HNC-8 System Operating Manual (Milling Machine)

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”.

Copyright of the manual should be reserved by HCNC. Any units and individuals' publication or duplication should be deemed as illegal behavior and we will hold them accountable.

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

Introduction

Dear customer:

We feel greatly honored and express sincere gratitude to you for using our products!

This manual describes matters concerning interface and operation of HNC-8 CNC milling system in detail. 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 that are not especially described as possible in this manual should be regarded as “impossible” or “not allowed”.

In order to ensure safety and normal use of the product, please thoroughly read this manual before installation or use.

Safety warning

Improper operation will result in a safety accident, so operators must be qualified for transportation, installation, commissioning and machining.

Copyright

1 Overview

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

1.1 Basic Operation

HNC-808Di-M system is a CNC controller for CNC milling machine. The MCP panel is furnished with 6 working mode keys "Jog, auto, single block, MDI, incremental/handwheel, and reset". During operation of CNC lathe, function description and content of these 6 working modes are shown below.

Working mode	Functional description	Function application
Jog	Control continuous movement of machine tool axis and auxiliary action by Jog key.	Preparation for parts machining and simple machining.
Auto	The machine tool runs continuously and automatically based on the edited program.	Continuous and automatic machining, program verification of parts.
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 runs the manually inputted program.	Automatic machining and coordinate setup of simple parts.
Incremental/handwheel	Accurately control axis movement of machine tool by key or handwheel.	Tool setting or manual machining of simple parts.
Reset	Control of the axis to return to the reference point.	Calibrate the position of machine tool after start.

* For the non-Di series version, the MDI working mode is configured as the MDI function set of NC panel

1.2 Basic Function

To complete different work under different working modes, corresponding application functions should be used. The NC panel of HNC-808Di-M CNC device is furnished with 6 function keys "machining, setup, program, diagnosis, maintenance and user-defined (MDI)". Every function key corresponds to a group of function sets. User can select corresponding functions and interfaces form the function set through function soft keys (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 of auto machining	1. Program editing: Edit new programs*, edit current loading programs, edit options; 2. Program machining: Machining program selection, program verification, program machining; 3. Tool setting: Coordinate system, tool compensation setup*; 4. Interface display: Path setup, display switch; 5. Others: User macro, machining information, parameter setup (user)*.
Setup	Functions of tool setting	Tool setting (coordinate system, workpiece measurement, automatic tool setting), tool compensation setup*, tool life management
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 mark
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 device configuration and configuration sequence setup function: Device configuration 2. Setup function of common parameters: Parameter setup 3. Setup function of user optional 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, technology package, and time setup
User-defined ** (MDI)	Functions of manual data input	Dwell, clear, save, input

Description:

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

**For other series than HNC-8 Di series, user-defined key is often set as the MDI function.

1.3 Basic Display Interface

HNC-808Di-M system 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, and so on.

