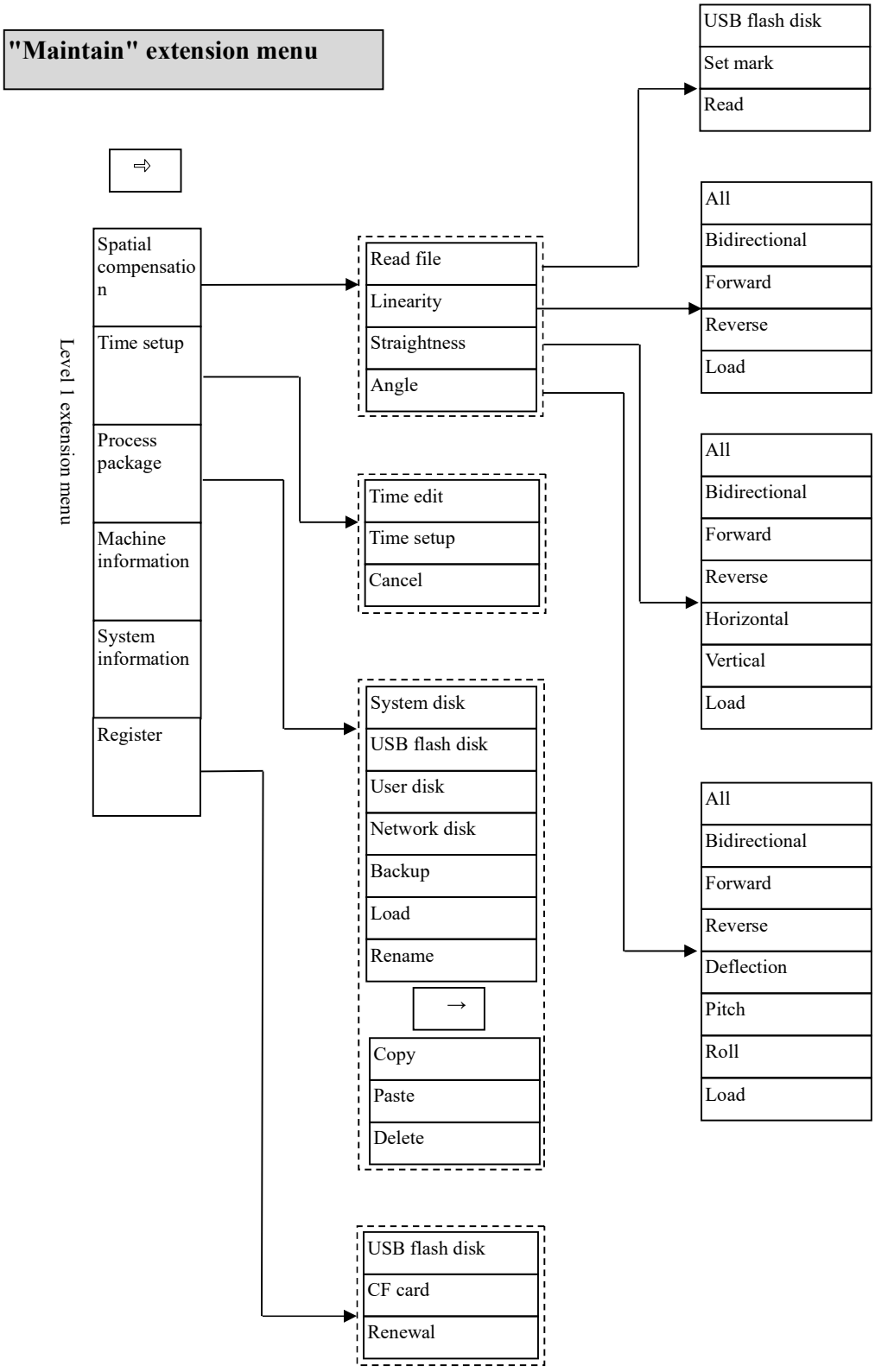


7) Menu structure of "Maintain" function set

(1) Level 1 main menu of "Maintain"



(2) Level 1 extension menu of "Maintain"



3.2 Display Interface and Basic Operation of "Machining" Function Set

3.2.1 Interface and Function of "Machining" Function Set

"Machining" function set integrates all functions necessary for parts machining and is compatible with some functions of function set "Setup", "Program" and "Diagnosis", which greatly reduces interface switching. Operations that can be conducted under the function set include selecting machining program, selecting edit program, editing new programs, verifying programs, tool setting, coordinate setup, any line, parameter configuration, coordinate display, graphic display, machining information display and user macro query. Level 1 main menu and level 1 extension menu of "Machining" function set are shown below.

Select program	Edit program	Verify program	Tool compensation setup	Coordinate system	Display setup	Machining information	⇨

Display switch	Parameter configuration	User macro	Graphics setup	Machining information	Any line	Relative clear	Machining configuration

Select program: Select a program from the target disk (system disk, USB flash disk, user disk and network disk) and load it as a machining program; or select a program and edit it through backstage editing; and edit a newly created program.

Edit program: Edit the loaded program, namely current machining program. A running program cannot be edited.

Verify program: Enable this function under "Auto" or "Single block" mode to quickly verify current loading program and the machine tool does not run.

Tool compensation setup: Tool offset and wear value can be set under the sub-interface of this function, or tool offset can be set through 『precutting diameter』 and 『precutting length』 soft keys; tool offset value can be increased or decreased by 『Tool post move』 soft key.

Function and operation of "Tool compensation setup" under "Setup" function set are the same as those of "Tool compensation setup" under "Machining" function set.

Coordinate system: This function can be used to set values of the workpiece coordinate system through direct input, current value input and incremental input mode.

Function and operation of "Coordinate system" under "Machining" function set are the same as those of "Coordinate system" under "Setup" function set. This

function will be introduced in "Setup" function set.

Display setup: This function can be used to set the display contents of joint coordinates and the display contents of big character coordinates.

Machining information: Cyclic switching display: Contents of "Machining information" and "G command modal"

Display switch: Cyclic switching display: Big character coordinate + program, joint coordinate, graph+ program, program

Parameter configuration: Parameters relating to machining and commissioning can be set here.

User macro: Display the value of some canned cycle macro-variables.

Graphics setup: Workblank setup and adjustments such as zoom in and zoom out in graphic display.

Machining information: Display and setup of machining statistics.

Any line: Operation and setup of any line function. This function is introduced in the subsequent chapters.

Relative clear: Set relative zero point during tool setting for ease of tool setting calculation.

Machining configuration: When there is no commands F/S in the program, it can be set by this function.

3.2.1.1 Machining set interface zoning

After startup, press 〖Machining〗 function key to enter the default interface of "Machining" function set, as shown below.

(1) Area---Title bar

(2) Area---Coordinates and graphics display window: Coordinate, graph and program display area.

(3) Area---G code display area: Preview or display codes of machining program.

(4) Area---Input box: Enter information to be inputted in this column.

(5) Area---Menu command bar: Operate system functions through function keys in the menu command bar.

(6) Area---Axis status display area: Display coordinate position, pulse value, breakpoint position, compensation value and load current of axis.

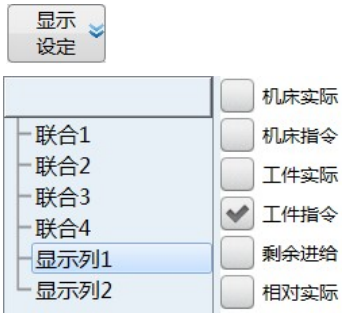
(7) Area---Auxiliary function: T/F/S information area.

(8) Area--Machining information area: Display G modal, program progress and workpiece quantity during machining.

3.2.1.2 Graphics and G code area display switching

For the switching of graphics and G code areas (2) and (3) display, press 『Display switch』 soft key under the interface to switch the display interface switches among 4 interfaces big character coordinate+ program, joint coordinate, graphics + program and program. (Detailed in 1.3.1)

3.2.1.3 "Big character coordinate" display setup in coordinate graphics display area



For big character display setup in coordinate graphics display area (2), press 『Display setup』 soft key under the interface to enter the sub-menu, select "Display column 1" and "Display column 2" and set big character coordinate content in the "Big character coordinate+ program" interface (for details, refer to 3.6.7.1).

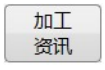
3.2.1.4 "Joint coordinate" display setup in coordinate graphics display area

For joint coordinate display setup in coordinate graphics display area (2), press 『Display setup』 soft key under the interface to enter the sub-menu, select "Joint 1-4" and set 4 coordinate contents in the "Joint coordinate" interface (for details, refer to "User setup" in 3.6.7.1) .

3.2.1.5 Switching of machining and commissioning information area display

For display switching of machining and commissioning information area (6), press 「Alt」+「Left and right cursors」 on the MDI keyboard to display the following items and values successively: Machine actual, machine command, workpiece actual, workpiece command, remaining feed, relative actual, relative command, breakpoint position, tracking error, workpiece zero, zero offset, compensation value, actual coordinate 2, synchronous error, handwheel offset, Z pulse offset, Z pulse interval 1, Z pulse interval 2, actual speed, motor position, command pulse, actual pulse, motor speed, waveform frequency, load current and temperature.

3.2.1.6 Switching of machining information area display



For display switching of machining information area (8), press 『Machining information』 soft key under the interface to switch G modal, machining quantity and other information.

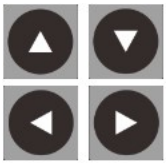
3.2.2 "Select Program" Sub-interface

Main function of "Select program" sub-interface includes selecting machining program, selecting editing program, editing and creating new programs. Existing programs in system disk, USB flash disk and network disk are optional.

Editing program and creating new programs are realized by "Backstage editing" in the sub-menu and the machine tool should not be at running status while editing the current machining program.

Press 〖Machining〗 function key to enter the level 1 menu of "Machining" function set and press 『Select program』 soft key to enter the interface, as shown below.

3.2.2.1 Select a program in the USB flash disk and load it as a current machining program



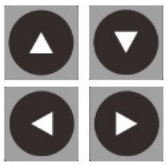
- Press 『Select program』 to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys 『System disk』, 『USB flash disk』, 『User disk』 and 『Network disk』, to enter corresponding program source disks;
- Press 「Cursor」 or 「PgUp/PgDn」 to select program file to preview program;

- Press 「Enter」 to load the selected program as a current

machining program and the interface returns to the previous menu and interface. After that, parts can be processed.

Note: If error is reported while loading a program, press 「Reset」 to clear it and press 『↑』 to return to level 1 interface);

3.2.2.2 Select a program in the directory as a current machining program



- Press 『Select program』 to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys 『System disk』, 『USB flash disk』, 『User disk』 and 『Network disk』, to enter corresponding program source disks;
- Press 「Cursor」 or 「PgUp/PgDn」 to select the file directory;

- Press 「Enter」 to activate the selected directory, enter the directory and display program files under it.
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the program file name;

- Press 「Enter」 to load the selected program as the current machining program and the interface returns to the previous menu and interface. After that, parts can be processed.

3.2.2.3 Exit from file directory

When the cursor is on a file name under the file directory, exit from the directory as below:

- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor onto the directory item (return to the previous directory identifier.);
- Press 「Enter」 to exit from current directory.

3.2.2.4 Edit current machining program in background

The current machining program cannot be edited while the program is running, but when it is not running, the background editing function can be used to edit the current processing program.

- Press "Program selection" soft key to enter the "Select program" sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 to select the file name of the current machining program;

- Press『Backstage editing』soft key to enter the editing interface to edit the current machining program;
- After editing or modification, press 『Save file』 soft key and a prompt message Saved will be given, then return to the previous interface or other operations.
- Before a file is saved, a prompt message "Save or not" will be given. Press 「Y」 to save the file and 「N」 or 「Reset」 not to save the file.

Note: When a program is not selected under "Select program" sub-interface (when the cursor is on the file directory), it is not allowed to enter the "Backstage editing" sub-interface.

3.2.2.5 Edit non-current machining program in background

- Press "Program selection" soft key to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys 『System disk』,『USB flash disk』,『User disk』 and『Network disk』, to enter corresponding program source disks;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the selected program file to preview the program;
- Press 『Backstage editing』 soft key to edit the program;
- After editing or modification, press 『Save file』 soft key and a prompt message Saved will be given, then return to the previous interface or other operations.
- Before a file is saved, a prompt message "Save or not" will be given. Press 「Y」 to save the file and 「N」 or 「Reset」 not to save the file.

Note: When there is amprogram in the "Backstage editing" interface, loading status of current machining program is not affected

3.2.2.6 Edit and create new programs in the background

- Press "Program selection" soft key to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys 『System disk』,『USB flash disk』,『User disk』 and『Network disk』, to enter corresponding program source disks;

- Press "Backstage editing" soft key to enter the "Backstage editing" sub-interface;
- Press 『New』 soft key and a prompt message "Please enter file name: O temp" will be given in the input box; (press「Reset」to exit the interface)
- Enter a new program name (figure or letter) by MDI keyboard;
- Press 「Enter」 to confirm the new file name to enter the program editing area;
- After editing or modification, press 『Save』 soft key and a prompt message Saved will be given, then return to the previous interface or other operations.
- Before a file is saved, a prompt message "Save or not" will be given. Press 「Y」 to save the file and 「N」 or 「Reset」 not to save the file.

Note: When a new program is created in the "Backstage editing" interface, it will be loaded as the current machining program automatically.

3.2.3 "Program Editing" Sub-interface

The "Edit program" sub-interface is mainly used to edit current machining program and creat new programs.

Press〖Machining〗function key to enter the level 1 menu of "Machining" function set and press 『Edit program』 soft key to enter the interface, as shown below.

Input box

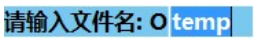
3.2.3.1 Edit current machining program

- Press 『Edit program』 soft key under the "Machining" function set and the cursor is in the editing area of current machining program to edit current machining program.
- After editing or modification, press 『Save』 soft key and a prompt message Saved will be given, then return to the previous interface or other operations;
- Before a file is saved, a prompt message "Save or not" will be given. Press 「Y」 to save the file and 「N」 or 「Reset」 not to save the file.

Note: 1. The machine tool should not be at running status while editing current machining program.

2. "Edit program" function cannot be used to edit other programs than current machining program. Otherwise, other programs should be set as the current machining program by "Select program" function.

3.2.3.2 Create new programs



- Select "Edit program" soft key under "Machining" function set to enter the "Edit program" sub-interface;
- Select 『Create』 soft key under the sub-interface and a prompt message "Please enter file name: O temp" in the input box; (Press 「Reset」 to exit the interface)
- Enter a new program name (figure or letter) by MDI keyboard;
- Press 「Enter」 to confirm the new file name to enter the program editing area;
- After editing or modification, press 『Save file』 soft key and a prompt message Saved will be given, then return to the previous interface or other operations.
- Before a file is saved, a prompt message "Save or not" will be given. Press 「Y」 to save the file and 「N」 or 「Reset」 not to save the file.

Note: After a new program is saved under "Machining" function set, it will be loaded as the current machining program automatically.

3.2.3.3 Block operation

"Block operation" function is often used for copy, paste and other operations of multiple program blocks. It defines initial block and final block of programs to define size and position of "block".

This function is easy for program editing, so this soft key is in the submenu of program editing. There are 4 program editing status: Under "Machining" function set, edit and create program "Backstage editing" function; under "Machining" function set, edit current machining program of "Program editing" function; under "Program" function set, create program of "New" function.

Block operation is described as below with copy and paste under the "Program editing" sub-interface as an example

- Press 『Edit program』 soft key to enter the sub-interface;
- Press 『Block』 soft key to enter the block operation sub-interface;
- Press Cursor or PgUp/PgDn to move the cursor to the first block of the blocks to be edited;
- Press 『Block head』 soft key
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the final block of the blocks to be edited;
- Press 『Block end』 soft key to select the big block program;
- Press 『Copy』 soft key;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the paste position
- Press 『Paste』 soft key to complete copy and paste.

3.2.4 "Verify Program" Sub-interface

The "Verify program" sub-interface is mainly used to quickly inspect programs and the machine tool does not run.

Verification program is valid under auto mode and single block mode. Press 『Verify program』 soft key and working mode turns from "Auto" to "Verify"

Press〖Machining〗function key to enter the level 1 menu of "Machining" function set and press 『Verify program』 soft key to enter the interface, as shown below.

3.2.4.1 "Verify program" runs

- Load programs under auto mode;
- Press 『Verify program』 soft key and working mode turns to "Verify";
- Press [Cycle start] to verify programs. (Feedrate override can control verification speed)

3.2.4.2 Exit from "Verify program"

- After the program runs correctly, exit from verification status automatically;
- If verification is not conducted correctly or misoperation occurs, press 「Reset」 to exit from verification status.

3.2.5 "Tool Compensation Setup" Sub-interface

"Tool compensation setup" function is mainly used to set tool offset value, tool wear value, tool nose radius value and tool nose position number in order to realize offset compensation, wear compensation and radius compensation of tool.

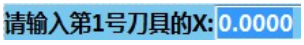
For ease of machining, the system configures "Tool compensation" function under "Machining" set and "Setup" set with the same function and operation. Operation of this function is described as below with "Tool compensation setup" under "Machining" set as an example.

Tool offset value can be inputted manually by MDI or calculated and inputted automatically by『Precutting diameter』and『Precutting length』. Other values should be inputted manually.

Press〖Machining〗function key to enter the level 1 menu of "Machining" function set and press 『Tool compensation setup』 soft key to enter the interface, as shown below.

3.2.5.1 Direct input of tool compensation value

- Press 『Tool compensation setup』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the position where compensation needs to be inputted;
- Press 「Enter」 to confirm, activate input status and the input box gives a prompt message Input a tool and an axis.
- Input correct figures using NC keyboard
- Press 「Enter」 to confirm, the original tool compensation value is replaced by the inputted value, the input box gives a prompt message "Modification succeeds and takes effect form next tool changing or rerun", and exit from input status.



3.2.5.2 Precutting input of tool compensation value

Tool setting is to determine the positional relationship between workpiece coordinate system and machine coordinate system. During tool setting of precutting, measure diameter or length of

workpiece so that the operator knows the position of the tool in the workpiece coordinate system (retract by translation). Then, the operator transmit this message to the system by『Precutting diameter』and『Precutting length』soft keys. The system reads the position of the tool in the machine coordinate system in order to determine the positional relationship between two coordinate systems.

- Press 『Tool compensation setup』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the tool axis where tool offset is set;
- Precut the outer diameter of workpiece and exit from the workpiece along Z axis. Measure diameter of workpiece, such as 9.16mm.
- Press 『Precutting diameter』 soft key (or 「X」) to activate the input box and input the measured value, such as 9.16;
- Press 「Enter」 to confirm input and the system automatically calculates X axis offset of the tool and clears X axis wear of it.
- Precut the end face of workpiece and exit from the workpiece along X axis. Measure the distance from the end face to the zero point of the workpiece, such as 0mm.
- Press 『Precutting length』 soft key (or 「Z」) to activate the input box and input the measured value, such as 0;
- Press 「Enter」 to confirm input and the system automatically calculates Z axis offset of the tool and clears Z axis wear of it.

3.2.5.3 Overall translation of tool offset

Specifications of workpieces are difference. If only coordinates change, the relative position between tools remains unchanged, but all tools areoffset. "Tool post move" can meet this requirements.

- Press 『Tool compensation setup』 soft key in the level 1 menu of machining function set to enter the sub-interface;

- Press 『Tool post move』 to activate the input box
- Input “X offset ␣Space␣ Z offset” (such as 0.1 ␣ 0.1)
- Press 「Enter」 to confirm input and X/Z offset of all tools in the tool offset table increases by the input value relatively.

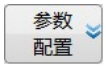
- If format of input data is incorrect (if there is no space key), a prompt message that input data is invalid will be given

3.2.5.4 Exit from input activation state

During tool compensation MDI input, precutting diameter input and precutting length input, due to input errors (such as tool number axis is selected incorrectly) and other reasons, the input activation status must be exited, and compensation value at the current activation must not be changed.

- To abandon current input due to misoperation or input error, press 「Reset」 to exit from the input status and the original tool compensation value remains unchanged.

3.2.6 "Parameter Configuration" Sub-interface



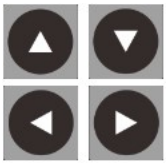
"Parameter configuration" sub-interface is mainly used to set parameters relating to machining user, such as display, protection and machine tool accuracy compensation.

Generally parameter values can be inputted by keys on the panel or backed up and imported. This section introduces direct input by panel keys only. For parameter backup, import and other operations, refer to Chapter 9 Machine Tool Commissioning.

All soft key functions under the "Parameter configuration" sub-interface are limited functions and password should be entered to enable them.

Press〖Machining〗function key to enter the level 1 menu of "Machining" function set and press 『→』 to enter the extension menu. Press 『Parameter setup』 soft key to enter the "User parameter" sub-interface and other sub-interfaces, as shown below.

3.2.6.1 Direct input of parameter configuration value



- Press 『User parameter』, 『Machine parameter』 and 『Axis parameter』 soft keys to enter different sub-interfaces;
- Press 『Enter password』 to activate the input box;
- Enter user password, such as “NC8”;
- Press 「Enter」 to confirm;
- Press 「Cursor」 or 「PgUp/PgDn」 to select the value on the right of parameter name;\
- Press 「Enter」 to activate value input status;
- Input the value to be set such as “1”;
- Press 「Enter」 to confirm and a prompt message "Setup succeeds, save to validate" will be given;
- Press 『Save』 soft key and a prompt message "Save the modified value or not" will be given;
- Press 「Y」 , a prompt message "Save successfully" will be given and parameters will take effect immediately.

3.2.6.2 Direct input of pitch error compensation value

- Press 「Pitch compensation」 soft key to enter the "Compensation information" sub-interface;

- Select compensation type, such as "Unidirectional compensation";
- Move 「Cursor」 to the value setup area of "Starting point", "Compensation interval", "Backlash compensation type", "Number of compensation points", "Initial number of data table" and "Backlash value";
- Press 「Enter」 to activate value input status;
- Input corresponding value in the input box (initial number of data table is often 710000)
- Press 「Enter」 to confirm;
- Press 『Save』 soft key and a prompt message "Parameters have been saved" will be given.

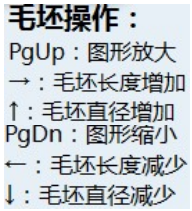
3.2.7 "Graphics Setup" Sub-interface

Select simulation display interface through cyclic switching of "Display switch" soft key in the level 1 menu under machining function set. This interface can display such information as workblank graphics, workpiece zero, tool shape and tool path.

For simulation, set simulated workblank size, zero position and tool type in the "Graphics setup" sub-interface.

Press〖Machining〗function key to enter the level 1 menu of "Machining" function set and press『→』to enter the extension menu. Press『Graphics setup』 soft key to enter the interface, as shown below.

3.2.7.1 Graphics workblank setup and zoom



- Press 『Workblank setup』 soft key under the "Graphics setup" sub-interface and the workblank setup dialog box will pop up;
- Move the cursor to the set values respectively;
- Input all corresponding values;
- Press 「Enter」 to validate input and exit from the dialog box;
- Press 「Reset」 or "Cancel" to abandon inputting values and exit from the dialog box;
- Press 「PgUp/PgDn」 to zoom out or in graphics (a prompt message will be given in the machining information area);
- Press 「Cursor」 to increase or decrease length and diameter of workblank (a prompt message will be given in the machining information area).

Note: 1. "Zero position" is the coordinate value of the program zero in the graphics coordinate system;

2. The origin of the graphics coordinate system is the intersection point between the right end of workblank graphics and the center line;

3. If the program zero is the front end face of the workpiece, the inputted "zero position" is 0.

3.3 "Setup" Function Set Interface and Basic Operation

3.3.1 "Setup" Function Set Interface and Function

"Setup" function set integrates operation functions relating to tool setting. Operations that can be conducted under the function set include tool setting, tool compensation setup*, coordinate setup, tool life management, automatic tool measurement and thread repair.

Level 1 main menu and level 1 extension menu of "Setup" function set are shown below.

Tool compensation setup	Tool post move	Trial cut Diameter Program	Precutting length	Coordinate system	Tool Life	Relative clear	⇨

Tool measurement	Thread Repair

- Tool compensation setup: Tool offset and wear value can be set under the sub-interface of this function, or tool offset can be set through 『Precutting diameter』 and 『Precutting length』 soft keys; tool offset value can be increased or decreased by 『Tool post move』 soft key.

  Function and operation of "Tool compensation setup" under "Setup" function set are the same as those of "Tool compensation setup" under "Machining" function set.

- Tool post move: Press 『Tool post move』 soft key to increase or decrease tool offset under the default interface of the setup function set.

  Function and operation of "Tool post move" under "Setup" default interface are the same as those of "Tool post move" under "Tool compensation setup" sub-interface

- Precutting diameter: Press 『Precutting diameter』 soft key to set X axis offset of tool and clear tool wear under "Tool compensation setup" sub-interface.

  Function and operation of "Precutting diameter" under "Setup" default interface are the same as those of "Precutting diameter" under "Tool compensation setup" sub-interface.

- Precutting length: Press 『Precutting length』 soft key to set Z axis offset of tool and clear tool wear under "Tool compensation setup" sub-interface.

  Function and operation of "Precutting length" under "Setup" default interface are the same as those of "Precutting length" under "Tool compensation setup" sub-interface.

- Coordinate system: This function can be used to set values of the workpiece coordinate system through direct input, current value input and incremental input modes.

  Function and operation of "Coordinate system" under "Setup" function set are the same as those of "Coordinate system" under "Machining" function set.

- Tool life: This function can be used to set tool life management method and strategy.

- Relative clear: Clear relative coordinates X and Z under this interface.

- Tool measurement: Realize automatic measurement of newly installed tools.

- Thread repair: It can perform thread repair processing on the reinstalled threaded workpiece.

Some functions of "Setup" function set are the same as those of "Machining" function set. Functions introduced in "Machining" function set are not introduced in this section.

After startup, press 〖Setup〗 function key to enter the default interface of "Setup" function set, as shown below.

3.3.2 "Coordinate System" Sub-interface

The coordinate value of "External zero offset", "Relative coordinate system", “G54-G59 coordinate system” and "G54.1P1- G54.1P60 coordinate system" can be set under the "Coordinate system" sub-interface (as shown below).

The coordinates of this sub-interface has 3 areas. Area 1 displays "External zero offset" and "Relative coordinate system", area 2 displays "Machine actual" and "Relative actual" coordinate systems and area 3 displays G54-G59 series coordinate systems.

The coordinate value of area 2 cannot be set (the cursor cannot enter this area). Area 1 and area 3 are switched by upper and lower cursors. The coordinate system of current area is selected using left and right cursors or PgUp/PgDn keys.

3.3.2.1 Direct input of coordinate value

This function can be used to input known zero coordinates of the workpiece into the selected workpiece coordinate system.

- Press 『Coordinate system』 soft key under the level 1 menu of "Setup" function set to enter the sub-interface
- Press 「Upper and lower cursors」 to select the coordinate system of area 1 or 3;
- Press 「Left and right cursors」 or 「PgUp/PgDn」 to select the set coordinate system;
- Press 「Enter」 to activate the input box;
- Input the coordinate value of the the workpiece zero in the input box;
- Press 「Enter」 to validate input, then exit from the dialog box.
- To abandon input, press 「Reset」to abandon input and exit from the input box

3.3.2.2 Input of current value

After tool setting is completed and the tool moves to the zero of workpiece coordinates, this function can be used to set the machine tool position in the selected coordinate system.

- Press 『Coordinate system』 soft key under the level 1 menu of "Setup" function set to enter the sub-interface
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to select the coordinate system;
- Press 『Current input』 soft key to activate input box, and a prompt message "Whether to set current position as the workpiece zero?" will be given
- Press 「Y」 to set coordinates of current machine tool as the zero of the selected workpiece;
- Press 「N」 or 「Reset」 to abandon setup and exit from the input box.

3.3.2.3 Input of incremental value

If the tool is worn or the position of coordinate system needs to be adjusted, this function can be used for incremental input of the coordinate zero.

- Press 『Coordinate system』 soft key under the level 1 menu of "Setup" function set to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to select the coordinate system;
- Press 『Incremental input』 to activate the input box;
- Input incremental value of the coordinates in the input box;
- Press 「Enter」 to confirm, and then exit from the input box;
- To abandon input, press 「Reset」to invalidate the input, and exit from the input box

3.3.3 "Tool Life" Sub-interface

Under the "Tool life" sub-interface (as shown below), 5 tool life determination benchmarks can be set by "Setup": Installation times, cutting time, cutting mileage, cutting energy consumption and spindle revolutions. To enable this function, set channel parameter 040130 under “Maintain”--> “Parameter setup” menu and the parameter value of tool life management mode is set as 1.

When the specified value is reached for one of the benchmarks, the system can determine tool life early warning or alarm status accordingly; The weighted sum of the several selected benchmarks can also be used as the basis for determining tool life. The selection of this strategy is selected by the soft key "Alarm Strategy" under this sub-menu

3.3.3.1 Tool life benchmark setup

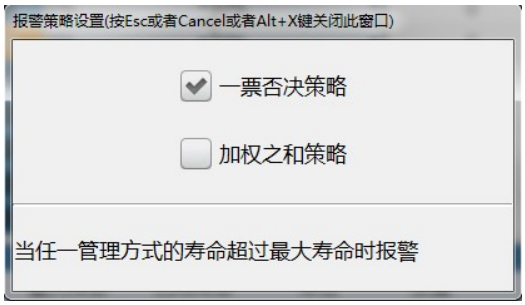
- Press 『Tool life』 soft key under the "setup" interface to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 or 『Find tool number』 soft key to move the cursor to the "Setup" column of the selected tool;

- Press 「Enter」, the life benchmark setup window will pop up (as shown below);
- Select management mode, life benchmark and weight by cursor;
- Press 「Enter」 to activate input;

- Press 「Enter」 to confirm the inputted values;
- Select "Enter" or "Cancel" by cursor;
- Press 「Enter」 to confirm, and exit from the setup window.

3.3.3.2 Tool life alarm strategy setup

- Press 『Tool life』 soft key under the "setup" interface to enter the sub-interface;
- Press 『Alarm strategy』 soft key, the strategy selection window will pop up (as shown below)
- Press 「Cursor」 to select alarm strategy;
- Press 「Enter」 to confirm, then exit from the selection window.



3.3.4 "Tool Measurement Sub-interface"

Under the extension menu of "Setup" function set, press 『Tool measurement』 to enter the "Tool measurement" default sub-interface (as shown below), namely "Measurement parameter" sub-interface.

This function is a limited function and it should be enabled with workshop administrator permission or above. For permission enabling, refer to 3.6.4.

"Tool measurement" function includes three stages: measurement preparation, datum tool calibration and automatic measurement. Besides, overall translation can be realized by "Offset setup" soft key under the sub-interface.

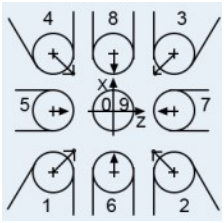
1. Preparation for measurement

Measurement preparation sets automatic measurement speed and tool nose mounting direction through "Measurement parameter" function.

Press『Measurement parameter』soft key under the "Tool measurement" interface to enter the "Measurement parameter" sub-interface (as shown above). The default interface of "Tool measurement" is the "Measurement parameter" sub-interface.

The above area 1 cannot be set in this interface and it is used to view whether tool setter is calibrated. The area 2 interface is used to set "Measurement parameter". Specific steps:

- Press 「Cursor」 to select parameters
- Press 「Enter」 to activate the input box
- Input data
- Press 「Enter」 to confirm. Including



- Length/width of measuring instrument: Length and width value of measurement instrument may not be very accurate, but they have certain influence on the maximum value of offset of tools. When tool offset is greater than 50mm and this function is used for the first time, pay attention to avoid collision.
- Calibrate tool nose direction: Calibrate tool nose direction of datum tool. Tool nose direction number in this section is the mounting direction number of tool nose, which affects feed and retract direction of the datum tool during automatic tool setting (as shown in the left figure).
- Measurement times: Times that the tool touches the measuring instrument during automatic tool setting, no more than 5 times.
- Measurement speed: Speed when the tool touches the measuring instrument for the first time during automatic tool setting
- Trigger speed: The speed of the tool touching the measuring instrument in the process of actual reading the value. The distance is 2mm by default.

2. Tool calibration

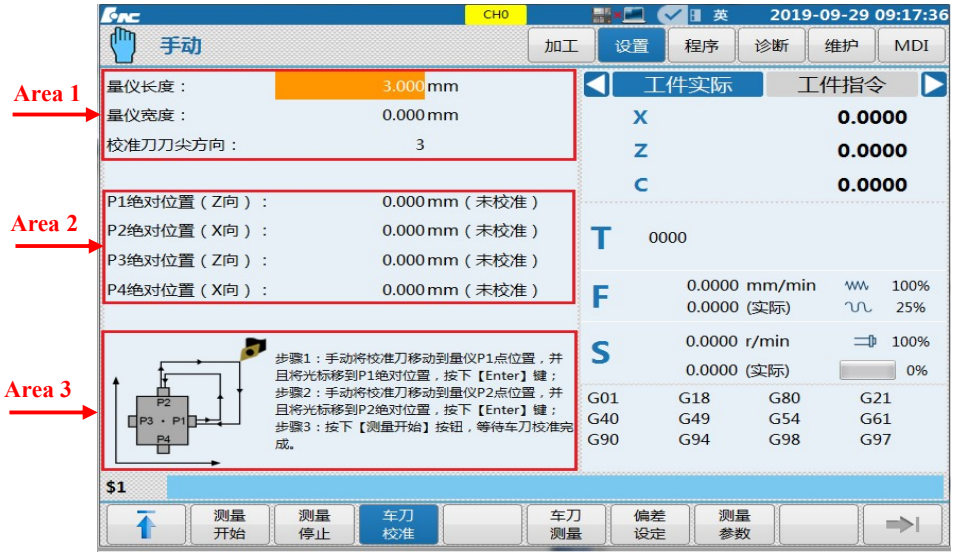
Datum tool calibration function can determine the positional relationship between datum tool and measuring instrument. The tool calibration process includes manual calibration and automatic calibration.

Press 『Lathe tool measurement』 soft key under the "Tool measurement" interface to enter the "Lathe tool calibration" sub-interface (as shown below).

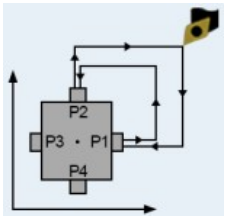
The below area 1 is used to set size of measuring instrument and tool nose direction number of datum tool. If this parameter has been set in "Measurement parameter", the parameter value is displayed accordingly (for ease of check).

The area 2 interface is used for calibration of datum tool. This step is to determine the position of the measuring instrument in the coordinates of machine tool through datum tool.

The area 3 interface displays the position of tool calibration point and tool calibration steps. Specific tool calibration steps are as follows:



1) Manual calibration of "Datum tool"



- Set working mode as 【Jog】
- Press 「Cursor」 to select the calibration position, such as "P1 absolute position";
- Move the datum tool to P1 point (as shown in the left figure);
- Press 「Enter」 to enter and display "P1 absolute position" value. This value is the Z axis coordinate value of the datum tool in the machine coordinate system (namely Z axis

coordinate value of the measuring instrument P1 point in the machine coordinate system).

- Likewise, enter "P2 absolute position", namely X axis value;

  (The tool corresponding to every tool number should be calibrated just at 2 points. e.g.: the tool corresponding to 3# tool nose direction number should be calibrated in P1 and P2 points.)

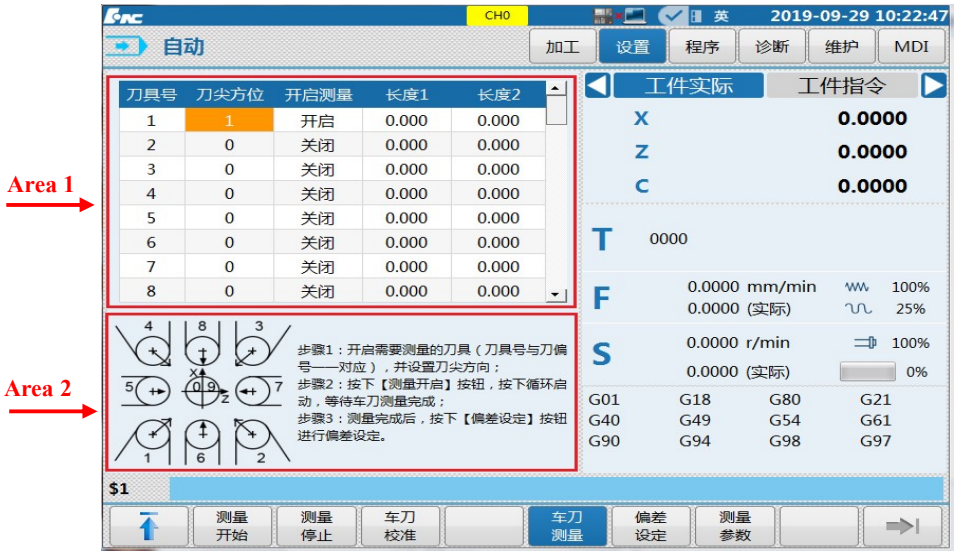
2) Automatic calibration of "datum tool"

- Switch working mode to 【Auto】 or 【Single block】;
- Press 『Start measurement』 soft key;
- Press【Cycle start】and the machine tool completes automatic calibration of datum tool.

3. Automatic measurement

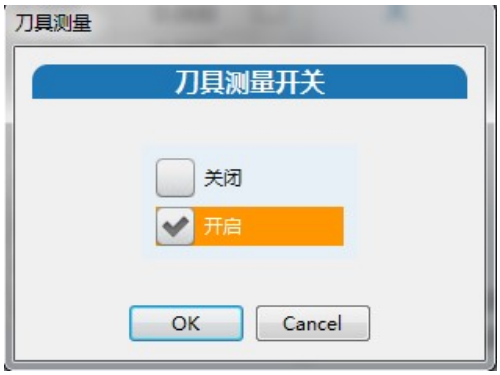
1) Tool selection

Press『Lathe tool measurement』soft key under the "Tool measurement" interface to enter the "Lathe tool measurement" sub-interface (as shown below).



The above area 1 interface is used for parameter setup, area 2 describes the corresponding relationship between tool nose direction number and tool mounting direction. This value is related to the movement direction of the tool during measurement. Specific operation steps of "Lathe tool measurement" function are as follows:

- Move the cursor to "Tool nose direction" column of the area 1 interface to set the mounting direction of the measured tool
- Move the cursor to "Start measurement" column of the area 1 interface to select the tool to be measured
- Press 「Enter」 and the following interface will pop up
- Press 「Cursor」 and 「Enter」to start tool measurement



Note: Tool nose direction number in this section is the tool nose mounting direction number and it affects the movement direction of the tool during automatic measurement. Thus, an incorrect tool nose direction number will result in collision between the tool and the measuring instrument.

2) Automatic tool setting

It is mainly used to accurately measure the positional relationship between all tools. Specific steps are as follows:

- Set working mode as 【Auto】 or 【Single block】;
- Complete the preparation for measurement under the "Tool measurement" interface;
- Press 『Start measurement』 soft key under the "Tool measurement" interface;
- Press 【Cycle start】, then each tool whose measurement function has been enabled will be measured automatically.

Note:

- Press 【Cycle start】 and a prompt message: “Execute G code and wait for user intervention” may appear when the measurement program runs. After the operator confirms that there is no measurement error, press 【Cycle start】 and the measurement program will continue running.
- When tool offset is greater than 50mm, pay attention to avoid

collision between the tool and the measuring instrument.

4. Offset setup

It is mainly used to accurately measure the positional relationship between the tool and the workpiece. Specific steps are as follows:

Press 『Offset setup』 soft key under the "Tool measurement" interface to enter the "Offset setup" sub-interface (as shown below). Area 1 realizes "Offset setup", area 2 realizes datum tool setting and area 3 display the workpiece zero offsets on X axis and Z axis

Specific steps for "Offset setup":

- Press 「Cursor」 and 「Enter」 to select and set "Datum tool number" in the area 1;
- If the selected datum tool has been set correctly and tool offset has been set, set "Whether to use the original tool offset value" as "1" in area 1.
- If offset of the selected datum tool is not set correctly or the original value is not set, set "Whether to use the original tool offset value" as "0" in area 1;
- If "Whether to use the original tool offset value" is set as "0" in area 1, the tool setting of the datum tool should be perform;
- Tool setting of standard tool can be set in area 2 (for tool setting, switch working mode to "Jog"). The tool offset will be updated and can be viewed in the tool offset interface.

Note:

Press 【Start measurement】 before lathe tool calibration and lathe tool measurement. If an emergency occurs in the measurement process,

press 【Feed hold】 to stop the program and 【Stop measurement】 to end the measurement process. If the measurement process cannot be interrupted, press 【Stop measurement】 to finish measurement.

3.4 "Program" Function Set Interface and Basic Operation

3.4.1 "Program" Function Set Interface and Function

"Program" function set mainly integrates management functions of program file and can used to create new programs. Level 1 main menu and level 1 extension menu of soft key function of "Program" function set are shown below.

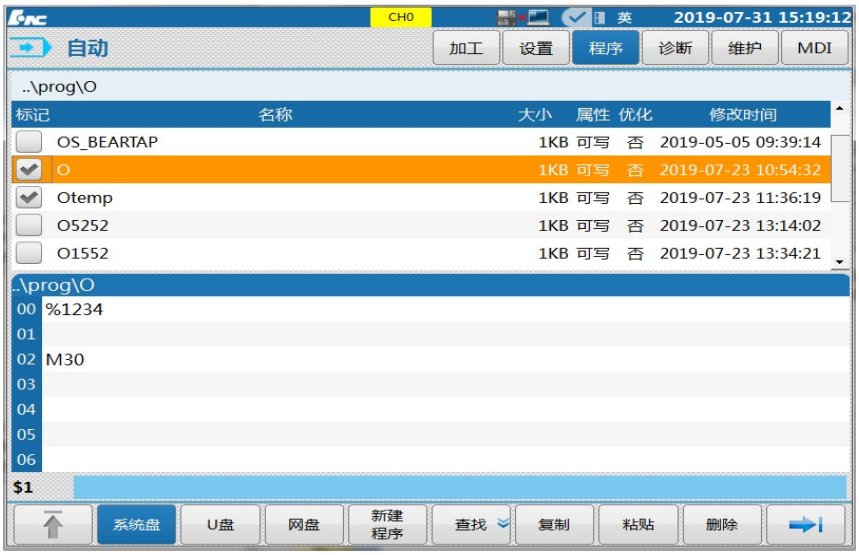
System Disk	USB flash disk	Network disk	User Disk	Find	Set mark	Copy	Paste ⇨
New program	New directory	Delete	Rename	Name sort	Time sort	Writable	read only

- System disk, USB flash disk, network disk: Source disk or target disk of program during program file management.
- New program: Create new programs with the same functions of 『New』 soft key under the 〖Machining〗 function set.
- Find: Search programs in the source disk of program files.
- Copy, paste: Copy programs in the source disk of program files and paste them to the target disk.
- Delete: Delete program files in the source disk of programs.
- Set mark: Mark programs in the source disk of programs in order to copy or paste multiple programs.
- Rename: Rename programs in the source disk of programs.
- Name sort, time sort: Sort programs in the source disk of programs in alphabetical order or modification time order in the program name.
- New directory: Create a new program directory in the target disk of programs.
- Writable, readable: Set program files as writable or readable.
- User disk: This system divides storage card (CF card) into operating

system disk area, CNC system disk area and user disk area, among which user disk is used for backup, storage and other uses and is unrelated to operation of machine tool.

After startup, press 〖Program〗 function key to enter the default interface of "Program" function set, as shown below.

Program can be selected under this interface. Move the cursor to file name of the program to view the first few lines of the program for ease of program identification.



3.4.2 Management of Files in System Disk, USB Flash Disk and Network Disk

3.4.2.1 Management program search

- Select the areas where the program searched may be located under the "Program" default interface, namely 『System disk』, 『USB flash disk』 and 『Network disk』;
- If the program to be searched is in the file directory, press 「Enter」 to open it;
- Press 『Find』 soft key, activate the input box, prompting to input the file to be searched;
- Input a file name to be searched, such as O0011;
- Press 『Enter』 to find the corresponding program;

3.4.2.2 Program copy and paste

- Press 『Find』 or 「Cursor」 and 「PgUp/PgDn」 under the "Program" default interface to select the program to be copied and pasted;
- Press 『Copy』 soft key and the input box will give a prompt message: Copy succeeds;
- Press『System disk』,『USB flash disk』and『Network disk』to select the target areas
- If the program needs to be pasted is in the file directory, select the file directory and press 「Enter」 to open it;
- Press 『Paste』 soft key and the input box will give a prompt message: Paste succeeds;

3.4.2.3 Program deletion

- Press 『Find』 or 「Cursor」 and 「PgUp/PgDn」 under the "Program" default interface to select the program to deleted;
- Press 『Delete』 soft key to delete the program and a prompt message Delete succeeds will be given.