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

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

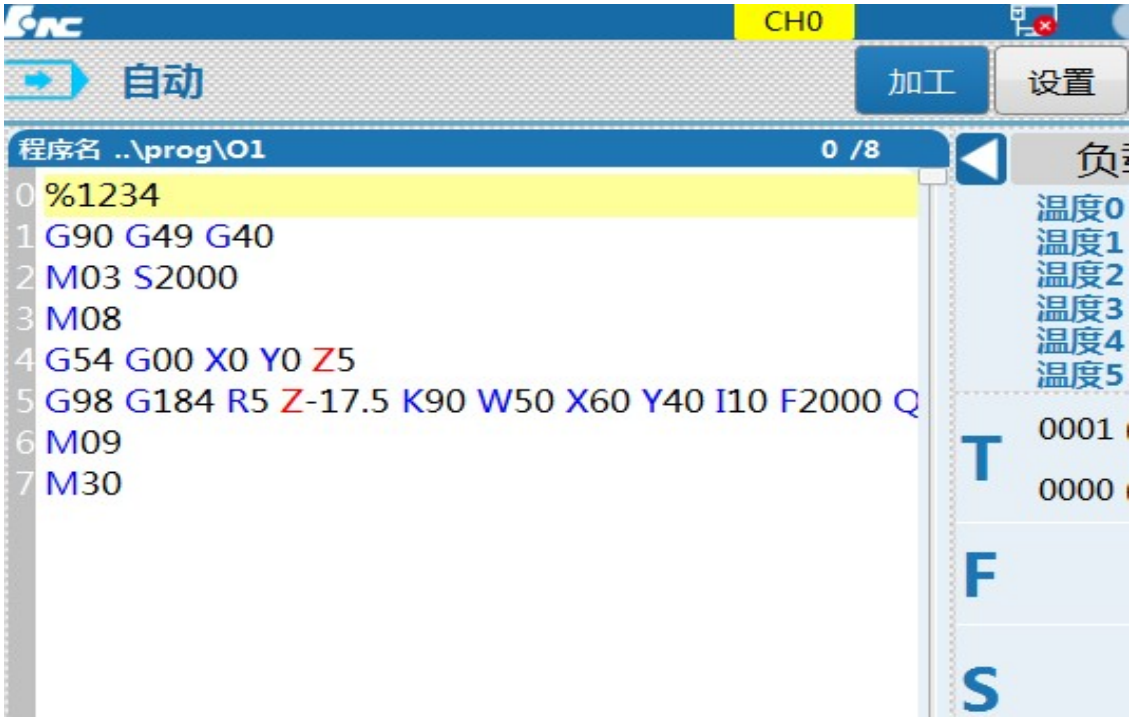
1.3.1 Machining Display Interface

The machining display interface enables the operator to observe the machining process and has 4 display forms: big character coordinate + program, joint coordinate, graphics path + program, and program. These 4 interface can be switched through 『Display switch』 soft key.

Big character coordinate + program display interface

Joint coordinate display interface

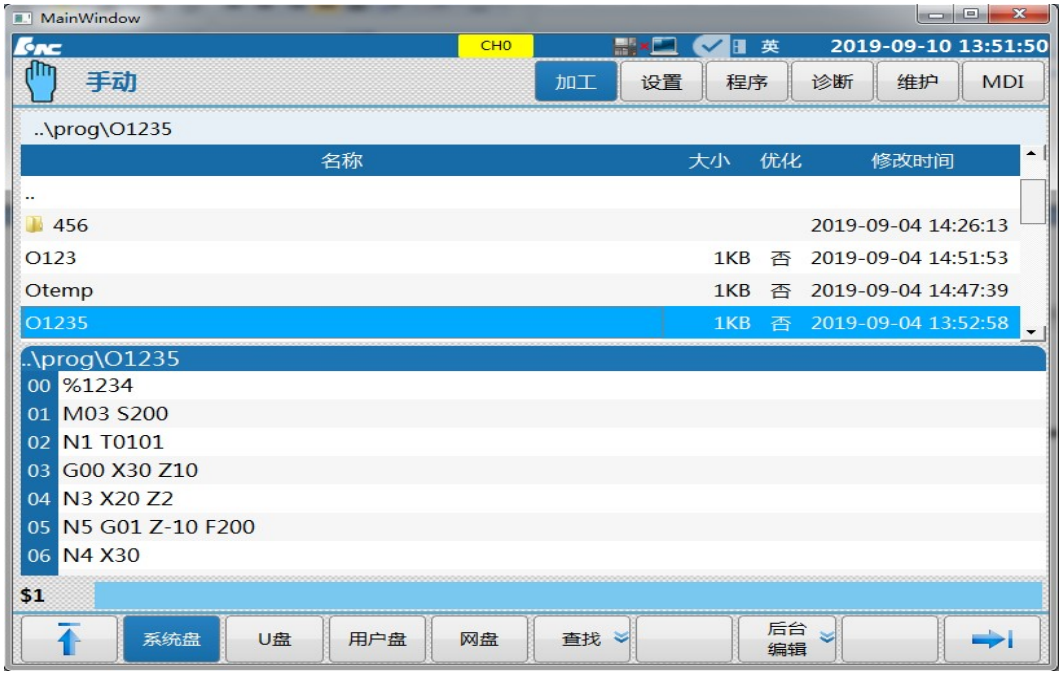
Graphics path + program display interface