3.4.3 Create New Programs

- Press 『System disk』, 『USB flash disk』 and 『Network disk』under the "Program" default interface to select the areas where a new program is created;
- To create new programs in the file directory, select the file directory and press 「Enter」 to open it;
- Press 『New』 soft key and the dialog box will give a prompt message: Input file name
- Input file name, such as “OHZ1”;
- Press「Enter」to confirm input, the working set switches from "Program" to "Machining" and the interface switches to "Edit program" sub-interface under the "Machining" function set.
- After program editing is completed as stipulated, press『Save』soft key to save programs, prompting Save succeeds.

Note 1: Both "Machining" function set and "Program" function set

have 『New』 function.

Note 2: When a new program is created under the "Machining" function set and working mode is "Auto", "Single block" and "Jog", the new program can be loaded automatically.

Note 3: While creating a new program under the "Program" function set, the interface and the menu will switch to "Machining" function set automatically, but the new program will not be loaded automatically.

3.4.4 Program Rename

- Press 『System disk』, 『USB flash disk』 and 『Network disk』 under the "Program" default interface to select the areas where the program to be renamed is located;
- If the program to be renamed is in the file directory, select the file directory and press 「Enter」 to open it;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the program to be renamed
- Press 『→』 to switch to the extension menu of the "Program" interface;
- Press 『Rename』 soft key and the dialog box will give a prompt message: Input a new file name;
- Input a new file name in the dialog box, such as “OHZ2”;
- Press 「Enter」 to confirm input and the original program is renamed.

3.4.5 Program Mark Setup

- Press『System disk』,『USB flash disk』and『Network disk』 under the "Program" default interface to select the areas where the directory or program to be marked is located;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the program to be marked;
- Press 『→』 under the "Program" default interface to switch to extension menu page of the "Program" interface;
- Press『Set mark』, then the program name is prefixed with “√”.

3.4.6 Programs Are Sorted By Name And Time

- Press『System disk』,『USB flash disk』 and『Network disk』 under the "Program" default interface to select the areas where the program to be marked is located;
- If the program to be marked is in the file directory, select the file directory and press 「Enter」 to open it;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the program area to be sorted;
- Press『→』 under the "Program" default interface to switch to extension menu page of the "Program" interface;
- Press『Name sort』 or『Time sort』 soft key to sort programs of this area as required.

3.4.7 Program Write/Read Setup

- Press 『System disk』, 『USB flash disk』 and 『Network disk』 under the "Program" default interface to select the areas where the program to be set is located;
- If the program to be set is in the file directory, select the file directory and press 「Enter」 to open it;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the program whose attribute is set;
- Press『→』 under the "Program" default interface to switch to extension menu page of the "Program" interface;
- Press 『Write』 or 『Read only』 to set program attributes.

3.4.8 Create a New Directory

- Press『System disk』,『USB flash disk』 and『Network disk』 under the "Program" default interface to select the areas where the new directory is to be created;
- To create new directories in the file directory, select the file directory and press 「Enter」 to open it;
- Press 「Cursor」 to move the cursor to the areas where a new directory is to be created;
- Press『→』 under the "Program" default interface to switch to

extension menu page of the "Program" interface;

- Press 『New directory』 soft key, then the dialog box gives a prompt message: "Please input a directory name";
- Input a directory name such as HCNC, a new directory is created.

3.5 "Diagnosis" Function Set Interface and Basic Operation

3.5.1 "Diagnosis" Function Set Interface and Function

"Diagnosis" function set mainly integrates such functions as fault alarm, fault diagnosis and machine debugging. Level 1 main menu and level 1 extension menu of soft key function of "Diagnosis" function set are shown below.

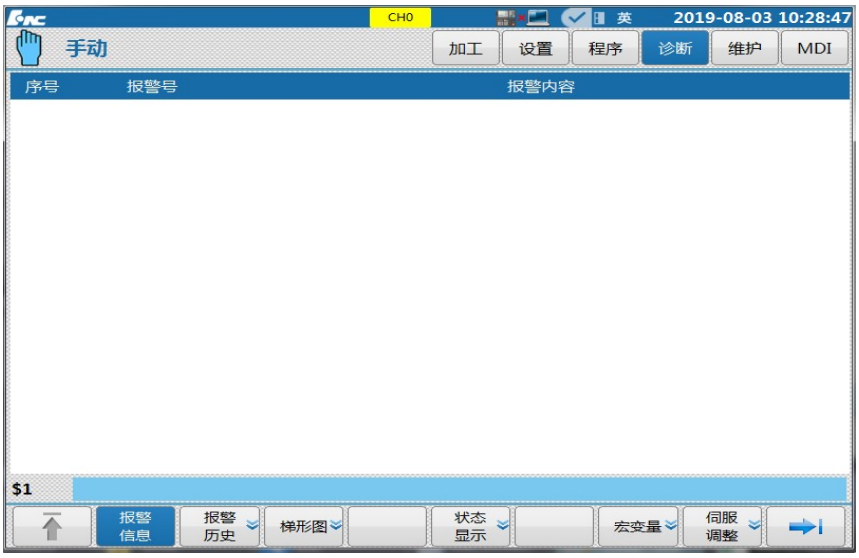
Alarm information	Alarm History	Ladder	Display status	Axis monitoring	Macro variable	Servo adjustment	⇨
Self-check	Log	Fault video	Screw load	QR code			

- Alarm message: Display current alarm message
- Alarm history: Save information of recent alarms and export historical alarms to USB flash disk and system disk using submenu "History export" soft key under the menu "Alarm management".
- Ladder: Track or record PLC signal in the way of period or trigger; configure operation, inspection and valuation method of PLC through the ladder option.
- Status display: Display and view the status of all registers.
- Axis monitoring: Display and view status of axis, including coordinate position, pulse value, breakpoint position, compensation value and load current of axis.
- Macro-variable: Display and view the value of macro-variables.
- Servo adjustment: Adjust servo parameters to realize commissioning of optimal speed, position, roundness, tapping, noise and gantry synchronization of machine tool based on the sampling result.
- Self-check: Record relevant health indexes during long-term operation of machine tool in order to predict health condition of machine tool.

- Log: Record the operation process completed by the system in the process of machine tool operation.
- Fault video: Record fault related data 10s before the fault occurs. The related data can be preset as position, speed and current of each axis.
- Screw load: Record operation frequency of all areas of screw during long-term operation of machine tool in order to determine the screw wear and other statuses.
- QR code: This function can be used to acquire data of machine tool such as machining status, alarm history and fault diagnosis, and upload them to cloud data center to realize full life circle management of machine tool.

Intelligent functions and interfaces of "Servo adjustment" and "Self-check" are not introduced in this chapter. For specific operations, refer to the subsequent chapters.

After startup, press 〖Diagnosis〗 function key to enter the default interface of "Diagnosis" function set, as shown below.



3.5.2 Alarm History Export

➢ Press『Alarm history』soft key under the "Diagnosis" function set default interface;

➢ Press 『Alarm history』 soft key under the 『Alarm history』 sub-interface;

➢ Press 『Alarm management』 soft key;

- Select 『System disk』, 『USB flash disk』 and 『User disk』 soft keys;

- Press 『History export』 soft key to export corresponding information to the selected disk.

3.5.3 Status Record Export

- Press『Alarm history』soft key under the "Diagnosis" function set default interface;

- Press 『Status record』 soft key under the "Alarm history" sub-interface;
- Press 『Status management』 soft key;

- Select 『System disk』, 『USB flash disk』 and 『User disk』 soft keys;

- Press 『Status export』 soft key to export corresponding information to the selected disk.

3.5.4 "Ladder " Sub-interface

This function is used for modification, monitoring and editing of system PLC. Press 『Ladder』 soft key under the "Diagnosis" interface to enter the ladder diagram sub-interface, as shown below.

Note: This interface requires the workshop administrator or higher permission to enter, please refer to section 3.6.4 for specific content.

3.5.4.1 Ladder monitoring

This function is used for monitoring system PLC

- Press 『Ladder』 soft key under the "Diagnosis" default interface;
- Press 『Ladder monitoring』 soft key to enter the ladder diagram monitoring sub-interface (as shown below);

3.5.4.2 Ladder diagram editing

This function is used for PLC modification

- Press 『Ladder』 soft key under the "Diagnosis" interface to enter the ladder diagram sub-interface;
- Press 『Ladder editing』 soft key to enter the ladder diagram editing interface (as shown below);
- Add elements of ladder diagram using straight line, normally open, normally closed, logic output, inverse output, vertical line, function module and double coil (see main menu and extension menu in the below figure)
- Search and edit ladder diagram using program list, delete vertical line, find, delete element, edit network and list editing (see main menu and extension menu in the below figure)
- "Update modification" is to save modifications of ladder diagram (see extension menu in the below figure)
- "Abandon modification" is to abandon modifications of ladder

diagram (see extension menu in the below figure)

3.5.4.3 Ladder diagram information

梯形图

This function is mainly used to edit name and corresponding point location of register and view related information of ladder diagram.

- Press 『Ladder』 soft key under the "Diagnosis" default interface;

梯图信息

- Press 『Ladder information』 soft key to enter the sub-interface and the sub-menu (as shown below).

1. Ladder diagram title

梯图标题

Record number of lines, period and program name in the related information interface of ladder diagram

2. Symbol table

符号表

Chinese name setup table of X, Y, F, R and G registers is mainly used to view point location information.

3. IO comparison table

IO对照表

Version 2.4 uses I register and Q register to replace output of X register and Y register in PLC. In this case, if different machine

tools are configured, only a set of PLC is required to be used in order to reduce problems of PLC. When equipped with different machine tools, the change of IO points does not need to change the PLC, only the IO comparison table needs to be modified, and the different XY registers correspond to the IQ registers with the IQ points remaining unchanged in the PLC.

4. K parameter

Its function is the same as that of P parameter, but the point setting of 0 or 1 can be modified directly in this interface.

5. Timer and counter

Used to monitor status of timer and counter in programs.

6. Alarm setup

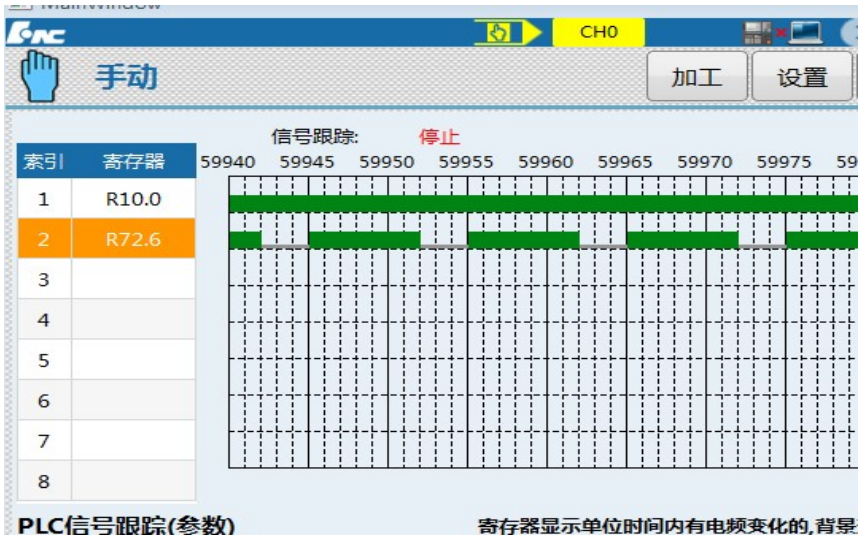
Used to view alarms corresponding to G3010.0-G3025.15 (alarm registers) in PLC.

3.5.4.4 Ladder diagram signal tracking

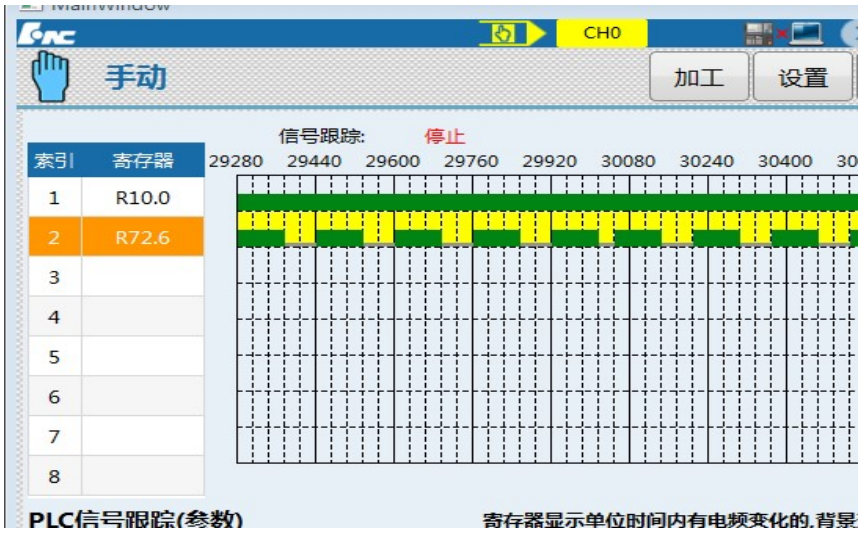
This function can be used to track change of register values through sampling.

- Press 『Ladder』 soft key under the "Diagnosis" default interface;
- Press 『Signal tracking』 soft key to enter the PLC signal tracking sub-interface (as shown below);

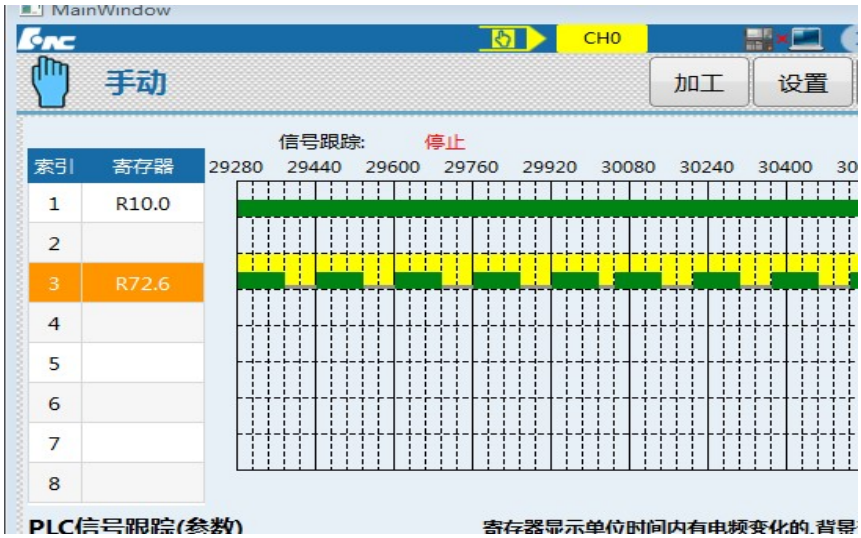
For example, sample "Running Allowed R10.0" and a blinking point R72.6, fill in the two registers in the corresponding boxes, click to start sampling, the changes of the two points will be displayed in histogram.



Press Zoom out and Zoom in to control the interval time between two sampling points. e.g.: For example, the figure below is a reduction of the sample in the above figure.



Press Move up and Move down to move the sample where the cursor is located to move up or down for a line. e.g.: Move R72.6 down for a line.



On the parameter interface, users can control the enabling conditions of sampling function. In the above example, if the start condition is changed to trigger start, and the trigger condition is changed to X483.7, then there will be no sampling start after clicking start, but pressing the X483.7 button once will start sampling. If the stop is changed to trigger stop with the stop condition

X483.6, then the sampling can be stopped by pressing the stop button or X483.6 button.

3.5.5 Display Of Register Status And Macro-variable Value

This function can be used to display and view status of registers and value of macro-variable for ease of fault analysis.

- Press 〖Diagnosis〗 function key to enter the default interface of function set;
- Press 『Status display』 or 『Macro』 soft key to display statuses of X, Y, F, G, R and B registers or values of macro-variable addresses;

3.6 "Maintain" Function Set Interface and Basic Operation

3.6.1 "Maintain" Function Set Interface and Function

Integrate such functions as parameter configuration, system commissioning and machine tool information under "Maintain" function set. The level 1 main menu and level 1 extension menu of soft key function of "Maintain" function set are shown below.

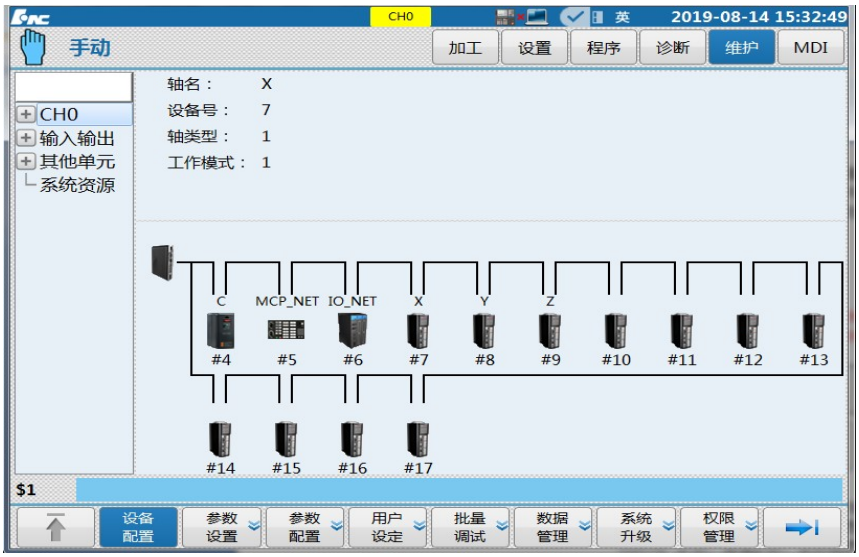
Device configuration	Parameter setup	Parameter configuration	User setup	Batch	Data management	System upgrade	Permission management ⇨

Spatial compensation	Time setup	Process package	Machine information	System information	Register

- Device configuration: View the number of drive, I/O, panel and other hardware devices and connection sequence of bus;
- Parameter setup: This interface includes all system parameters, under which NC parameter, machine user parameter, channel parameter, coordinate axis parameter, error compensation parameter, device interface parameter and data table parameter can be set;
  - ✧ NC parameters: Common parameters of the CNC system (such as interpolation period and resolution).
  - ✧ Machine user parameters: Common parameters relating to machine tools and users (type of measuring instrument and maximum number of channels).
  - ✧ Channel parameters: Common parameters of CNC system in each channel (such as small line segment parameter)
  - ✧ Axis parameter: Related parameters of logical axis (electronic gear ratio and acceleration/deceleration time constant, etc.)
  - ✧ Error compensation parameter: Set related parameters of error compensation of logical axis (such as backlash compensation type of axis 0)
  - ✧ Device interface: Interface parameters relating to connection between the physical device and the system (such as device type and device ID)
  - ✧ Data table: the data table storing the corresponding compensation values of error compensation parameters.
- Parameter configuration: Under this interface, there are commonly used parameters for users, and they are classified according to application types, so that users can operate during setting; the parameters under this interface include user parameter, machine parameter, axis parameter, pitch compensation, IO device, local spindle device, bus axis device and function parameter;
- User setup: The settings relating to user application. This interface includes display seting, P parameter, M code, PLC switch, communication setup, personalized setup and closed-loop switching;
- Batch: Loading and backup of PLC, parameter, canned cycle, G code and other files;
- Data management: Loading and backup of various types of data;

- System upgrade: System upgrade and backup;
- Permission management: To set the administrators of different permissions. Different permissions have certain influence on the structure of interfaces and menus;
- Spatial compensation: Space error compensation setup;
- Time setup: System time setup;
- Process package: Load or back up the technology package file;
- Machine information: Edit or display machine tool information;
- System information: Display information of this system;
- Registration: Display registration code and related information of the machine tool;

After startup, press 〖Maintain〗 function key to enter the default interface of "Maintain" function set, as shown below.



3.6.2 Parameter Setup

1) Parameter selection

- Press 〖Device configuration〗, 〖Parameter configuration〗 and 〖Parameter setup〗 to select parameter set;
- Press 「Left and right cursors」 to move the cursor and select the parameter classification column or the parameter setup column

2) Parameter input activation

- When the cursor moves to parameter classification column, press

「Enter」 to open categories of the selected parameter

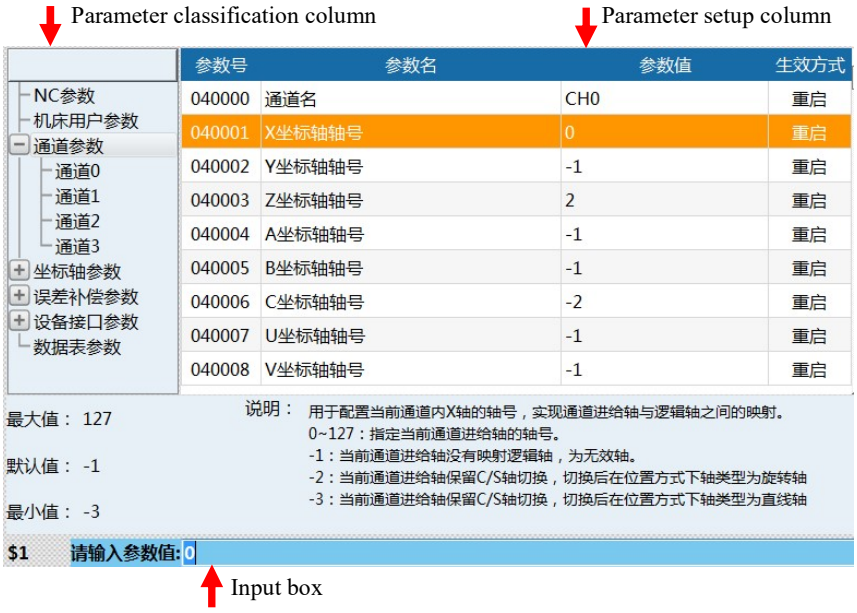
- When the cursor moves to the parameter setup column, press 「Enter」 to activate "Input box"

3) Parameter input

- After input box is activated and corresponding values are inputted, press 「Enter」 to confirm input.