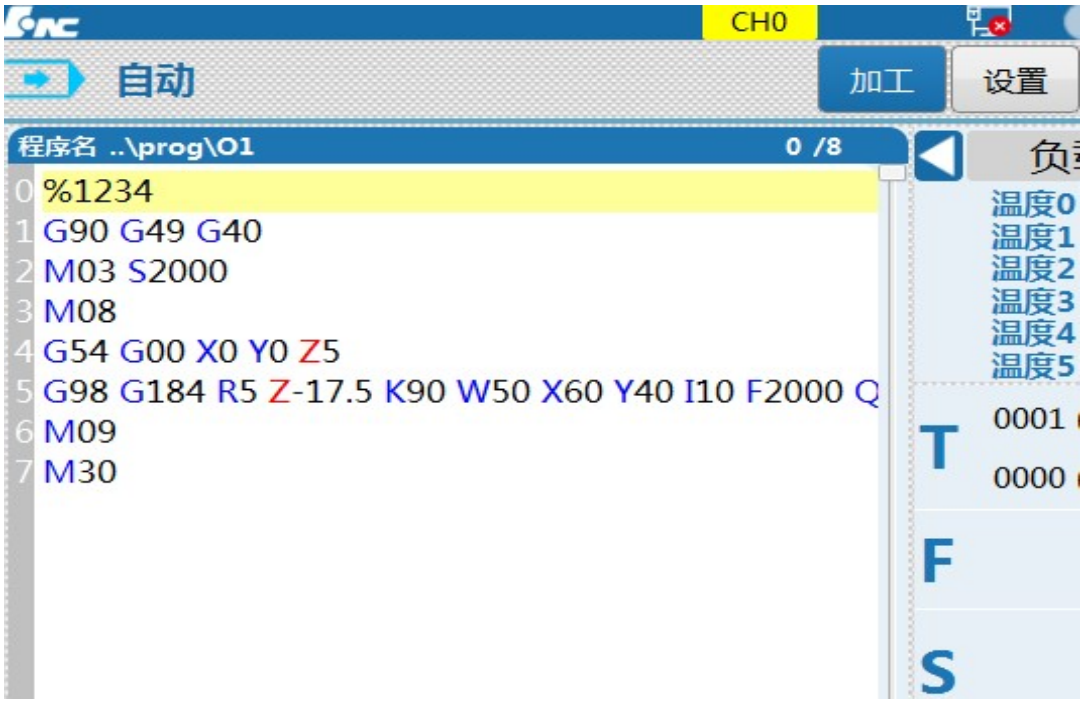
Program display interface

1.3.2 Program Selection And Edit Interface

This system can select programs by 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 programs found.