4) Exit the input

- After activating the input box, users need to give up the input, press "Reset" to exit the input and keep the original value.



3.6.3 Parameter Validation and Operation

There are 5 effective types of parameters in this system: effective immediately, effective after save, effective after reset, effective after restart, and solidified (which cannot be set). The specific operations are as follows:

1) Effective immediately

- After parameters are input into the input box, press 「Enter」 to confirm. The parameters are input successfully and take effect immediately.

2) Effective after save

- After inputting the parameters in the input box, press the "Enter" key to confirm the input, and the input box will prompt "Set successfully, and it will take effect after saving";

- Press the "Save" or "↑" soft key, the input box prompts "Save the modification or not? (Y/N)";
- Press 「Y」 or 「Enter」, the input box prompts "saved successfully" . The parameters take effect;
- Press 「N」 to give up saving and restore the original value.

3) Effective after reset

- Press 「Enter」 to confirm the input after inputting the parameters, the input box prompts "Set successfully, it will take effect after reset";
- Press 『Save』 or 『↑』 soft key, the input box prompts "Save the modification or not? (Y/N)"；
- Press 「Y」 or 「Enter」, the input box prompts "Saved successfully, please press Reset";
- Press 「Reset」, the input box prompts "Reset successfully", and parameters take effect;
- Press 「N」 to give up saving and restore the original value.

4) Effective after restart

- Press 「Enter」 to confirm the input after inputting parameter in the input box. The input box prompts "Set successfully, restart the system to take effect after save";
- Press 『Save』 or 『↑』 soft key and the input box gives a prompt message "Whether to save the modified value? (Y/N)"；
- Press 「Y」 or 「Enter」, the input box prompts "Save successfully, please power off and restart";
- After the controller is powered off, restart it to validate parameters.

Note:

- Parameter setting and modification are restricted operations, so when setting and modifying parameters, passwords with corresponding permission should be entered.

3.6.4 Management Permission Classification and Switching

In response to the different application requirements of CNC machine tools, the system has 5 types of operating permissions, which are operator, workshop administrator, machine tool manufacturer, CNC manufacturer and system administrator. Management functions of various permissions are roughly as follows:

"System administrator": Development, test and customer service. It has system software maintenance permission.

"CNC manufacturer": Product manufacturing and quality inspection . It has permissions of system upgrade, system parameter setup, PLC program editing and limited-time shutdown setup.

"Machine tool manufacturer": Machine tool commissioning. It has permissions of partial system parameter modification, error compensation data entry and shutdown timer setting.

"Workshop administrator": Machining commissioning. It has permissions of partial user parameter modification, editing parts program and editing tool compensation data.

"Operator": Machining operation. It has permissions of editing tool compensation data and selecting program.

Operation function \ Permission type		System administrator	CNC manufacturer	Machine tool manufacturer	Workshop administrator	Operator
Parameter classification	User parameter	Yes	Yes	Yes	No	No
	Machine parameter	Yes	Yes	Yes	No	No
	Axis parameter	Yes	Yes	Yes	No	No
	Pitch compensation	Yes	Yes	Yes	No	No
	I/O device	Yes	Yes	No	No	No
	Local spindle device	Yes	Yes	No	No	No
	Bus axis device	Yes	Yes	No	No	No
	Function parameter	Yes	Yes	Yes	No	No
System upgrade		Yes	Yes	No	No	No
Permission management		Yes	Yes	Yes	Yes	Yes
Batch commissioning		Yes	Yes	Yes	No	No
User setup (except the display setup)		Yes	Yes	Yes	Yes	No
Data management		Yes	Yes	Yes	Yes	No
Spatial compensation		Yes	Yes	Yes	No	No
Time setup		Yes	Yes	Yes	No	No
Technology package		Yes	Yes	Yes	No	No
Registration		Yes	Yes	Yes	No	No

Alarm history		Yes	Yes	Yes	Yes	No
Ladder	Ladder diagram monitoring	Yes	Yes	Yes	Yes	No
	Ladder diagram editing	Yes	Yes	Yes	No	No
	Ladder diagram information	Yes	Yes	Yes	No	No
	Signal tracking	Yes	Yes	Yes	No	No
Status display		Yes	Yes	Yes	Yes	No
Macro-variable		Yes	Yes	Yes	Yes	No
User macro		Yes	Yes	Yes	Yes	No
Servo adjustment		Yes	Yes	Yes	No	No
Log clearing		Yes	No	No	No	No
Program Function Set	Select program, find, sort	Yes	Yes	Yes	Yes	No
	Verify, any line	Yes	Yes	Yes	Yes	Yes
	Relative clear	Yes	Yes	Yes	Yes	Yes
	Display mode, path switching	Yes	Yes	Yes	Yes	Yes
	Machining statistics	Yes	Yes	Yes	Yes	Yes
	Machining optimization	Yes	Yes	Yes	Yes	No
	Program editing	Yes	Yes	Yes	Yes	No
	Read only, writable attribute setting	Yes	Yes	Yes	Yes	No
	User macro	Yes	Yes	Yes	Yes	No
Setup Function Set	Tool compensation	Yes	Yes	Yes	Yes	Yes
	Coordinate system	Yes	Yes	Yes	Yes	Yes
	Workpiece measurement	Yes	Yes	Yes	Yes	Yes
	Magazine, tool life setup	Yes	Yes	Yes	Yes	No
	Broken tool detection	Yes	Yes	Yes	Yes	No
	Automatic tool setting	Yes	Yes	Yes	Yes	No
Machining Function Set	Parameter configuration	Yes	Yes	Yes	No	No
	User macro	Yes	Yes	Yes	Yes	No

	Edit program	Yes	Yes	Yes	Yes	No
	Select program (except "System disk")	Yes	Yes	Yes	Yes	No
	Other operations	Yes	Yes	Yes	Yes	No

This system can set permission of "Operator" or "Workshop administrator" as default permission through parameter 000359 in NC parameter table, and other permissions can be switched by entering a password after startup. The permission switching operation is as below:

- Press 〖Maintain〗 function key to enter the default interface of the "Maintain" function set
- Press 〖Permission management〗 soft key to enter the "Permission management" sub-interface
- Press 〖Logout〗 soft key to exit the current permission;
- Press 〖Left and right cursors〗 to select the required permission;
- Press〖Login〗soft key to activate the input box, and a prompt message "Please enter a login password" will be given;
- Enter a password in the input box, such as “HCNC1”;
- Press 「Enter」 to confirm the input. Then the permission is modified successfully.

3.6.5 System Upgrade

The "System upgrade" function is available for CNC manufacturer only. Thus, permissions should be set under the "Maintain" function set after startup (The permission is not saved after shutdown).

After permission is set, press 『System upgrade』 soft key under "Maintain" function set to enter the "System upgrade" sub-interface (as shown below)

- Press 『Window switch』 soft key to select the "Upgrade selection" window
- Press 「Left and right cursors」 to select the required items; (BTF is to upgrade all items)
- Press 「Enter」 to confirm the selection;
- For backup, select "Backup" (the default backup target disk is the user disk);
- Press『Window switch』soft key to select the upgrade patch file source selection window below the interface (the default upgrade source disk is USB flash disk);
- Press 「Up and down cursors」 to select the upgrade package file (the upgrade package file name must be suffixed with . BTF);
- Press 「Enter」 to confirm and start upgrade

Note:

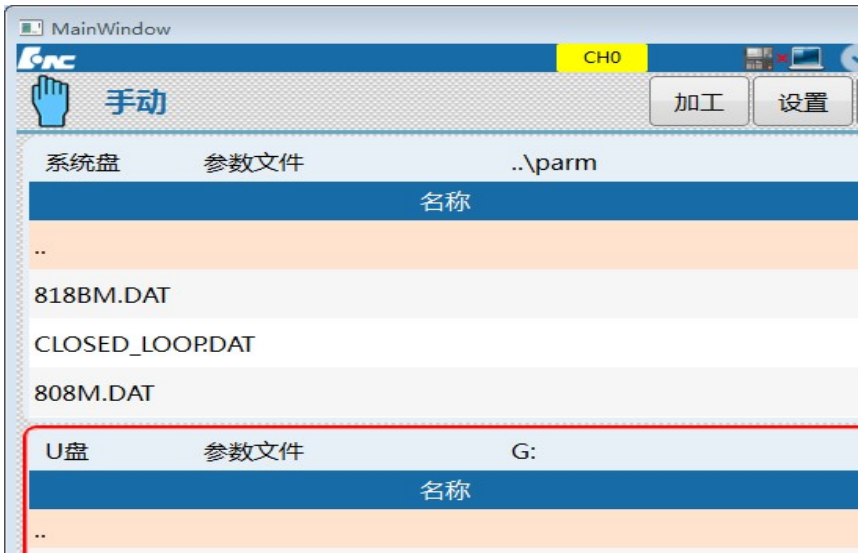
- The upgrade must be conducted by technical personnel of HCNC company.
- The system upgrade must be conducted in "Emergency stop" status
- The upgrade package file must be .BTF file, and the file name is suffixed with .BTF
- The default system backup disk is the user disk, pay attention to the size of user disk during backup

3.6.6 Data Management

On the data management interface, user can load/back up parameters, PLC, canned cycle, log, compensation, oscilloscope and other files. This section takes the loading/backup system parameter file as an example to illustrate, the steps of loading and backing up other files (except for error compensation files) are the same.

Press 『Data management』 soft key under the main menu of the "Maintain" function set to enter the data management sub-interface (as shown below).

- Press「Cursor」to select the type of data to be loaded or backed up;
- Press 「Enter」 to confirm the selection;
- Press 『USB flash disk』 or 『User disk』, and select 『Load』 or 『Backup』 to enter the load or backup sub-interface (as shown below)



- To load data in USB flash disk or user disk to system disk, press 『Window switch』 soft key to move the red box to USB flash disk or user disk to the lower part of the above figure;
  - Press 「Enter」 to open the file directory, and press 「Cursor」 to select the data file to be loaded;
  - Press 『Load』 soft key, the input box gives a prompt message "Load the selected file or not?"
  - Press 「Y」 to load data file;
  - Press 「N」 or 「Reset」 to give up loading data file.
- To back up data in system disk in USB flash disk or user disk, press 『Window switch』 soft key to move the red box to USB flash disk or user disk above the above figure;
  - Press 「Enter」 to open the file directory and press 「Cursor」 to select the data file to be backed up;
  - Press 『Backup』 soft key and the input box gives a prompt message "Back up the selected file or not?"
  - Press 「Y」 to back up the data file;
  - Press 「N」 or 「Reset」 to give up backing up data file.

Note Power failure is strictly prohibited during backup or loading.

3.6.7 User Setup

用户
设定

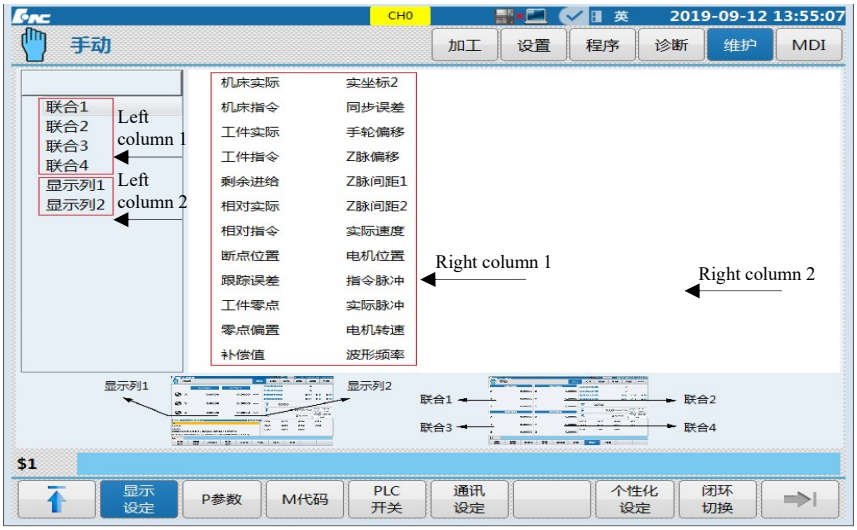
User setup is used for common display and control switches of PLC. Users can set different functions according to different needs.

Press 『User setup』 soft key under the main menu of the "Maintain" function set to enter the "User setup" submenu (as shown below).

3.6.7.1 Display setup

As mentioned in the introduction of the processing interface, the soft key "Display Switching" under the "Processing" function set can switch between "Large-character coordinates + program", "joint coordinates", "graphics + program", and "program". "Big character coordinate + program" and "Joint coordinate" interfaces can be set here.

Press 『User setup』 soft key under the "User setup" sub-interface to enter the "Display setup" submenu (as shown below).



When the cursor moves to the left column 1 in the above figure (joint coordinates 1-4), content of the right column 1 is displayed on the right of the above figure, and the display content in the "Joint coordinate" interface can be selected from the right column 1 (as shown in the below left figure).

Joint coordinate display

Big character coordinate + program display

When the cursor moves to the left column 2 in the above figure (display columns 1-2), contents of the right column 2 is displayed on the right of the above figure, and the display content in the "Big character coordinate + program" interface can be selected from the right column 2 (as shown in the above right figure)

3.6.7.2 Set "P parameter"

P parameter is a parameter relating to machining and system operation. Each PLC subprogram switch and the PLC determination are set by P parameter which can be set under the "User setup" interface. Whereas this parameter has a great effect on safety of machine tool, please be sure to modify it under the guidance of related authorized person.

P parameter corresponds to the parameters after the machine user parameter 010300, and corresponds to the same memory address together with these user parameters. The『P parameter』 soft key is a shortcut key.

Press 『P parameter』 soft key under the "User setup" sub-interface to enter the "P parameter" sub-interface (as shown below).

索引	参数号	参数名
1	010329	润滑时间(单位:s)
2	010330	停润滑时间(单位:s)
3	010331	刀架最大工位数
4	010332	刀架选择
5	010333	主轴波动检测时间(ms)
6	010334	X轴过载检测延时(ms)
7	010335	Z轴过载检测延时(ms)
8	010336	螺纹90度退尾开关(1:打开)
9	010337	跟随式攻丝(0:关闭,1:打开)

It should be noted that when a function is enabled through setting P parameters, not only should PLC switch be set, but also other relevant parameters and functions should be set. e.g.: When the lathe spindle indexing function is enabled, not only should the ON/OFF parameters of lathe spindle indexing be set, but also the indexing parameters of lathe spindle should be set; otherwise, the spindle cannot realize the indexing function.

3.6.7.3 Set "M code"

M code table is mainly used to set the determinations such as M code validation, validation sequence when M codes and G00 are in the same line and whether M codes are identified in any line scanning.

Press 『M code』 soft key under the "User setup" sub-interface to enter the "M code" submenu (as shown below).

M指令名称	组号	类型	任意行扫描	G00同
M00	0	后置	否	否
M01	1	后置	否	否
M02	2	后置	否	否
M03	3	前置	否	否
M04	3	前置	否	否
M05	3	后置	否	否
M06	6	同步	否	否
M07	7	同步	是	否
M08	8	同步	是	否
M09	9	同步	否	否
M10	10	后置	否	否

Under the "M code" sub-interface, M codes have four setups: type (pre or post), whether any line is scanned, G00 synchronization, and spindle control. Where, setup of M3, M4 and M5 codes is cured and cannot be modified. Specific definitions are as follows:

M3 defines CW rotation of spindle 1;

M4 defines CCW rotation of spindle 1;

M5 defines stop of spindle 1;

a) Pre or post

When M code and G code are in the same line, M code is set to the pre to validate M code earlier; M code is set to the post to validate M code later. To validate M code and G code simultaneously, they should be set to be synchronous.

b) Any line scanning mode

Scanning mode means that before the system runs a program, all modal information before any line is scanned, and the specified line inherits the information including coordinates, tool and compensation of the previous statement.

Scanning mode is divided into two mode: scanning without Z axis return and scanning with Z axis return, which are controlled by parameter 040113. It will take a long time for scanning a very large program. The subprogram scanning is also supported.

Non-scanning mode means that the system does not calculate the modal information before any line, and it is the default status.

Note: 040113 parameter value

0: Non-scanning mode (default)

1: Scanning without Z axis return

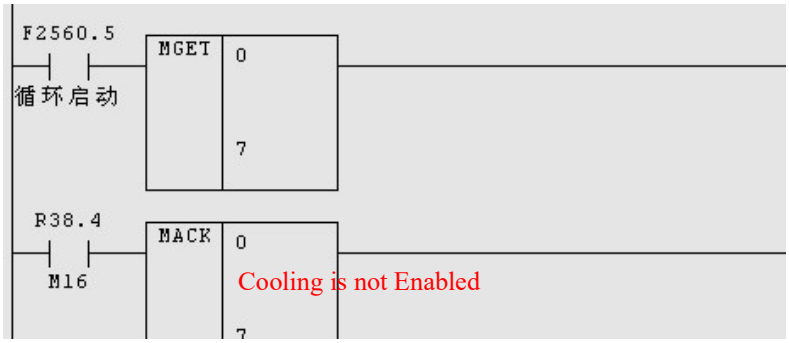
2: Scanning with Z axis return

c) G00 synchronization

It should be noted that synchronization is the pre of post setting when M command and G command are on the same line; but it is a special situation that M command and G00 are on the same line, and it should be set separately.

Take M7 as an example. When M7 and G00 are in the same line, it is the synchronous M code. When M7 and G01 are in the same line, it is the pre-M-code.

Note: For the M codes with the G00 synchronization attribute turned on, an immediate response is required in the PLC. In order to realize the waiting function when in the same line with G01, PLC needs to be modified, as shown in the figure below



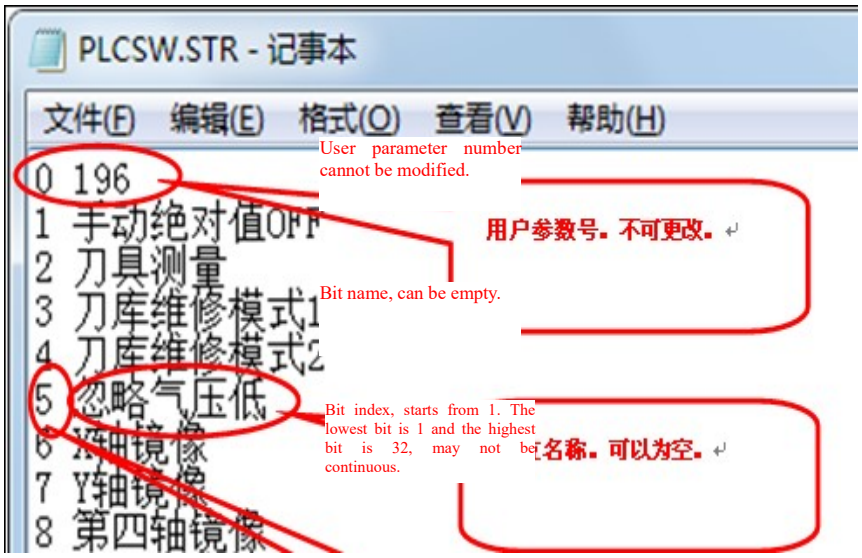
d) Spindle control

When automatic C/S switching of spindle is performed (position mode/speed mode), the switched axis name and axis number as well as start and stop of axis need to be defined by M code in the PLC program. The M code defined in PLC can be marked through this setting.

Note: When there are multiple spindles, be sure to mark M codes for the newly added spindle control.

3.6.7.4 Set "PLC switch"

PLC switch setup function is to decompose the designated user P parameter to 32 bits and each bit is a configurable PLC switch. PLC switch is configured by PLCSW.STR configuration file under parm directory. The file format of PLCSW.STR is shown below:



After PLCSW.STR file is made, select “PLC switch file” to import PLCSW.STR file into the system through "**Data management**". As shown below

Press “**Maintain→ User setup→ PLC switch**” menu to enter the PLC switch function interface, and operate the designated P parameter by bit, as shown below:

Effective setup: Protect “ON” and “OFF” menus, the “ON” and “OFF” menus can be operated only when the "Effective setup" menu is pressed;

ON: Set the bit of focus as ON;

OFF: Set the bit of focus as OFF;

The setup result is saved in the designated user P parameter. As shown below, user P parameter 196 in this example is designated by PLCSW.STR file.

3.6.7.5 Communication setup

This function can realize the communication between the upper computer of the CNC system and the computer and the communication of shared disk of regional machine tools.