Program selection interface



Program editing interface

1.3.3 Machining Setup Interface

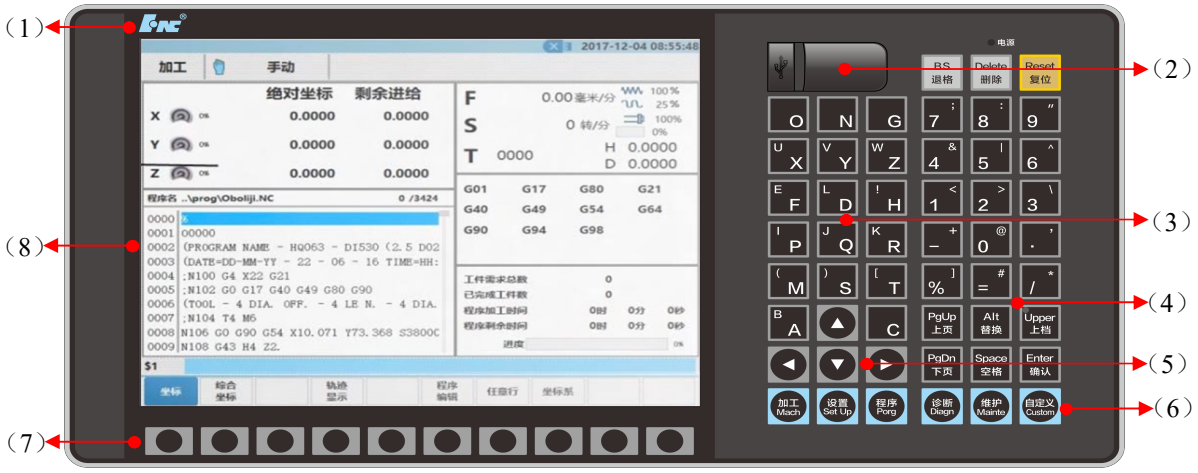
Machining setup interface

2 Operating Equipment

2.1 System Host Panel (NC Panel)

2.1.1 System Host Panel Zoning

HNC-808Di-M system 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-M CNC system is shown below

(1) ---Title bar

- Machining mode: Working mode of the system can switch among auto, single block, jog, incremental, reset, and emergency with corresponding keys on the control panel of the machine tool;
- System alarm message;
- Level 0 main menu name: Display currently activated main menu keys;

- Connection of USD flash disk and network:;
- System logo, time.

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

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

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

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

(6) ---Axis status display: 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

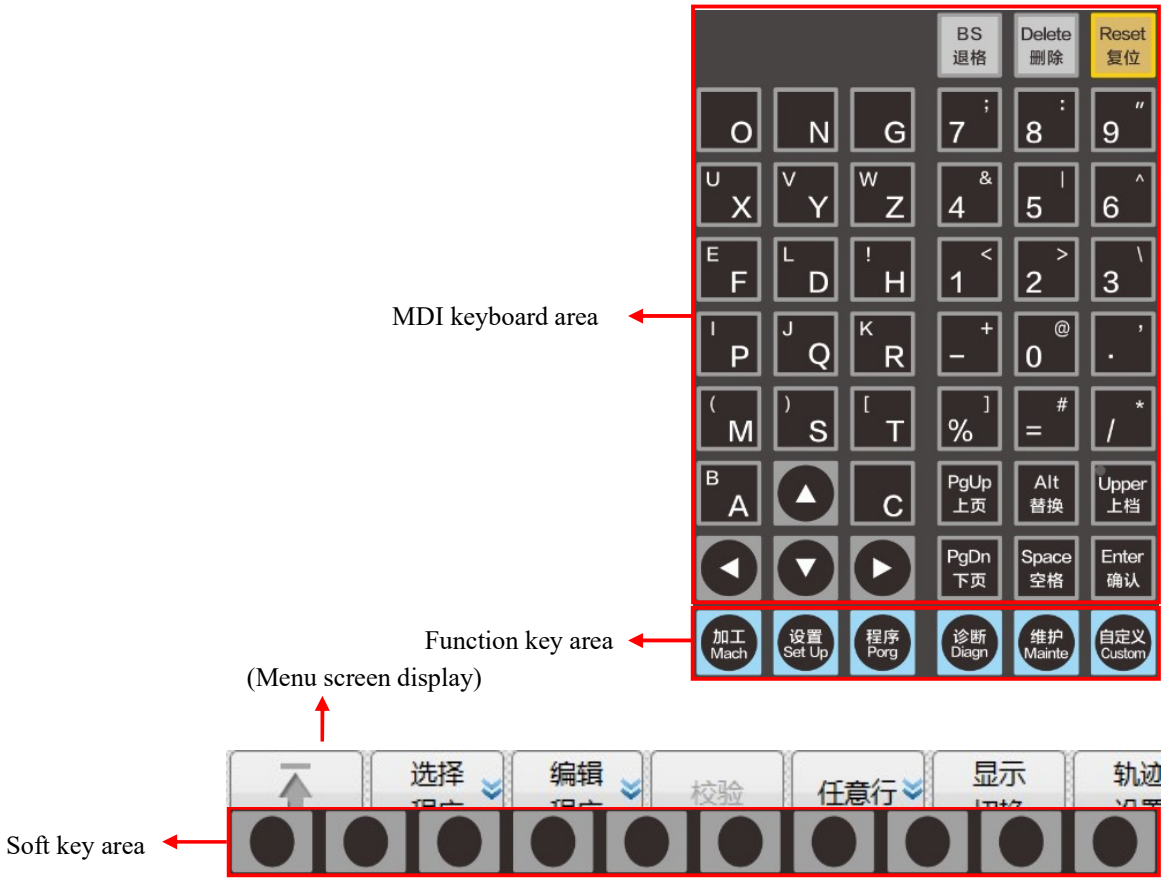
Input and edit command by this keyboard. Most keys have functions of upper and lower characters. Press "Shift" key and letter/number key simultaneously to input the upper letter/number.