Note: The network can be connected only after NC parameter 000050 【Whether to enable network】 is enabled

a) Communication between the upper computer of the CNC system and the computer

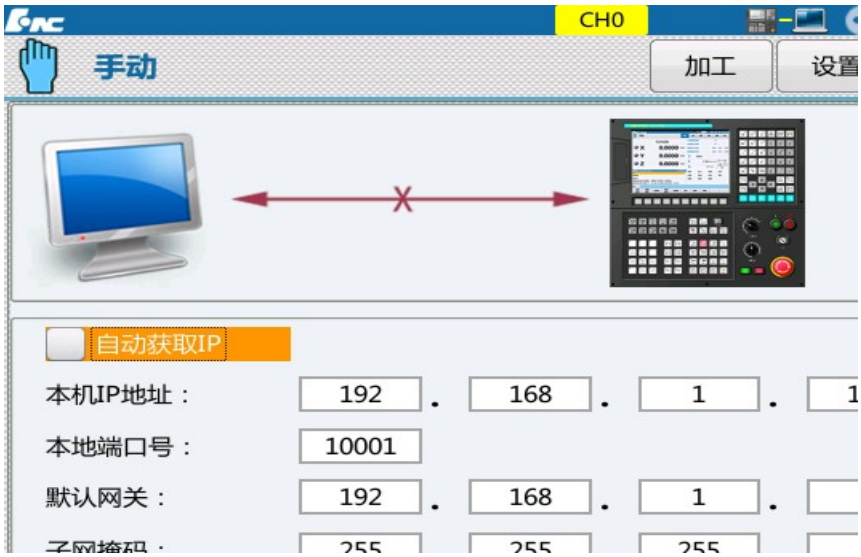
For communication between the upper computer of the CNC system and the computer, ip of the computer and ip of the CNC system needs to be set to the same network segment,

ip of the CNC system is 192.168.1.113 by default. ipv4 of the computer is set to 192.168.1.XXX at the time of connection, and the default gateway and subnet mask of the computer are consistent with those of the CNC system. Specific steps:

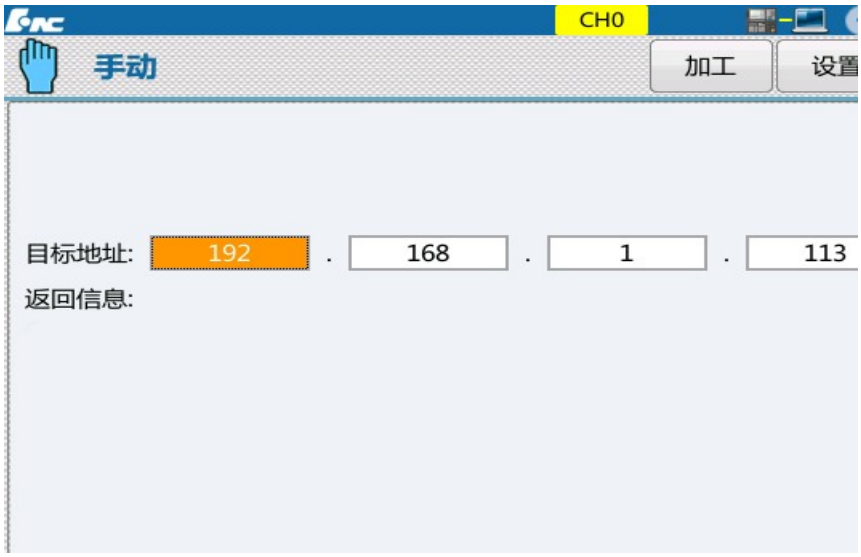
➢ Press 『User setup』 soft key under "Maintain" function set to enter the

sub-interface;

- Press 『Communication setup』 soft key to enter the communication setup sub-interface;
- Press 『Local』 soft key to enter the "Local" connection sub-interface (as shown below);
- Move the cursor to where "Local IP address" is set and set default IP address of the system, namely 192.168.1.113.
- Move the cursor to "Local port number", "Default gateway" and "Subnet mask" of the system, and set them as "Local port number", "Default gateway" and "Subnet mask" of the computer to be connected.
- Connect Internet accesses of the computer and system IPC using network cable. It should be noted that it cannot be connected to M3 of system IPC or EtherCAT internet access.



- Users can PING system on computer or PING computer on system. The PING interface of this system is shown in the figure.

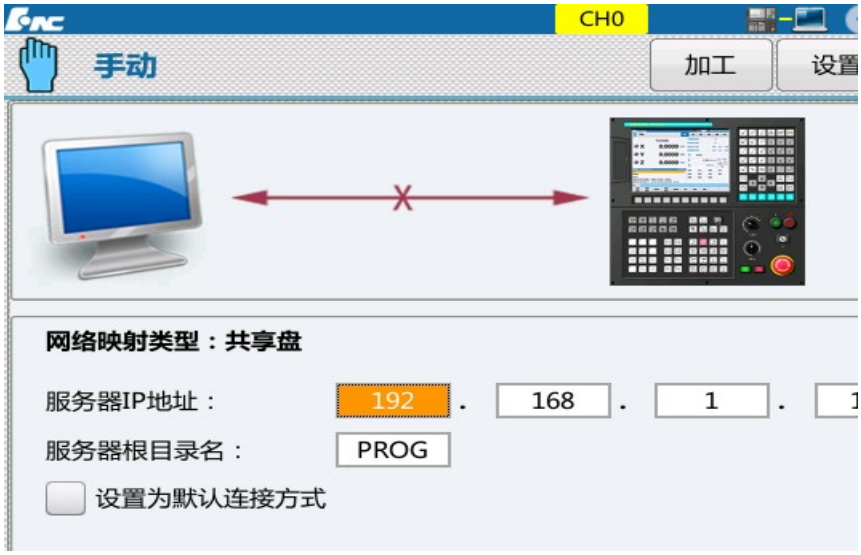


- While PING computer in the system, fill ip address of the computer in PING interface and click on 『PING start』 soft key;
- To terminate PING, press 「Reset」.

Note: Network cable must be connected well and Internet access must be selected correctly.

b) Shared disk communication

If all machine tools in the machine tool factory are networked, all machine tools can share codes and various configuration files in the shared disk. When the machine tools are connected to the shared disk, open the shared disk interface, as shown below:



- Press 『User setup』 soft key under "Maintain" function set to enter the sub-interface;

- Press 『Communication setup』 soft key to enter the communication setup sub-interface;
- Press 『Shared disk』 soft key to enter the "Shared disk" connection sub-interface (as shown above);
- Move the cursor to where "Local IP address" is set and fill IP address of shared disk of machine tool manufacturer.

c) FTP

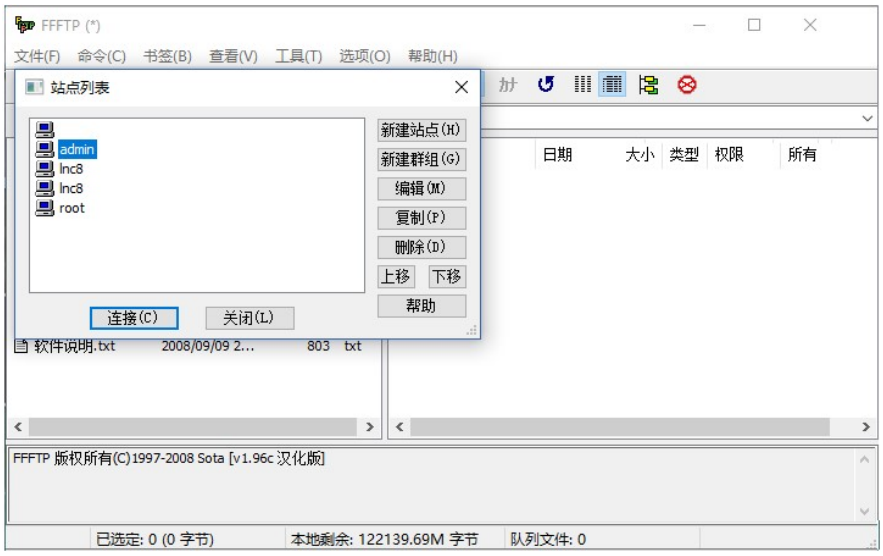
FTP

FTP is a kind of connection software used to copy data from the computer to the system or from the system to the computer.

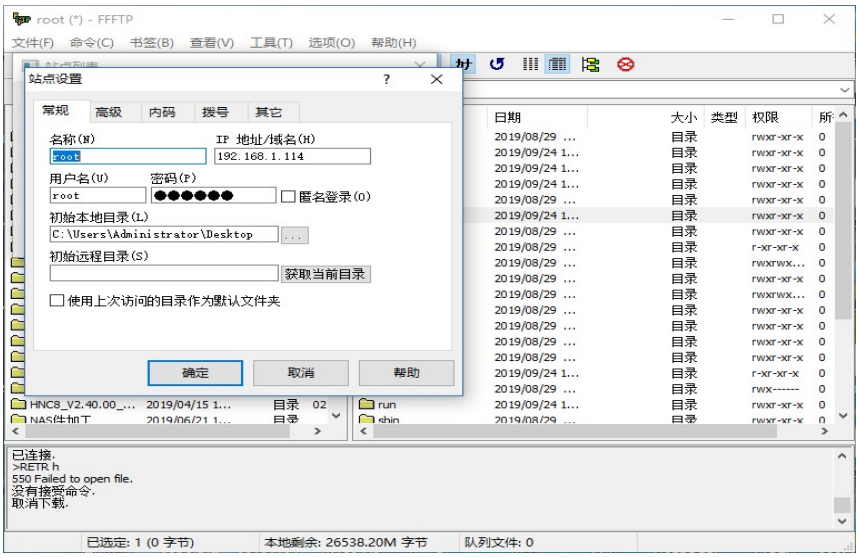
Before FTP is used, ensure smooth network according to the previous section.

Then, install the software FFFTP on computer (download from the official website of HCNC company)

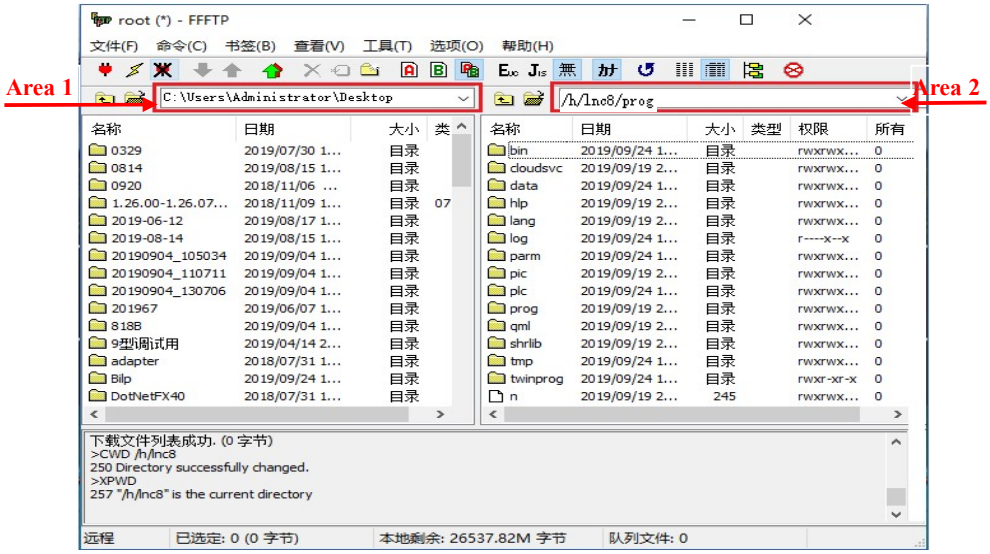
After ping the system and the computer, click on FFFTP icon and the interface is displayed as below:



Users can select root login from the above interface station list. If there is no required station, build a new station (such as root), and the station setup interface is shown below:



- ip address is set to the ip of the system;
- Select and set the user name (such as root);
- Login password is set as 111111;
- Click on Enter to log in to the system, and the login interface is shown below.



The transmission path of the system can be set in the above interface area 2, and the path is often written in /h/lnc8/prog. The computer path can be set in area 1.

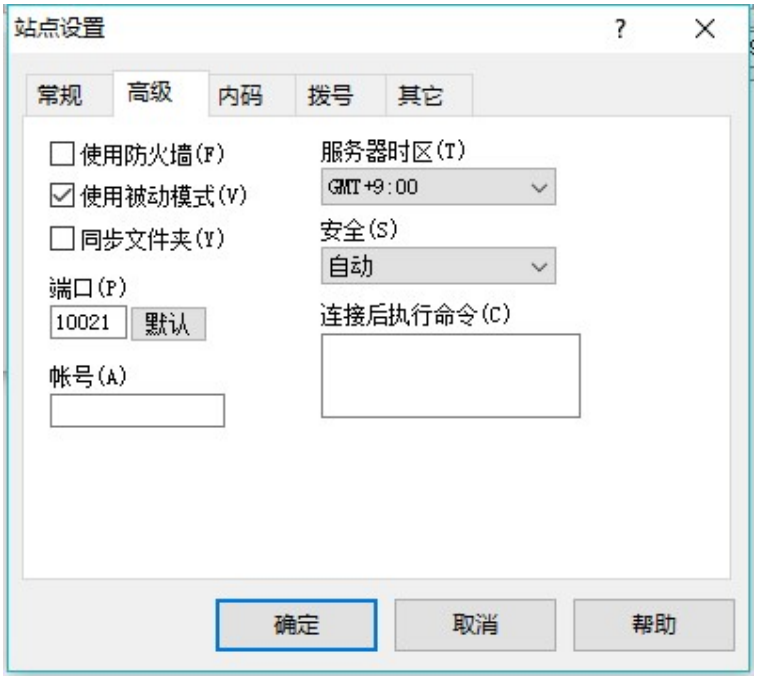
After a path is designated, drag the file using mouse to transmit files between the system and the computer.

Note: If ffftp cannot be connected, check the following setups

1. Whether the parameter 00050is enabled

2. Whether the computer and the system are connected by ping

3. Whether port number is correct, including port number 10001 in the CNC system interface and port number under the advanced menu of ffftp.



d) plc online commissioning

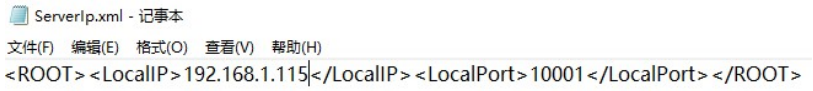
Online commissioning of PLC refers to monitoring and modifying the system PLC on computer. If it is inconvenient to modify and commission PLC on system, this function can be used. The following conditions must be met in use.

- The system and the computer must be connected.
- A suitable adapter is needed,

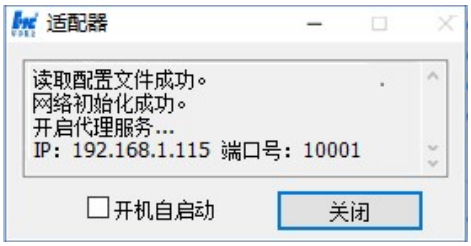
The setting steps of HCNC adaption software are as follows (it should be noted that 1.26 and 2.XX series adapters are different).

(1) Turn on the adapter and find ServerWindowD.exe and ServerIp.xml

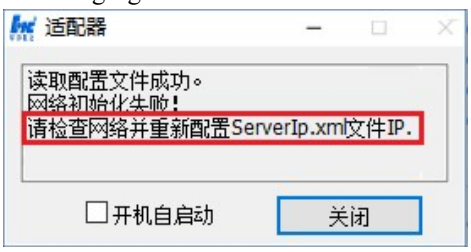
(2) Open Serverlp.xml and replace LocalIP with ip of local computer, which is 192.168.1.115 in this example. Port number must be consistent with that on the system. Save after editing.



(3) Open ServerWindowD.exe and the normal effect is shown below

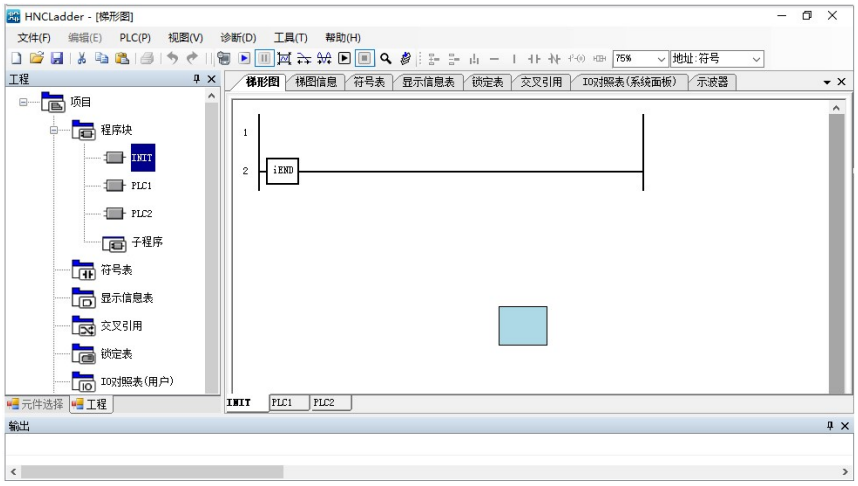


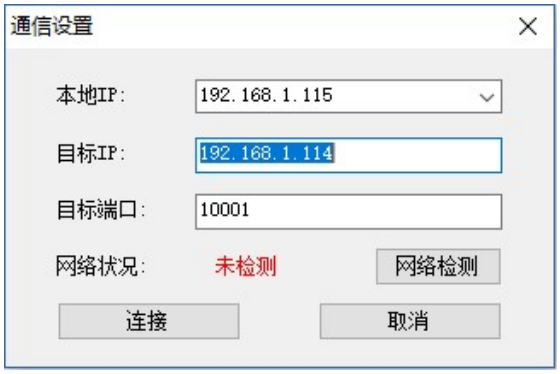
If Serverlp.xml is not modified, an alarm will be given as shown in the following figure



(4) Find online commissioning in the second page of ladder diagram---ladder diagram information menu under the system diagnosis interface, and click on it, the system will give a prompt message that PLC is being commissioned online.

(5) Open ladder diagram editing tools of version V2.0, and click on Network setup under the tool menu





Correctly fill in ip of the local computer and the system, then click on Connect

(6) Click on "Start program monitoring" under the diagnosis menu, the plc commissioning software will load the system system automatically, as shown below.

3.6.7.6 Personalized setup

Through personalized setup: Language setup, the functions including language setup, resolution setup, and skin setup can be realized. Press 『User setup』 soft key under "Maintain" function set and press 『Personalized setup』 soft key to enter the personalized setup sub-interface, as shown below. The modification of personalized setup takes effect after system restart.

1. Language setup

This function can be used to modify system interface language among Chinese, English and Russian.

2. Resolution setup

This function can be used to modify resolution to 800*600 or 1024*768

Note: If BIOS is set to 1024*768, the system supports two resolutions. If BIOS is set as 800*600, software supports 800*600 only (BIOS is set as 1024*768 by default).

3. Skin setup

Black and blue skins are supported.

3.6.8 Technology Package Setup

This function is used to back up process-related optimal parameters, record them in XML file, and export them to other machine tools.

Press 『Technology package』 soft key under the "Maintain" function set to enter the "Technology package" sub-interface, as shown below.

"Technology package" of this system is shown below:

- Backup

  Users select an XML file and click on Backup to back up parameter values corresponding to no (parameter id) into the XML file.

- Load

  Users select an XML file and click on Load to load values in val of the XML file to parameters corresponding to no of the system.

- Rename

  Users select an XML file and click on Rename, and the system will prefix the file name with “CB_”. If “.XML” is not added, the system will suffix the file with “.XML” automatically

- Copy, paste

  Users can copy, paste and delete XML file among different disks (except network disk).

Note:

- The directory of technology package is parm and the technology file can be renamed.
- Naming rule for XML file: CB_*.XML. “CB”及“XML” are in capital form and the format is shown below:

```
<?xml version="1.0" encoding="GB2312"?>
<CRAFTBAG version="1.0">
    <item type="parm" no="000029" val="0"/>
    <item type="parm" no="000032" val="20000"/>
    <item type="parm" no="000067" val="60"/>
    <item type="parm" no="000069" val="0"/>
    <item type="parm" no="000077" val="0"/>
    <item type="parm" no="010103" val="0x0"/>
    <item type="parm" no="040087" val="0.0000"/>
    <item type="parm" no="040088" val="0"/>
    <item type="parm" no="040107" val="1"/>
    <item type="parm" no="040089" val="0.0000"/>
    <item type="parm" no="040158" val="0"/>
    <item type="parm" no="040199" val="0.0000"/>
    <item type="parm" no="040216" val="0.0000"/>
    <item type="parm" no="040334" val="0"/>
    <item type="parm" no="302154" val="0"/>
</CRAFTBAG>
```

version is 1.0;

encoding is GB2312;

Start with “CRAFTBAG” in capital form and attribute version is 1.0;

- type is the type. It is lowercase “parm” if it is a parameter;

no is the parameter number;

val is the parameter value;

4 Power-on, Power-off, Safety Protection, Emergency Stop

This chapter mainly introduces power-on, power-off, emergency stop, reset, and overtravel release of machine tool and CNC device.

4.1 Power-on

Operation name	Power-on	Working mode	Emergency stop
Basic requirements	(1) Check whether the machine tool status is normal; (2) Check whether the power voltage is consistent with the requirements; (3) check whether the connection is correct and secure.		

SN	Operation steps	Button	Description
1	Press [Emergency stop]		● Safety protection
2	Turn on [Air switch of machine tool]		● Power on the machine tool
3	Press [System power-on]		● Power on the system
4	Release [Emergency stop]		● Rotate right to release [Emergency stop] button ● System reset

Note: After power-on, check whether the indicator light on the panel is normal, and release emergency stop button.

4.2 Power-off

Operation name	Power-off	Working mode	Emergency stop
Basic requirements	(1) Stop operation of machine tool; (2) Disable auxiliary function.		

SN	Operation steps	Button	Description
1	Press [Emergency stop]		● Safety protection

2	Press [System power-off]		● System power-off
3	Turn off [Air switch] of the machine tool		● Power off the machine tool

Note: If users power it off and then power it on again, users must keep it off for more than 20 seconds.

4.3 Overtravel Protection and Release

4.3.1 Over Travel Protection

There is a limit switch at each end of the travel of servo axis, which is used to prevent from damaging servo axis due to collision. When the servo axis touches the travel limit switch, the hardware overtravel protection will occur. When the hardware overtravel protection of an axis occurs (the indicator light of "**Overtravel release**" lights up), the system regards its status as the emergency stop and the machine tool stops operation.

This system also can set software overtravel protection through parameters 100006, 100007, 101006, 101007, 102006 and 102007. That is, when the machine tool runs beyond the parameter setting range, the machine tool gives an alarm and stops operation.

4.3.2 Hardware Overtravel Release

Operation name	Overtravel release	Working mode	Jog, handwheel
Basic requirements	(1) If an axis of the machine tool exceeds the travel, all axes must not move and the system gives an alarm.		

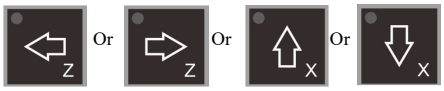
SN	Operation steps	Key	Description
1	Press【Jog】or【Handwheel】		● Set effective working mode
2	Press [Overtravel release] and [Axis feed]		● Press [Overtravel release] and [Axis feed] simultaneously ● Select [Axis feed] in the reverse direction of overtravel axis

Note:

- Under jog (handwheel) mode, enable the axis to exit the overtravel status in the reverse direction;
- While the machine tool is moving to exit overtravel status, please be sure to pay attention to movement direction and movement speed in order to avoid collision;
- If "Overtravel release" key is released, "Error" in the operating status bar changes to "Normal ", which means the normal work is restored and operation can be resumed.

4.3.3 Overtravel Release

Operation name	Overtravel release	Working mode	Jog, handwheel
Basic requirements	(1) In case of overtravel of an axis of the machine tool, all axes must not move and a prompt message will be given.		

SN	Operation steps	Key	Description
1	Press 【Jog】 or 【Handwheel】		● Set the effective working mode
2	Press [Axis feed]		● Press [Axis feed] in the reverse direction of the overtravel axis
3	Press 「Reset」		● Clear alarm

4.4 Emergency Stop

4.4.1 Feed Hold

Press [Feed hold] button when the machine tool runs the program automatically to suspend the machining program. But it cannot be stopped immediately while a threading program is being executed.

4.4.2 Reset

When the system is in the alarm state, the coordinate axis moves abnormally, the output is abnormal or the input needs to exit, user can press the "reset" button to make the system in the reset state. The system "reset" status is as follows:

- All axes stop running (except during threading);
- M and S function output is invalid;
- Stop automatic operation and hold the modal function.

Besides, validation of some parameters is "Effective after reset". Press Reset to validate these parameters after they are input and saved.

4.4.3 Emergency Stop

During the running of machine tool, in case of danger or an emergency, after pressing **the "Emergency stop"** button, the CNC system enters emergency stop status, and servo feed and spindle rotation stop immediately (the feed drive power supply in the control cabinet is cut off); release the "**Emergency stop**" button (rotate right this button) , and the system enters reset status.

Before emergency stop is released, confirm whether fault cause has been eliminated. After emergency stop is released, return all feed axes to the reference point in order to ensure coordinate correctness of each axis.

Note:

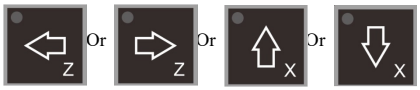
- Press **"Emergency stop"** button to reduce electric shock of device before power-on and power-off.

5 Manual Operation and Speed Override

5.1 Manual Reference Point Return

The precondition for controlling movement of the machine tool is to establish the machine tool coordinate system. For this purpose, return all axes of the machine tool to the reference point after the system is powered on and reset. The methods are as follows:

Operation name	Manual reference point return	Working mode	Reference point return

Basic requirements	With the reference point as the boundary, ensure the machine feed axis stops in the opposite direction stipulated by parameter "Reference point return direction"		
SN	Operation steps	Key	Description
1	Press 【Reference point return】		● Set effective working mode
2	Press [Axis feed]		● Specify the [Axis feed] button of reference point return

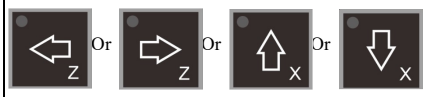
Note:

- When the absolute encoder motor is used for the machine tool, the system needs not return to the reference point;
- While returning to the reference point, the [Axis feed] key is determined based on “Reference point return direction” parameters (100011, 101011 and 102011).
- Press axis direction selection keys (X, Y and Z) simultaneously to return axes (X, Y and Z) to return to the reference point at the same time;
- After all axes return to the reference point, as long as the servo drive does not give an alarm during operation, other alarms need not return to the reference point (including pressing the emergency stop button);
- When zero pulse of the motor and mounting position of the travel switch are too small, reference point return may be inaccurate, and there is often a distance roughly equal to 1 screw pitch. At this time, it is necessary to move the travel switch for a certain distance.

5.2 Move Coordinate Axis by Manual Feed

In this mode, movement of the coordinate axis can be controlled continuously. Generally it is used for machining of simple parts. Press 【Jog】 working mode key, [Axis feed] key and [Feedrate override] key on the control panel and MPG to move coordinate axis of the machine tool manually.

Operation name	Move coordinate axis by manual feed	Working mode	Jog
Basic requirements	(1) The need for continuous movement of machine tool		

SN	Operation steps	Key	Description
1	Press 【Jog】		● Set the effective working mode
2	Select [Feedrate override]		● The product of default speed and feedrate override
3	Press [Axis feed]		● If the key is released, feed stops

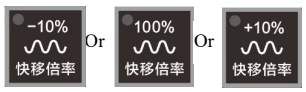
Note:

- Set the default speed by "Slow speed jog speed" parameter (X: 100032, Y: 101032, Z: 102032) in coordinate axis parameters;
- Press each [Axis feed] key, the indicator light lights up and the corresponding machine tool axis moves continuously. Release it, the indicator light lights off, and the machine tool stops moving;
- Press multiple [Axis feed] keys simultaneously in jog mode, the corresponding axes move continuously.

5.3 Rapidly Move Coordinate Axis Manually

In this mode, the axis can be moved rapidly and manually. Press 【Jog】 working mode, [Rapid traverse override] and [Rapid traverse]+[Axis feed] on the control panel and MPG to complete this operation.

Operation name	Rapidly move coordinate axis in Jog mode	Working mode	Jog
Basic requirements	(1) The need for rapid movement of machine tool		

SN	Operation steps	Key	Description
1	Press 【Jog】		● Set effective working mode
2	Select [Rapid traverse override]		● The product of default speed and rapid traverse magnification
3	Press [Rapid traverse] and [Axis feed]		● Press [Rapid traverse] and [Axis feed] simultaneously ● If the key is released, feed

			stops

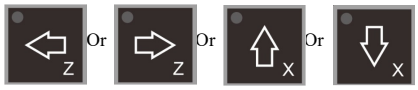
Note:

- Set the default speed by “High speed jog speed” parameter (X: 100033, Z: 102033) in coordinate axis parameter;
- Based on 100% of the default speed, increase and decrease rapid traverse override rate as per 10%;
- Under other modes than "Jog" mode, the [Rapid travers] key is invalid.

5.4 Incremental Feed of Coordinate Axis

In this mode, the coordinate axis can be moved quantitatively by pulse mode. It is often used for tool setting or magazine commissioning and other operations to control the accurate positioning of machine tool.

Press【Incremental】working mode, [Override] and [Axis feed] keys on the control panel an MPG to move coordinate axis of the machine tool with incremental feed.

Operation name	Incremental feed of coordinate axis		Working mode	Incremental
Basic requirements	(1) For manual accurate positioning			
SN	Operation steps	Key		Description
1	Press【Incremental】			● Set effective working mode
2	Press [Override]			● The product of magnification and 0.001mm
3	Press [Axis feed]			● Press once to move an incremental value to corresponding axis in the corresponding direction.

Note:

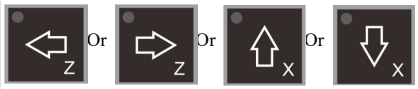
- Each time users press the [axis feed] key once, the axis moves a fixed incremental distance, and then press it again after releasing it, the axis moves a fixed incremental distance. Pressing and holding it will not generate the movement;
- The incremental distance is the product of 0.001mm and magnification;
- HNC-808Di-TU system has three magnifications: ×1, ×10 and ×100;

- [Magnification] keys are interlocked keys, that is, after one of them is valid, the others cannot be valid.

5.5 Move Coordinate Axis by Handwheel

In this mode, the axis can be moved continuously and quantitatively. It is often used for tool setting or magazine commissioning and other operations to control accurate positioning of machine tool.

Press 【Handwheel】 working mode, [Override] and [Axis feed] on the control panel and MPG to move coordinate axis of the machine tool by handwheel feed.

Operation name	Move coordinate axis by handwheel		Working mode	Handwheel
Basic requirements	(1) The need for continuous accurate movement of machine tool			
SN	Operation steps	Key		Description
1	Press 【Handwheel】			● Set effective working mode
2	Press [Override]			● The product of magnification and 0.001mm
3	Press [Axis feed]			● It moves an incremental value to the corresponding axis in the corresponding direction as the handwheel rotates one graduation.
4	Rotate [Handwheel]			● Continuously and accurately move the machine tool

Note:

- When the handwheel rotates, the movement distance per graduation is the product of 0.001mm and magnification;
- HNC-808Di-TU system has three magnifications: ×1, ×10 and ×100;
- [Magnification] keys are interlocked keys, that is, after one of them is valid, the others cannot be valid.
- The handwheel should rotate at the speed of no greater than 5r/s. If the handwheel rotates too fast, the movement distance is not equal to the pulse count of handwheel, or the feed axis cannot stop

immediately when the handwheel stops rotating. The former is by default for HNC-808Di-TU system.

5.6 Manual Spindle Control

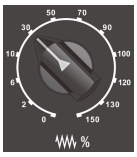
SN	Operation name	Start operation	Terminate operation	Description	Effective working mode
1	Spindle rotation CW	Press [Spindle rotation CW] key 	Press [Spindle stop] or [Reset] key 	1. [Spindle rotation CW], [Spindle rotation CCW] and [Spindle stop] are interlocked; 2. While spindle control needs to be changed for auto operation, switch to jog mode and then switch back to auto mode.	Handwheel, incremental, jog
2	Spindle rotation CCW	Press [] key 	Press Spindle rotation CCW [Spindle stop] or [Reset] 		
3	Spindle stop	Press [Spindle stop]	Press [Reset]		
4	CW rotation/stop of power head	Press [A] 	Press [A] 	1. Press [A] to control CW rotation or stop of power head.	
5	CCW rotation or stop of power head	Press [B] 	Press [B] 	1. Press [A] to control CCW rotation or stop of power head.	
6	Spindle speed override	Press [Spindle override] 		1. Based on 100% of the default speed, increase or decrease rapid traverse override rate as per 10%; 2. Override range: 0%-150%.	Handwheel, incremental, jog, auto, MDI

5.7 Other Manual Operations

SN	Operation name	Start operation	Stop operation	Description	Effective working mode
1	Machine center feed	Press [Machine center feed] 	Press this key again	Press this key to cyclically control advancing and stop of the machine center. The stop is the default.	Jog, handwheel, incremental
2	Machine center jog	Press [Machine center jog] 	Stop after jogging once	Press this key once, and the machine center can advance a fixed distance.	
3	Machine center retract	Press [Machine center retract] 	Press this key again	Press this key to cyclically control retracting and stop of the machine tool. The stop is the default.	
4	Clamping jaw ON/OFF	Press [Clamping jaw] 	Press this key again	Press this key to cyclically control ON or OFF of clamping jaw. It is OFF by default.	
5	Magazine rotation CW	Press [Magazine rotation CW] 	Stop after rotating a cutter position	Press this key once, and the magazine rotates a tool position clockwise.	
6	Lighting ON/OFF	Press [Lighting] 	Press this key again	Press this key to cyclically control ON or OFF of light. It is OFF by default.	Jog, handwheel, incremental, auto, MDI, single block
7	Lubrication ON/OFF	Press [Lubrication] 	Press this key again	Press this key to cyclically control ON or OFF of lubricating pump. It is OFF by default.	
8	Chip removal rotation CW	Press [Chip removal rotation CW] 	Press this key again	Press this key to control CW rotation or stop of chip removal motor cyclically. It is OFF by default.	
9	Cooling ON/OFF	Press [Cooling] 	Press this key again	Press this key to control start and stop of cooling pump cyclically. It is the stop by default.	

5.8 Speed Override

5.8.1 Feedrate Override



In auto mode or MDI operation mode, when the feedrate programmed with **F** command is too high or too low, rotate the feedrate override to adjust the programmed feedrate. The override range: 0%-120%.

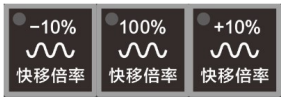
If the feedrate is not specified in the program in auto mode , and "Machining preparation" function is not enabled, the system runs as per the set value of channel parameter "Default feedrate" 040030. If "Machining preparation" function is enabled, the system runs as per the machining preparation speed, and feedrate override in the operational process is valid.

In the manual continuous feed mode, the override can be used to adjust the manual feedrate, and the system runs at the set value of axis parameter "Slow speed jog speed" 100032 (X axis), 101032 (Y axis) and 102032 (Z axis).

Note:

- When feed override is set to 0, the feed override rate is 0, and the rapid traverse override rate changes to 0 temporarily. If the feed override rate is changed to non-zero, the rapid traverse override rate recovers to the original value;
- During thread machining, the feed override is invalid, and the system runs at the original speed;

5.8.2 Rapid Traverse Speed Override



When a program runs in auto or MDI mode, G00 speed of X, Y and Z axes is set by "Maximum rapid traverse speed" parameters 100034, 101034 and 102034;

Under jog rapid traverse mode, manual rapid traverse speed can be regulated by "Rapid traverse override". Manual rapid traverse speed of X, Y and Z axes is set by "High speed jog speed" parameters 100033, 101033 and 102033;

The manual rapid traverse, G00 rapid traverse, and the rapid traverse block in the canned cycle, G28 and G29 can be increased and decreased by "Rapid traverse override". The override range is 0%-100%.

Note:

- As requested by some users, the rapid traverse override is set as follows: When the rapid traverse override is set to 0%,the actual rapid traverse rate is 2%. Actual rapid traverse rate is 0% only when feed override is 0.

5.8.3 Increment Magnification Selection

Under incremental and handwheel mode, the axis move a fixed distance after an command pulse is issued (the handwheel rotates a graduation or [Axis feed] is pressed once). The value of fixed distance is controlled by incremental magnification.

HNC-808Di-TU has three incremental magnification buttons "**×1**", "**×10**" and "**×100**". The corresponding relationship between incremental magnification keys and distance values is shown below:

Incremental magnification	**×1**	**×10**	**×100**
Increment value **(mm)**	**0.001**	**0.01**	**0.1**

Note

- These keys are interlocked, namely others will be invalid (the indicator light lights off) when one is pressed (the indicator light lights up).

6 Program Editing and Management

Operation name	Search of machining or editing program	Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be searched have already existed	Display interface	3.2.2 "Select program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface and menu
3	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk, and user disk
4	Press 「Cursor」 or 「PgUp/PgDn」		● Complete the search, and move the cursor to the program name to be searched ● The searched programs can be used for

			two purposes
5.1	Press 「Enter」		● The searched programs are used for machining program loading and running
5.2	Press 『Backstage editing』		● The searched programs are used for entering program editing status

6.1 Program Search

HNC-808DiM system has program search function under "Machining" function set and "Program" function set, but the usage of programs searched under two function sets is different. Programs found under "Machining" function set are used for machining program loading, program backstage editing and other operations; while programs found under "Program" function set are used for program management including program copy, paste, delete and program transmission among different disks.

6.1.1 Search of Machining or Editing Program

6.1.1.1 Direct search

6.1.1.2 Search programs under different disks by "Search" function

Operation name	Search of machining or editing program		Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be searched have already existed		Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description	
1	Press 〖Machining〗		● Default interface, main menu	
2	Press 『Select program』		● "Select program" sub-interface, level 2 menu	
3	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk and user disk	
4	Press 『Find』		● Prompt: Enter a file name	

5	(Enter a file name)	---	● e.g.: Onc123 请输入查找的文件:Onc123
6	Press 「Enter」		● Complete searching programs, and move the cursor to the program to be searched; ● Searched programs can be used for two purposes
7.1	Press 「Enter」		● The searched programs are used for machining program loading and running
7.2	Press 『Backstage editing』		● The searched programs are used for entering program editing status

6.1.1.3 Search programs under the directory by "Search" function

Operation name	Search of machining or editing program	Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be searched have already existed	Display interface	3.2.2 "Select program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface, level 2 menu
3	Press 『System disk』 or 『USB flash disk』,etc.		● Select system disk, USB flash disk, network disk, and user disk
4	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the selected file directory name
5	Press 「Enter」		● Confirm and open the directory
6	Press 『Find』		● Prompt: Enter a file name
7	(Enter a file name)	---	● e.g.: Onc123 请输入查找的文件:Onc123
8	Press 「Enter」		● Complete searching programs, and move the cursor to the program to be searched; ● Searched programs can be used for two purposes

9.1	Press 「Enter」		● Th searched programs are used for machining program loading and running
9.2	Press 『 Backstage editing』		● The searched programs are used for entering program editing status

6.1.2 Management Program Search (to Be Transmitted or Deleted)

6.1.2.1 Direct search

Operation name	Program search management (copy, paste, etc.)	Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be searched have already existed	Display interface	3.4 "Program" function set interface

SN	Operation steps	Key	Description
1	Press 〖Program〗		● Default interface, main menu
2	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk, and user disk
3	Press 「Cursor」 or 「PgUp/PgDn」		● Complete search ● Move the cursor to the program name to be searched

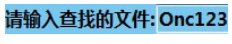
6.1.2.2 Search programs under different disks by "Search" function

Operation name	Program search management (copy, paste, etc.)	Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be searched have already existed	Display interface	3.4 "Program" function set interface

SN	Operation steps	Key	Description
1	Press 〖Program〗		● Default interface, main menu
2	Press 『System disk』 or 『USB flash disk』,etc.		● Select system disk, USB flash disk, network disk, and user disk

3	Press 『Find』		● Prompt: Enter a file name
4	(Enter a file name)	---	● e.g.: Onc123 
5	Press 「Enter」		● Complete search ● Move the cursor to the program name to be searched

6.1.2.3 Search programs under the directory by "Search" function

Operation name	Program search management (copy, paste, etc.)		Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be searched have already existed		Display interface	3.4 "Program" function set interface
SN	Operation steps	Key		Description
1	Press 〖Program〗			● Default interface, main menu
2	Press 『System disk』 or 『USB flash disk』, etc.			● Select system disk, USB flash disk, network disk and user disk
3	「 Cursor 」 or 「PgUp/PgDn」			● Move the cursor to the selected file directory name
4	Press 「Enter」			● Confirm and open the directory
5	Press 『Find』			● Prompt: Enter a file name
6	(Enter a file name)	---		● e.g.: Onc123 
7	Press 「Enter」			● Complete search ● Move the cursor to the program name to be searched

6.2 Program Editing

This system enters program editing status in 4 ways. 1. " New program" under "Machining" function set; 2. "Program editing" under "Machining" function set; 3. "Backstage editing" under "Machining" function set; 4. "New programs" under "Program" function set.

6.2.1 Create New Programs

The "New programs" function can be found both under "Machining" function set and "Program" function set. Although the two " are operated in different ways, their functions are basically identical. Configuration of two "Create new programs" can simplify operator's use.

6.2.1.1 Create new programs under "Machining" function set

1) Create new programs under "Program editing" sub-interface

Operation name	Create new programs under "Machining" function set	Working mode	Auto, single block, jog, MDI
Basic requirements	The new program name cannot be the same as the existing program names	Display interface	3.2.3 "Edit program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Edit program』		● Enter current loading program editing status
3	Press 『New』		● Prompt: Please enter a file name
4	(Enter a program name)	---	● Input program name su请输入文件名: O321
5	Press 「Enter」		● Confirm the new file name and enter the editing area ● If the name has already existed, the system will prompt and ask to re-enter
6	(Edit program)	---	● Complete program editing
7	Press 『Save file』		● A prompt message Program has been saved will be given ● The new program is immediately loaded as machining program

2) Create new programs under "Backstage editing" sub-interface

Operation name	Create new programs under "Machining" function set	Working mode	Auto, single block, jog, MDI

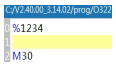
Basic requirements	The new program name cannot be the same as the existing program names	Display interface	3.2.2 "Select program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface, level 2 menu
	Press 『Backstage editing』		● Enter the "Backstage editing" sub-interface, level 3 interface
3	Press 『New』		● Prompt: Please enter a file name
4	(Enter a program name)	---	● Input program name suc请输入文件名: O321
5	Press 「Enter」		● Confirm the new file name and enter the editing area ● If the name has already existed, the system will prompt and ask to re-enter
6	(Edit program)	---	● Complete program editing
7	Press 『Save』		● A prompt message Program has been saved will be given ● The new program is immediately loaded as the machining program

6.2.1.2 Create new programs under "Program" function set

Operation name	New programs created under "Program" function set	Working mode	Auto, single block, jog, MDI
Basic requirements	The new program name cannot be the same as existing program names	Display interface	3.4 "Program" function set interface

SN	Operation steps	Key	Description
1	Press 〖Program〗		● Default interface, main menu
2	Press 『New』		● A prompt message "Please enter a file name" will be given 请输入文件名: O321
3	(Enter a file name)		● e.g.: Onc321

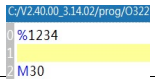
4	Press 「Enter」		● Confirm file name and switch to "Machining" function set ● Enter the editing area 
5	(Edit program)	---	● Complete program editing
6	Press 『Save』		● A prompt message Program has been saved will be given ● The new program is not loaded as machining program

6.2.2 Modification and Editing of Program

Existing programs should be edited and modified in "Machining" function set of this system. There are two types of edited and modified programs, current loading program and non-loading program.

6.2.2.1 Editing and modification of current loading program

Operation name	Editing and modification of current loading program	Working mode	Auto, single block, jog, MDI
Basic requirements	Existing loading programs	Display interface	3.2.3 "Edit program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Edit program』		● Enter current loading program editing status 
3	(Edit program)	---	● Edit and modify existing loading programs
4	Press 『Save』		● Program has been saved

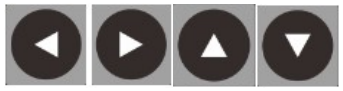
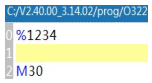
Note:

- The machine tool should not be at running status while editing current machining program.

6.2.2.2 Backstage editing and modification of non-loading program

Operation name	Backstage editing and modification of non-loading program	Working mode	Auto, single block, jog, MDI

Basic requirements	Existing programs to be modified	Display interface	3.2.2 "Select program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface
3	Press 「Cursor」		● Select current programs to be edited and modified
4	Press 『Backstage editing』		● Enter program editing status 
5	(Edit program)	---	● Edit and modify the existing loading programs
6	Press 『Save file』		● Program has been saved

Note:

- While selecting current loading program under the "Select program" sub-interface, current loading program also can be edited through "Backstage editing" function.

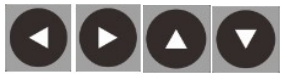
6.2.3 Save as

"Save as" function of this system is to integrally and quickly copy the program of current editing status. Thus, current loading program can enter program editing status in the "Edit program" sub-interface and non-loading programs can enter program editing status in the "Backstage editing" sub-interface in order to realize the saving.