Function key function

HNC-808Di-M 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).

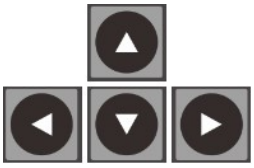
Soft key function

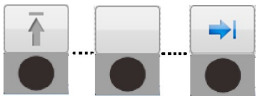
There are 10 soft keys below the screen, on which there are no fixed signs. The keys on the Left and right ends are to return to previous menu or continue the lower-level menu key, and others are function soft keys. All soft key functions correspond to menus displayed on the screen. The functions differ with change of menus (for specific functions, refer to chapter 3).



2.1.4 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.

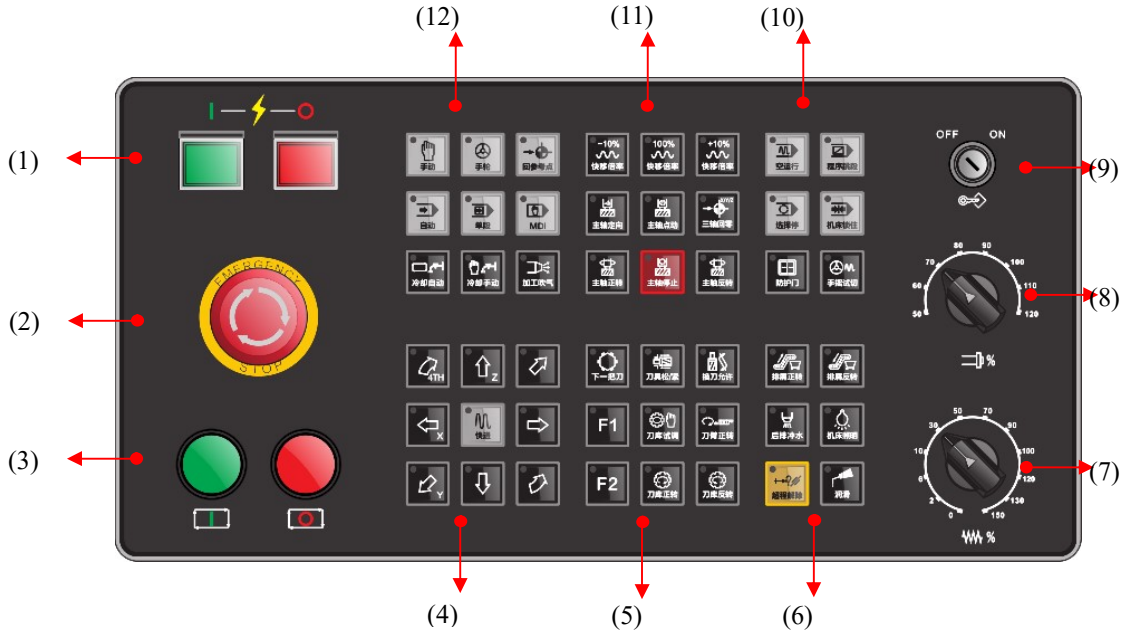
		
	Cursor shift key/ 「Cursor」	Control the cursor to move horizontally and vertically.
	Program name symbol key/ 「%」	Based on the lower character, program name symbol of subprogram
	Backspace key/ 「Backspace」	Delete characters forward, and so on.
	Delete key/ 「Delete」	Delete current program and delete characters backwards, 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, etc.) 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 conform input.
	Page up or page down key/ 「Page up or page	Switch previous and next pages in the same display interface.

	down」	
	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-M display screen, namely soft keys. In different function sets or levels, their 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 in blue, and it is in gray when the function set menu is in the level 1. In 10 soft keys, the rightmost key is to go to the next menu. The arrow is valid when it is in 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 text descriptions in the subsequent chapters, the key name will be replaced with the 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) ---Emergency stop button

(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 switch

(8)---Spindle override switch

(9)---Editing lock ON/OFF

(10)---Operation control key area

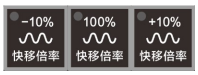
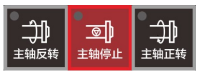
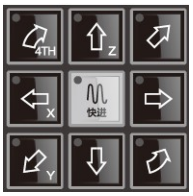
(11)---Rapid traverse override control key area