6.2.3.1 Save the current loading program as

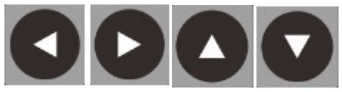
Operation name	Save the current loading program as	Working mode	Auto, single block, jog, MDI
Basic requirements	The file name cannot be the same as the existing program names	Display interface	3.2.3 "Edit program" sub-interface

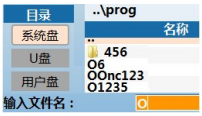
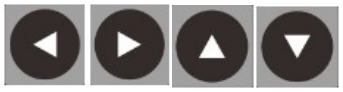
SN	Operation steps	Key	Description

1	Press 〖Machining〗		● Default interface, main menu
2	Press 『Edit program』		● Enter current loading program editing status
3	Press 『Save as』		● Storage target selection dialog box ● "System disk, USB flash disk or user disk" can be selected ● The file directory of each disk can be selected 
4	Press 「Cursor」		● Move the cursor to the selected target disk or file directory name
5	Press 「O」		● Activate file name input box
6	(Input the saved file name)		● If the name is the same as the existing program name, the original program will be covered
7	Press 「Enter」		● The current loading program is saved in the target position ● Exit the storage target selection dialog box ● The saved file is the current editing program

6.2.3.2 Save the non-loading program as

Operation name	Save the non-loading program as	Working mode	Auto, single block, jog, MDI
Basic requirements	The name cannot be the same as the existing program names	Display interface	3.2.2 "Select program" sub-interface

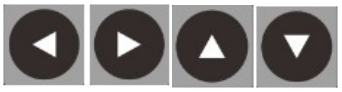
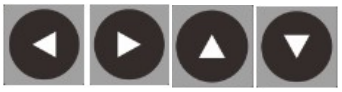
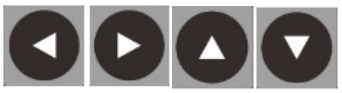
SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu
2	Press 『 Select program』		● "Select program" sub-interface
3	Press 「Cursor」		● Select the current program to be saved
4	Press 『 Backstage editing』		● Enter program editing status

5	Press 『Save as』	文件 另存	● Storage target selection dialog box ● "System disk, USB flash disk or user disk" can be selected ● The file directory of each disk can be selected 
6	Press 「Cursor」		● Move the cursor to the selected target disk or file directory name
7	Press 「O」	O	● Activate file name input box
8	(Input the saved file name)		● If the name is the same as the existing program name, the original program will be covered
9	Press 「Enter」	Enter 确认	● Current loading program is saved in the target position ● Exit the storage target selection dialog box ● The saved file is the current editing program

6.2.4 Copy and Paste of Block

Operation name	Copy and paste of program block	Working mode	Auto, single block, jog, MDI
Basic requirements	Enter program editing status	Display interface	3.2 "Machining" function set interface 3.3 "Program" function set interface

SN	Operation steps	Key	Description
1	(Edit program)	新建 程序 后台 编辑 编辑 程序	● There are 4 ways to enter program editing status: ✧ "New programs" function under "Machining" function set ✧ "Edit program" function under "Machining" function set ✧ "Backstage editing" function under "Machining" function set ✧ "New programs" function under "Program" function set
2	『Block』	块操作	● Enter "Block operation" sub-interface

3	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the head of the copied block
4	『block head』		---
5	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the end of the copied block
6	『Block end』		---
7	『Copy』		---
8	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to where the current program or other programs are to be pasted
9	『Paste』		● Paste succeeds
10	『Save』		● Exit and save program

6.3 Program management

6.3.1 Rename of file directory and program

Operation name	Rename of file directory and program	Working mode	Auto, single block, jog, MDI
Basic requirements	The existing programs can be found	Display interface	3.4.4 "Program rename" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Program〗		● Default interface, main menu
2	(Search directory and program)	---	● Move the cursor to the directory and program to be renamed according to 6.1.2
3	Press 『→』		● Enter "Program" set, level 1 extension menu
4	Press 『Rename』		● Prompt: Enter a new file name
5	(Rename a file)	---	● Enter a new file name

6	Press 「Enter」		● Confirm the new file name ● Prompt: Rename the old file as a new file

6.3.2 Copy and Paste of File Directory and Program

Operation name	Copy and paste file directory and program	Working mode	Auto, single block, jog, MDI
Basic requirements	The existing programs can be found	Display interface	3.4.2.2 "Program copy and paste" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Program〗		● Default interface, main menu
2	(Search the program to be copied)	---	● Move the cursor to the program name to be copied according to 6.1.2
3	Press 『→』		● Enter "Program" set, level 1 extension menu
4	Press 『Copy』		● Prompt: Select the target disk for paste
5	(Select the target disk or directory)	---	● Move the cursor to the target disk or the file directory according to 6.1.2 the search function of "Program" function set
6	Press 『Paste』		● Prompt: Paste succeeds

6.3.3 Program Deletion

6.3.3.1 Program deletion under "Machining" function set

Operation name	Deletion of program (under "Machining" function set)	Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be deleted can be found	Display interface	3.2.2 "Program selection" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu

2	Press 『Select program』		● "Select program" sub-interface
3	(Search directory and program)	---	● Move the cursor to the program name to be deleted according to 6.1.2 the search function of "Program" function set
4	Press [Delete]		● Prompt: "Confirm to delete the selected file? (Y/N)"
5	Press 「Y」 or 「N」		● Press 「Y」 to complete deletion ● Press 「N」 to abandon deletion

6.3.3.2 Program deletion under "Program" function set

Operation name	Deletion of program (under "Program" function set)	Working mode	Auto, single block, jog, MDI
Basic requirements	Programs to be deleted can be found	Display interface	3.4 "Program" function set interface

SN	Operation steps	Key	Description
1	Press 〖Program〗		● Default interface, main menu
2	(Search the program to be deleted)	---	● Move the cursor to the program name to be deleted according to 6.1.2 the search function of "Program" function set
3	Press [Delete]		● Prompt: "Confirm to delete the selected file? (Y/N)"
4	「Y」 or 「N」		● Press 「Y」 to complete deletion ● Press 「N」 to abandon deletion

7 Auto Operation

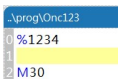
7.1 Auto Operation

7.1.1 Load Machining Program

The machining program can be loaded only under "Machining" function set. Although new programs can be created under "Program" function set, the interface will switch to "Machining" function set when this operation is conducted, and it cannot be loaded as the machining program automatically

7.1.1.1 Load a new program as machining program

Operation name	Load a new program as machining program	Working mode	Auto, single block, jog, MDI
Basic requirements	Create new programs under "Machining" function set	Display interface	3.2.3 "Edit program" sub-interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● Maintain the original interface
2	Press 〖Machining〗		● Default interface, main menu
3	Press 『Edit program』		● The cursor enters the loaded program editing area
4	Press 『New』		
5	(Enter a file name)	---	● Enter a new file name, such as “nc123” ● The address word of the new file name is O and needs not inputted
6	Press 「Enter」		● Confirm the input, and the file name is Onc123 ● The cursor enters the editing area 
7	(Edit program)	---	● Edit program and complete
8	Press 『Save』		● The newly edited program is loaded as machining program immediately ● Prompt: file has been saved

Note:

- After the new program is saved under "Machining" function set, it can be loaded as the current machining program automatically
- The new program cannot be loaded as the machining program under "Program" function set.

7.1.1.2 Load existing programs as machining program

Operation name	Load existing programs as machining program		Working mode	Auto, single block, jog, MDI
Basic requirements	The program to be loaded has already existed in the disk		Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description	
1	Press 【Auto】		● Maintain the original interface	
2	Press 〖Machining〗		● Default interface, main menu	
3	Press 『Select program』		● Search programs as per 6.1.2	
4	Press 『System disk』, etc.		● Select system disk/USB flash disk/network disk/user disk	
5	(Search the loading program)	---	● Select the program to be loaded as "Current machining program" ● Search programs as per 6.1.1	
6	「Enter」		● Loading completed	

7.1.2 Program Running

Operation name	Program running		Working mode	Auto
Basic requirements	The machining program has been loaded		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	

1	Press 【Auto】		● Maintain the original interface
2	Press 〖Machining〗		● Default interface, main menu
3	(Loading program)	---	● Load the machining program as per 7.1.1
4	(Safety check)	---	● Complete deceleration and lock, etc.
5	Press [Cycle start]		● Execute a program automatically

Note:

- Before running a new program automatically, complete tool setting;
- Although automatic machining can not only be conducted under the "Machining" function set, it is easier to operate and observe under "Machining" function set.

7.1.3 Program Verification

Operation name	Program verification	Working mode	Auto, single block
Basic requirements	The machining program has been loaded	Display interface	3.2.4 "Verify program" sub-interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● Maintain the original interface
2	Press 〖Machining〗		● Default interface, main menu
3	(Loading program)	---	● Load machining program as per 7.1.1
4	Press 『Verify program』		● The working mode is displayed as "Verify" ● The『Verify program』 soft key is highlighted
5	Press [Cycle start]		● Exit the verification after automatic operation ● Press [Reset] to exit verification

7.1.4 Program Graphics Simulation

Operation name	Program graphics simulation	Working mode	Auto, single block
Basic requirements	The machining program has been loaded	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● Maintain the original interface
2	Press 〖Machining〗		● Default interface, main menu
3	(Loading program)	---	● Load machining program as per 7.1.1
4	Press 『Display switch』		● Press this key once to switch to an interface and, can be switched switch among different interfaces cyclically ● Select “Graphics+ program” interface
5	Press [Cycle start]		● Conduct automatic operation and realize graphics simulation

Note

- For the size and position of workblank used for graphics simulation, refer to 3.2.7 "Graphics setup" sub-interface description

7.2 Auto Operation Control

7.2.1 Single-block Operation

Operation name	Single-block operation	Working mode	Single block
Basic requirements	Complete the loading of machining program	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description

1	Press 【Single block】		● Maintain the original interface
3	Press 〖Machining〗		● Default interface, main menu
4	(Loading program)	---	● Load the machining program as per 7.1.1
5	Press [Cycle start]		● Press Start once to execute a block of program, and cycle in turn

Note:

- Like the auto operation mode, in the single block mode the verification and simulation can also be performed.

7.2.2 Block Skip Operation

Operation name	Block skip operation	Working mode	Auto, single block
Basic requirements	Program block number is prefixed with "/", e.g.: /N1 X30 Z5	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description
1	Press 【 Jog 】 or 【Handwheel】	 or 	● This function can only be set under jog, handwheel, and incremental mode
2	Press [Block skip]		● The blocks with the block skip symbol will be skipped
3	Press 【Auto】		● Maintain the original interface
4	Press 〖Machining〗		● Default interface, main menu
5	(Loading program)	---	● Load machining program as per 7.1.1
6	Press [Cycle start]		● Skip the marked block during auto operation.

Note:

- If the block with skip symbol "/" is executed after the [Block skip] key is pressed, the system skips over this block of block

and directly executes the next block of commands.

- If the [Block skip] key is not pressed, the system still executes this block of command in order.

7.2.3 Run from Any Line

Operation name	Run from any line	Working mode	Auto, single block
Basic requirements	Cannot start from the subprogram line	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● Maintain the original interface
2	Press 〖Machining〗		● Default interface, main menu ● Correctly load the programs which need the any line execution
3	Press 『→』		● Enter the extension menu
4	Press 『Any line』		● Enter the "Any line" submenu
5	Press 『Designated line number』 and 『Designated N number』	 Or 	● The indicator light lights off ● Suspend the operation
6	(Input line number)	---	● Input a value, such as 8
7	Press 「Enter」		● Confirm the input ● The cursor moves to the line in front of the input ● Move the cursor to the selected any line by 「Cursor」
8	Press [Cycle start]		● Start to run from the designated line

Note:

- "Any line mode selection" parameter 040113 can be set as 0-2 and the function is shown below:

  0: Non-scanning mode. The modal before the target line is not inherited;

1: Scanning mode except Z axis. The modes before the target line are inherited except Z axis mode;

2: Full scanning mode. The modal before the target line are inherited.

- The parameter Any axis in-position sequence 040114 can set the in-position sequence of each axis. The parameter is of the numerical type. The corresponding relationship between the bit and the axis is shown below:

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

XYZABCUVW from low bit to high bit. The larger the value is the later the axis reaches in-position. 0 means the axis is not configured.

For milling system, 040114=211 means X/Y axis moves to the right position and then Z axis reaches the right position.

For lathes system, 040114=101, it means that X/Z arrives at the right position simultaneously and Y does not move.

- While using the "Designated N number" function, there should be the command address N at the block head.

7.2.4 Stop Operation

Operation name	Stop operation		Working mode	Auto
Basic requirements	There is M00 "Stop operation" command in the loading program		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Auto】		● Maintain the original interface	
2	Press 〖Machining〗		● Default interface, main menu ● Load machining program correctly	
3	Press [Cycle start]		● The program is running	
4	(Execute M00 command)	----	● The program suspends ● Manual tool change and other operations can	

			be executed
5	Press [Cycle start]		● Continue running subsequent programs

7.2.5 Optional Stop

Operation name	Optional stop	Working mode	Auto
Basic requirements	There is M01 "Optional stop" command in the loading program	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● Maintain the original interface
2	Press [Optional stop]		● If this step is not executed, continuously run the program
3	Press 〖Machining〗		● Default interface, main menu ● Load machining program correctly
4	Press [Cycle start]		● The program is running
5	(Execute M01 command)	----	● The program suspends ● If step 2 is not executed, the program does not stop but runs continuously
6	Press [Cycle start]		● Continue running subsequent programs

7.2.6 Dwell Operation

Operation name	Dwell operation	Working mode	Auto
Basic requirements	The program continuously runs	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description

1	Press 【Auto】		● Maintain the original interface
2	Press 〖Machining〗		● Default interface, main menu
3	(Running program)	---	● The program is running
4	Press [Feed hold]		● The indicator light lights off ● Dwell operation
5	Press [Cycle start]		● Continuous operation

Note:

- Feed hold cannot be validated immediately during thread machining until the thread command is completed.

7.2.7 Terminate Operation

Operation name	Terminate operation		Working mode	Auto
Basic requirements	The program continuously runs		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Auto】		● Maintain the original interface	
2	Press 〖Machining〗		● Default interface, main menu	
3	(Running program)	---	● The program is running	
4	Press [Feed hold]		● The indicator light lights off ● Suspend operation	
5	Press 【Jog】		● Close MST manually	
6	(Disable M and S functions)	---	● Disable MST manually	

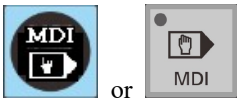
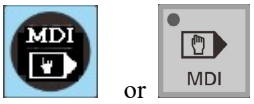
7	Press [Emergency stop]		● Terminate operation ● Reset

7.3 MDI Operation

There are two types of “MDI” keys of HNC-808Di-TU system,

- MDI key is the working mode key 【MDI】 on the lower panel
- MDI key is the function set key 【MDI】 on the upper panel

For HNC-808Di-TU system, The MDI key is the working mode 【MDI】 key. Operation and function of two types of "MDI" keys are basically identical.

Operation name	MDI operation		Working mode	MDI
Basic requirements	The system can run normally		Display interface	
SN	Operation steps	Key	Description	
1	Press 【MDI】		● Enter the MDI interface and the main menu ● The cursor is in the editing area	
2	(Edit MDI program)	---	● Edit multiple lines at a time before operation	
3	Press 『Input』		● Input all programs in the editing area	
4	Press [Cycle start]		● The machine tool runs as per the input program ● Programs in the editing area are reserved, even though the interface is switched	
5.1 To run MDI program in single block				
5.1.1	Press 【MDI】		● Enter the MDI interface and the main menu ● The cursor is in the editing area	
5.1.2	Press 【Single block】		● The indicator light of single-block lights up	

5.1.3	Press [Cycle start]		● The machine tool runs as per the input program ● Programs in the editing area are reserved, even though the interface is switched
5.2 To rerun programs in the editing area			
5.2.1	Press 『Input』		● Repeat steps 3 and 4
5.2.2	Press [Cycle start]		---
5.2 To suspend a program			
5.3.1	Press 『Dwell』		● The machine tool stops operation and is at feed hold status ● Press [Cycle start] to continuously run the subsequent programs
5.3 To cancel this operation			
5.4.1	Press 「Reset」		● This operation is canceled, and reenter to resume the operation ● Cannot reset during threading and drilling
5.4 To save MDI program in the system disk			
5.5.1	Press 『Save』		● The input box gives a prompt message: Please enter a file name
5.5.2	(Enter a file name)	---	● The program will be saved to PROG directory of the system disk
5.5.3	Press 「Enter」		● The program has been saved and a prompt message will be given
5.5 To clear programs in the MDI editing area at a time			
5.6.1	Press 『Clear』		● Power off to clear programs in the MDI editing area ● The edited MDI program cannot be cleared through switching the interface

Note:

- "MDI" is the key on the upper panel or the lower panel and it should be set through parameter 000371. When it is set as 0, it is the key of the lower panel. When it is set as 1, it is the key of the upper panel.
- If 〖MDI〗 key is on the lower panel, “MDI” is the working

mode key. There is an independent interface for MDI working mode. If the working mode is switched, the interface changes accordingly.

- If〖MDI〗key is on the upper panel, “MDI” is the function set key. This function is valid under auto mode or single block mode. The system will give a prompt message and an alarm after switching to other working modes.

7.4 Handwheel Precutting

This function controls the machine tool axis to run as per the programmed path through the MPG. It is often used to check whether tool setting is correct when the tool approaches the workpiece in order to avoid damaging the workpiece. This function is valid under automatic mode or single block mode.

Operation name	Handwheel trial cut	Working mode	Automatic
Basic requirements	Machining preparation is at normal automatic operating state of program	Display interface	See Chapter 3 "Machining" function set interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● This function can be executed under auto operation
2	Press [Handwheel simulation]		● If this function is valid, the indicator light lights up
3	Press [Cycle start]		● The commands run normally before the machine tool axis moves, such as the spindle rotation CW ● Then, the feed axis of the machine tool does not move
4	(Rotate the handwheel)		● If the handwheel rotates CW, the feed axis of the machine tool moves forward with the program ● If the handwheel rotates CCW, the feed axis of the machine tool moves backward with the program
5	(Check tool position)	---	● Visually inspect correctness of tool position
6	Press [Handwheel simulation]		● The unction is turned off and the indicator light lights off ● The machine tool continuously and automatically runs the subsequent programs until shutdown

7.5 Machining Information Query

Operation name	Machining information query	Working mode	Auto, jog, handwheel, reference point return, MDI
Basic requirements	---	Display interface	See Chapter 3 "Machining" interface

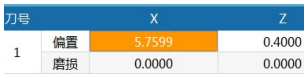
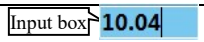
SN	Operation steps	Key	Description
1	Press〖Machining〗		● Default interface, main menu
2	Press 『→』		● Enter the extension menu
3	Press 『Machining information』		● The system displays number of processed parts and information related to system operation time ● Enter the "Operation control" sub-interface and display operational statistical data
4	Press 『Preset』		● Set total number of processed parts
5	Press 『Reset』		● Reset time and number of processed parts
6	Press 『Operation statistics』		● Export or eliminate machining information file

8 Tool Setting and Machining Setup

8.1 Directly Input Tool Compensation Value

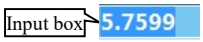
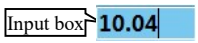
8.1.1 Directly Input Tool Compensation Value Under "Machining" Function Set

Operation name	Input tool compensation value for precutting	Working mode	Jog, handwheel
Basic requirements	Jog or handwheel mode	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Enter "Tool compensation table" 
2	Press 『Tool compensation setup』		● Enter "Tool compensation table" 
3	Press 「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set
4	Press 「Enter」		● Activate input box 
5	(Input offset value)	---	● Input “10.04” 
6	Press 「Enter」		● Confirm the input ● Skip to Z axis offset position of the tool

8.1.2 Directly Input Tool Compensation Value Under "Setup" Function Set

Operation name	Directly input tool compensation value	Working mode	Jog, handwheel
Basic requirements	Jog or handwheel mode	Display interface	3.3 "Setup" function set interface

SN	Operation steps	Key	Description
1	Press 〖Setup〗		● "Setup" default interface, main menu
2	Press 『Tool compensation setup』		● Enter "Tool compensation table" 
3	Press 「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the position to be set. e.g.: X axis offset of tool 1 should be set
4	Press 「Enter」		● Activate input box 
5	(Input offset value)	---	● Input "10.04" 
6	Press 「Enter」		● Confirm the input ● Skip to Z axis offset position of the tool

Note:

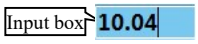
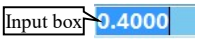
- When the compensation value is inputted directly, the original wear value is not cleared.
- Directly input tool compensation value, including offset value, wear value, tool nose radius, and tool nose direction number

8.2 Tool Setting for Precutting

8.2.1 Tool Setting for Precutting Under "Machining" Function Set

Operation name	Input tool compensation value for precutting	Working mode	Jog, handwheel
Basic requirements	Jog or handwheel mode	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● "Tool compensation setup" sub-interface

2	Press 『 Tool compensation setup』	刀补 设置	● Enter "Tool compensation table" 
3	Press 「Cursor」 or 「PgUp/PgDn」	PgUp 上页 PgDn 下页	● Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set
4	(Outer diameter of the workpiece precut)	---	● After precutting of workpiece, exit along Z axis (for ease of measurement).Then X coordinate of the tool in the workpiece coordinate system can be obtained
5	(Measure outer diameter of workpiece)	---	● Obtain X coordinate value of the tool under the workpiece coordinate system through measurement. e.g.: 10.04
6	Press 『 Trial cut diameter』	试切 直径	● Activate input box 
7	(Input the measured value)	---	● Input “10.04” 
8	Press 「Enter」	Enter 确认	● Confirm the input and clear wear value of the tool number. ● The input box exits the activation state
9	Press 「Cursor」 or 「PgUp/PgDn」	PgDn 下页	● The cursor moves to the next setup position. e.g.: Set Z axis offset of tool 1
10	(Precut end face of workpiece)	---	● After the precutting of workpiece, exit along X axis (for ease of measurement) in order to obtain Z coordinate of the tool under the workpiece coordinate system.
11	(Measure the length of workpiece)	---	● Obtain Z coordinate value of the tool under the workpiece coordinate system through measurement. ● If the zero point of the workpiece is on the front end and the end face for precutting is the front end, Z axis coordinate value of the tool in the workpiece coordinate system is 0 ● This value may be positive or negative
12	Press 『Precutting length』	试切 长度	● Activate input box 
13	(Input the measured value)		● Input "0" 
14	Press 「Enter」	Enter 确认	● Confirm the input and clear wear value under tool number ● The input box exits the activation state

Note:

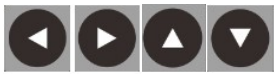
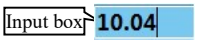
- Confirm "Precutting diameter" and "Precutting length", and then

clear the original wear value;

- The tool compensation value inputted for precutting only affects offset value and wear value of the tool.

8.2.2 Tool Setting for Precutting Under "Setup" Function Set

Operation name	Input tool compensation value for percutting	Working mode	Jog, handwheel
Basic requirements	Jog or handwheel mode	Display interface	3.3 "Setup" function set interface

SN	Operation steps	Key	Description
1	Press 〖Setup〗		● Enter "Tool compensation table" 
2	Press 「Cursor」		● Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set
3	(Outer diameter of the workpiece precut)	---	● After the precutting of workpiece, exit along A axis (for ease of measurement) in order to obtain X coordinate of the tool under the workpiece coordinate system
4	(Measure outer diameter of workpiece)	---	● Obtain X coordinate value of the tool under the workpiece coordinate system through measurement. e.g.: 10.04
5	Press 『 Precutting diameter』		● Activate input box 
6	(Input the measured value)	---	● Input “10.04” 
7	Press 「Enter」		● Confirm the input and clear wear value under tool number. ● The input box exits the activation state
8	Press 「Cursor」		● The cursor moves to the next setup position. e.g.: Set Z axis offset of tool 1
9	(Precut the end face of workpiece)	---	● After the precutting of workpiece, exit along X axis (for ease of measurement) in order to obtain Z coordinate of the tool under the workpiece coordinate system.
10	(Measure the length of workpiece)	---	● Obtain Z coordinate value of the tool under the workpiece coordinate system through measurement. ● If the zero point of the workpiece is on the front end and the end face for precutting is the front end,

			Z axis coordinate value of the tool in the workpiece coordinate system is 0 ● This value may be positive or negative
11	Press 『 Precutting length』	试切 长度	● Activate the input box Input box 0.4000
12	(Input the measured value)		● Input "0" Input box 0
13	Press 「Enter」	Enter 确认	● Confirm the input and clear wear value under tool number ● The input box exits the activation state

Note:

- Confirm "Precutting diameter" and "Precutting length" and then clear the original wear value;
- The tool compensation value inputted for precutting only affects offset value and wear value of tool.

8.3 Overall Translation of Tool Post

8.3.1 Overall Translation of Tool Post Under "Machining" Function Set

Operation name	Overall translation of tool post	Working mode	Jog, handwheel
Basic requirements	The machine tool is at the stop state	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set

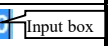
SN	Operation steps	Key	Description
1	Press 〖Machining〗	加工 Mach	● "Tool compensation setup" sub-interface
2	Press 『 Tool compensation setup』	刀补 设置	● Enter "Tool compensation table" 刀号 X Z; 1 偏置 5.7599 0.4000; 磨损 0.0000 0.0000
3	Press 『 Tool post move』	刀架 平移	● Activate the dialog box 0.0000 0.0000 Input box ● Press「Esc」to exit the activation status
4	(Input translation amount)	Space 空格	● Separate the two coordinates values by Space ● e.g.: X/Z offset is 0.02/0.01, fill in 0.02 space 0.01

5	Press 「Enter」		● Confirm the input and all tool compensation amounts are translated by a set value

Note:

- Confirm "Tool post move" and the original wear value will not be cleared;
- Tool post translation function cannot be used during program run.

8.3.2 Overall Translation of Tool Post Under "Setup" Function Set

Operation name	Overall translation of tool post		Working mode	Jog, handwheel
Basic requirements	The machine tool is at the stop state		Display interface	3.3 "Setup" function set interface
SN	Operation steps	Key	Description	
1	Press 〖Setup〗		● "Tool compensation setup" sub-interface 	
2	Press 『Tool post move』		● Activate the dialog box  ● Press 「Esc」 to exit the activation state	
3	(Input translation amount)		● Separate the two coordinates values by Space ● e.g.: X/Z offset is 0.02/0.01, fill in 0.02 space 0.01	
4	Press 「Enter」		● Confirm the input and all tool compensation amounts are translated by a certain distance	

Note:

- Confirm "Tool post move" and the original wear value will not be cleared.

8.4 F/S Machining Setup

When F and S commands are not used in machining program, this function can be used to designate F and S values in machining program. This function also can be used to modify the default spindle rotation speed.

When 010103 parameter value is set as 1 or includes 1, this function is

valid.

Operation name	Machining configuration	Working mode	Jog, auto
Basic requirements	When there is no F/S command in machining program	Display interface	See Chapter 3 "Machining" interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Switch to the machining interface
2	Press 『 Machining configuration』		● F and S setup menus pop up
3	(Set machining F and S values)		● If there is no F or S in machining program, this value prevails ● Set the default S (rotation speed in JOG)

Note:

- If there is no F/S value in the program, the value in the status bar can be validated immediately.
- If F/S is set in the program, the machining configuration column is invalid, and the data in machining configuration will be changed by the program.

8.5 Tool Measurement

"Tool measurement" function is realized by three stages. First, make preparations for tool measurement; second, automatically measure the relative positional relationship between tools using measuring instrument; third, set the positional relationship between the datum tool and the workpiece using offset setup function. If the datum tool is not set, tool setting should be conducted in this interface.

Operation name	Tool measurement	Working mode	(Handwheel jog) and (auto)
Basic requirements	The machine tool is configured with tool setter	Display interface	See 3.3.4 "Tool measurement" sub-interface

SN	Operation steps	Key	Description
1. Preparation for tool measurement			
1	Press 【Handwheel】		● In handwheel mode: Touching tool setter, tool setting and other operations are completed

2	Press 〖Setup〗		● Set tool measurement in the second page of the interface
3	Press 『→』		● Enter the extension menu
4	Press 『Tool measurement』		● Enter the tool measurement sub-interface
5	Press 『Measurement parameter』		● Enter the measurement parameter sub-interface ● Fill length and width of the measuring instrument, calibration of tool nose direction, measurement times, measurement speed, and trigger speed (which must be filled. See 3.3.4 for specific definition) ● Press 「Cursor」 to select the items to be filled ● Press 「Enter」 to activate the input
6	Press 「Cursor」		
7	Press 「Enter」		
8	(Fill in measurement parameter)	---	---
9	Press 「Enter」		● Press 「Enter」 to confirm the input ● Press 「Reset」 to abandon the input
2. Datum tool calibration			
10	Press 『Lathe tool calibration』		● Enter the lathe tool calibration sub-interface ● Calibration is the precondition of measurement
11	Press 「Cursor」		● Move the cursor to the position of the measuring instrument required ➢ Z axis moves to P1 ➢ X axis moves to P2 ● The tool moves to P point corresponding to the tool setter ● Press 「Enter」 to confirm position, and generate the initial value of tool setting P1绝对位置（Z向） P2绝对位置（X向） P3绝对位置（Z向） P4绝对位置（X向）
12	(Move the tool manually)	---	
13	Press 「Enter」		
14	Press 【Auto】		● Switch working mode to "Auto"
15	Press 『Start measurement』		● Press 『Start measurement』 and a prompt message Check the cursor position will be given ● When the tool offset is greater than 50mm, pay attention to avoid the collision during measurement.
16	【Cycle start】		● The tool starts tool setting automatically based on the set value of "Measurement parameter" ● Calibrate the datum tool automatically, and determine the positional relationship between the datum tool and the

			measuring instrument.
3. Automatic measurement, determine the position between tools			
17	Press 『Lathe tool measurement』		● Enter the lathe tool measurement sub-interface
18	(Set the tool for automatic measurement)	---	● Select the tool for automatic measurement by 「Cursor」 ● Activate "Tool nose direction" and determine "ON/OFF" by 「Enter」, and input
19	Press 『Start measurement』		● Press 『Start measurement』, and a prompt message Check the cursor position will be given ● When tool deviation between tools is greater than 50mm, collision should be avoided during measurement.
20	Press [Cycle start]		● The tool starts tool setting automatically based on the set value of "Measurement parameter" ● Generate relative position value between measurement tools automatically
4. Set the positional relationship between the datum tool and the workpiece			
21	Press 『Offset setup』		● Enter the offset setup interface ● Press 「Cursor」 to select the set parameters, and 「Enter」 to activate and input parameter value
22	(Set "Datum tool number")	---	● Select a measurement tool as the datum tool and fill in tool number
23	(Set "Whether to use the original tool offset")	---	● If the datum tool has been set and the offset value should be adopted, fill in "1" ● If the datum tool is not set or the value is not adopted, fill in "0"
24	(If the original tool offset is not used, tool setting is needed)	---	● If "Whether to use the original tool offset value" is set as "0", the tool setting should be completed under this interface. ● Working mode should be switched to handwheel during tool setting.

Note:

- This function is a limited function, and it should be enabled with relevant permissions. For permission enabling, refer to 3.6.4.

8.6 Thread Repair Function

This function is used to repair the newly clamped thread parts, and tool setting is needed for the parts before this function is used (to determine the positional relationship between the tool and the workpiece zero).

Operation name	Thread repair	Working mode	(Jog, handwheel) and (auto)
Basic requirements	Tool setting is needed for the newly clamped thread parts	Display interface	See Chapter 3 "Setup" interface

SN	Operation steps	Key	Description
1	Press 〖Setup〗		● Enter the "Setup" function set, default interface
2	Press 『→』		● Enter the extension menu
3	Press 『Thread repair』		● Enter the thread repair sub-interface
4	Press 『Invalid recutting』		● The spindle cannot rotate then
5	Rotate 【Handwheel】	 or 	● Rotate Z/X axis of the machine tool and align the tool nose at the peak of the thread groove
6	Press 『Set all axes』		● Input coordinate values of "Lead axis/spindle" of the tool in the machine coordinate system automatically. 
	Or press 『Input』		● Input coordinate values of "Lead axis/spindle" of the tool under the machine tool coordinate system manually.
7	Press 『Invalid re-cutting』		● Perform the thread repair machining
8	Press 【Auto】		● Run thread programs, and repair thread parts ● X coordinate value of thread programs, which should be consistent with the bottom diameter of repaired thread
	(Thread repair machining)	---	● Leave a certain margin for the first repair, and meet the repair requirements after multiple repairs

Note:

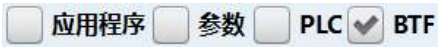
- Before repairing the thread, it is necessary to detect the tracking error value when the lead axis is running at 1000mm/min and fill it with the parameter value of the axis parameter 102049.
- While setting thread repair parameter, "Invalid recutting" in the submenu must be pressed to set tool setting or clear data;
- After thread repair parameter is set, "Valid recutting" must be pressed to realize thread repair function;
- Thread repair function can be used to repair single-start thread only.

9 Machine Tool Commissioning

9.1 System Upgrade

9.1.1 System Upgrade

Operation name	System upgrade	Working mode	Emergency stop
Basic requirements	●The machine tool must be at "Emergency stop" state ●System upgrade package file name is "**.BTF"	Display interface	3.3 "Maintain" function set interface

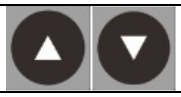
SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● Enter the "Maintain" function set interface
2	Press 『System upgrade』		● Enter the System upgrade" sub-interface
3	Press 『Window switch』		● Switch to "Upgrade selection" window on the upper part of the interface
4	Press 「Cursor」		● Select upgrade item ● Select BTF item for comprehensive upgrade
5	Press 「Enter」		● Confirm the selected item 
6	Press 『Window switch』		● Switch to the upgrade package file source selection window on the lower part of the interface 
7	Press 『USB flash disk』/『User disk』		● Select the upgrade package file from USB flash disk by default ● The upgrade package file in the user disk also can be selected
8	Press 「Cursor」		● Select upgrade file package ● The upgrade package file must be BTF file
9	Press 「Enter」		● Start system upgrade ● Do not power off before upgrade is completed

Note

- **The upgrade requires the corresponding system management permission, and generally the upgrade operation should be performed by HCNC technicians;**
- **Do not power off during system upgrade**

9.1.2 System Backup

Operation name	System backup	Working mode	Emergency stop
Basic requirements	The machine tool must be at "Emergency stop" state	Display interface	3.3 "Maintain" function set interface

SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● Enter the "Maintain" function set interface
2	Press 『System upgrade』		● Enter the "System upgrade" sub-interface
3	Press 『Window switch』		● Switch to the “Upgrade selection” window on the upper part of the interface
4	Press 「Cursor」		● Select the item to be backed up 
5	Press 「Enter」		● Confirm the selection
6	Press 「Cursor」		● Select backup 
7	Press 「Enter」		● Confirm the backup 
8	Press 『Window switch』		● Switch the cursor to the backup target disk window
7	Press 『USB flash disk』/『User disk』		● The default backup target disk is the user disk
8	Press 「Cursor」		● Move the cursor to the file directory to be backed up
9	Press 「Enter」		● Start system backup ● Do not power off before backup is completed

Note:

- **While backing up a file to the system disk, size of system disk should be noted. The backup package of V2.40.00 version**

software is about 200MB

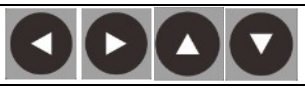
9.2 Batch Commissioning

This function is limited by permission. This function can load/back up PLC, canned cycle, parameter, G code, parameter configuration, and other files necessary for commissioning separately or in batches.

Operation mode and object of "Batch commissioning" function are similar to "Data management" function, where, there are more "Data management" files and only a single file can be operated.

9.2.1 Batch Load Commissioning

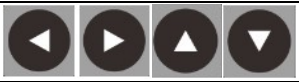
Operation name	Batch load commissioning	Working mode	Emergency stop
Basic requirements	The machine tool must be at "Emergency stop" state Files loaded in batches must be .tar files	Display interface	3.3 "Maintain" function set interface

SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● Enter the "Maintain" function set interface
2	Press 『Batch commissioning』		● Enter the "Batch commissioning" sub-interface
3	Press 『Window switch』		● Switch to system disk ● Enter file type selection window
4	Press 「Cursor」		● Move the cursor to type of the file to be loaded
5	Press 「Enter」		● Confirm the selection type
6	Press 『USB flash disk』/『User disk』		● Select the source disk of the loaded file
7	Press 『Window switch』		● Switch the cursor to the file source disk window
8	Press 「Cursor」		● Move the cursor to the type of the file to be loaded
9	Press 『Load』		● A prompt message "Whether to load file XXXX.tar?(Y/N)"

10	Press 「Y」 or 「Enter」	 or 	● A prompt message "Loading a file succeeds, please power off and restart!"

9.2.2 Batch Backup Commissioning

Operation name	Batch backup commissioning	Working mode	Emergency stop
Basic requirements	The machine tool must be at "Emergency stop" state	Display interface	3.3 "Maintain" function set interface

SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● Enter the "Maintain" function set interface
2	Press 『Batch commissioning』		● Enter the "Batch commissioning" sub-interface
3	Press 『Window switch』		● Switch to the system disk ● Enter the file type selection window
4	Press 「Cursor」		● Move the cursor to the type of the file to be backed up
5	Press 「Enter」		● Confirm the selection type
6	Press 『USB flash disk』/『User disk』		● Select a target disk to back up files
7	Press 『Window switch』		● Switch the cursor to the file source disk window
8	Press 「Cursor」		● Move the cursor to the file directory to be loaded
9	Press 『Backup』		● A prompt message "Please enter a backup package name " will be given
10	(Enter a backup package name)	---	● File name must be suffixed with .tar
11	Press 「Y」 or 「Enter」	 or 	● Complete the backup and a prompt message Backup succeeds is given

Note: Do not power off during backup or loading

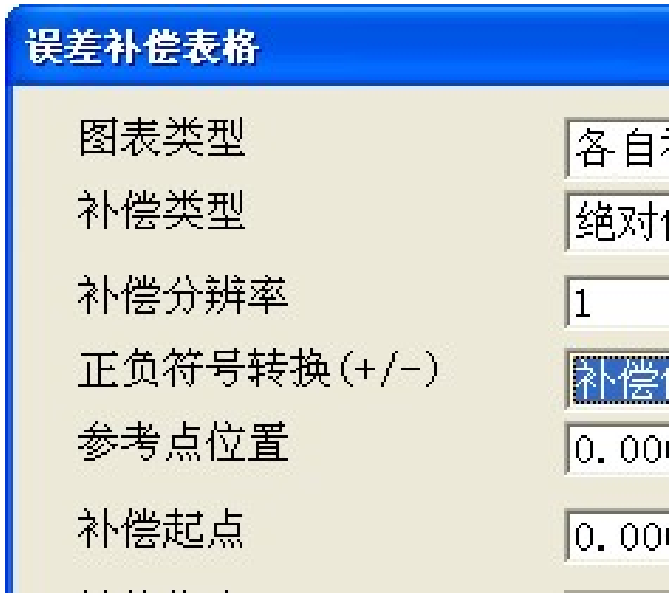
9.3 Pitch Error Compensation

Due to manufacturing error of machine tool, there is a certain error between the actual position and the command position of machine tool axis. This function can decrease error of actual position and command position through increasing or decreasing the actual movement of machine tool.

The laser interferometer can measure error between the actual position and the command position of machine tool axis, and the error compensation data file is generated. HNC-818D-TU system only supports *.rtl file generated by Renishaw laser interferometer.

9.3.1 Generation Of Pitch Error Compensation Data File

Presently, HCNC CNC system supports direct import of error compensation data file (*.rtl) generated by Renishaw laser interferometer only. When Renishaw software generates error compensation file, error compensation table should be set according to the following requirements (as shown below).



- "Chart type" **must** select "Separate compensation";
- "Compensation type" **must** select "Absolute ";
- "Compensation resolution" must be "1"um;
- "Conversion of symbols (+/-)" **must** select "Compensation value";
- "Reference point " **must be "0"**;
- The "Compensation start point" is the machine coordinate position

of the compensation start point, **it must be 0**;

- The "Compensation end point" is the machine coordinate position of the compensation end point;
- "Compensation interval" must be a positive value.

9.3.2 Operation Of Pitch Error Compensation Sub-interface

Users can enter this function sub-interface under "Machining" and "Maintain" function sets.

Enter the pitch error compensation sub-interface under the "Maintain" function set

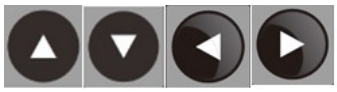
Operation name	Operation of pitch error compensation sub-interface	Working mode	Auto, single block, jog, incremental
Basic requirements	The machine tool is at stop state	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set

SN	Operation steps	Key	Description
1	Press 〖Maintain〗	维护 Mainte	● "Tool compensation setup" sub-interface
2	Press 『Parameter setup』	参数配置	● Enter the "Parameter setup" sub-interface
3	Press 『Pitch compensation』	螺距补偿	● Enter the pitch error compensation sub-interface (as shown below)

9.3.3 Import of Pitch Error Compensation Data File

Direct import of pitch error compensation data file under the "Maintain" function is introduced below (only available for direct import of rtl thread repair file generated by Renishaw software).

Operation name	Import of pitch error compensation file	Working mode	Auto, single block, jog, incremental
Basic requirements	Pitch error compensation data file has been generated correctly	Display interface	3.2 "Parameter setup" sub-interface under "Maintain" set

SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● "Maintain" main menu
2	Press 『Parameter setup』		● "Parameter setup" sub-interface and menu
3	Press 『Pitch compensation』		● "Pitch compensation" sub-interface and menu (as shown above)
4	Press 「Cursor」		● Select compensation axis
5	Press 「Cursor」 or 「Enter」	 	● Setup option of direction key (options in the above area 1, see Remarks for specific options) ● Press 「Enter」 to confirm option or setup
6	Press 『One-click clear』		● If pitch error compensation data is not imported for the first time, please press One-click clear to clear the pitch error compensation data ● If pitch error compensation data is imported for the first time, this step can be omitted
7	Press 『Load rtl』		● Enter "Error compensation data file (*.rtl)" for search ● Select the pitch error comensation compensation data file of corresponding axis
8	Press 『USB flash disk』/『User disk』		● Select compensation data file disk

9	Press 「Cursor」		● Select compensation data file
10	Press 「Enter」		● Press 「Enter」 to import thread compensation data
11	Press 「Reset」		● after the thread compensation data is imported successfully, press Reset to validate them
12	(Check thread compensation data)		● Check the thread compensation type, start point position, number of compensation points, compensation interval, and initial number of data table ● During unidirectional compensation type, check whether the backlash compensation is enabled, and the backlash value ● If the pitch error compensation data is imported incorrectly, execute the steps from step 4 again

Remarks: Options of area 1

- Selection of compensation axis: Axis 0, axis 1 and axis 2.....;
- Pitch error compensation type: Inhibit, unidirectional compensation, bidirectional compensation;
- Start point: The same as "Renishaw error compensation table";
- Compensation interval: The same as "Renishaw error compensation table";
- Compensation point number: The same as "Renishaw error compensation table";
- Initial number of data table: Initial number is 710000;
- Backlash compensation type: See parameter 300000 (which can be set as 0, 1, 2);
- Backlash value : See parameter 300001.

10 Use and Maintenance Information

10.1 Environmental Conditions

Operating conditions are shown below:

Environmental	conditions
Operating temperature (℃)	Non-freezing, 0- +45
Temperature variation	<1.1℃/min
Relative humidity	90%RH or lower (non-condensation) Normal condition: 75% or smaller Short-term (within a month): No more than 95%
Storage temperature (℃)	Nonfreezing, -20- +60
Storage humidity	Non-condensation
Surroundings	Indoor (sunproof) Anticorrosive, burn, frog, dust
Height	No more than 1000m above the sea level
Vibration (m/s)	5.9(0.6G) or lower at 10-60Hz

10.2 Grounding

Correct grounding is very important for electrical device and the aim is:

- protecting workers from electric shock arising from abnormal phenomena;
- Protect electronic devices from interference of the machine and other electronic devices nearby, which may result in abnormal operation of control device.

While installing machine tool, it must be reliably grounded, and neutral line in the power grid cannot be used as the earth wire; otherwise, personal injury or device damage may be caused and exceptional operation of device may be caused.

10.3 Power Supply

Power supply of the controller is supplied by the electrical control cabinet of the machine tool. For power supply of machine tool, please refer to installation manual of machine tool.

10.4 Dust Removal of Filter Fan Screen

Fan is an important element for ventilation and heat dissipation of CNC device. In order to prevent dust from entering the device from the fan, filter screens must be installed at air inlet and air outlet.

Dust will gradually stop up the filter screen after long-term use and consequently ventilation conditions will worse and even normal operation of devices will be affected. Thus, users should regularly clean all filter screens. Generally, filter screen of fan should be cleaned every three months and cleaning period should be reduced under poor conditions.

10.5 Use After Long-time Idle

If CNC device is reused after left unused for a long time, remove dust and dry it. Then, inspect connection and grounding of CNC device. Power on to run it for a period of time and rerun it when there is no system failure.