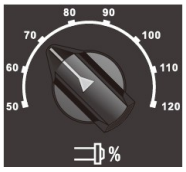
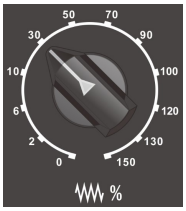
(12)---Working mode selection key area

2.2.2 Definition of Machine Operation Panel

This manual describes function and status of all keys based on standard PLC of HNC-808Di-M 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 the handwheel mode.	Handwheel
	Reference point return Working mode key /【Reset】	Select the reference point return mode key.	Reference point return
	Incremental Working mode key /【Incremental】	Select the incremental mode.	Incremental
	Jog Working mode key /【Jog】	Select the jog mode.	JOG
	MDI Working mode key /【MDI】	Select MDI mode.	MDI
	Auto Working mode key /【Auto】	Select the auto mode.	Auto
	Single block key ON/OFF key /【Single block】	1) Switching of block-by-block operation or continuous operation programs. 2) The indicator light lights up when the single block is valid.	Auto, MDI (Including single block)
	Handwheel ON/OFF key / [Handwheel]	1) Whether to enable the handwheel function. 2) When this function is enabled, the handwheel controls the tool to run as per the programmed path. While rotating the handwheel CW, the subsequent programs continues to run; while rotating the handwheel CCW, the executed programs retracts in the reverse direction.	Auto, MDI (Including single block)

	Block skip ON/OFF key /[Block skip]	1) When a program block is prefixed with "/", whether to skip the program block.	Automatic, MDI (Including single block)
	Optional stop ON/OFF key / [Optional stop]	1) When a program executes “M00” command, whether to stop; 2) If this key has been pressed before program execution (indicator light lights up), when the program executes “M00” command, the feed hold is performed, and then press cycle start to continue running the subsequent programs. If this key is not pressed, consistently run the program.	Auto, MDI (Including single block)
	Overtravel release key / [Overtravel release]	1) Cancel limits of machine tool; 2) Press and hold this key to release alarm and run the machine tool.	Handwheel, jog, incremental
	Cycle start key / [Cycle start]	Run program and MDI commands.	Auto, MDI (Including single block)
	Feed hold key / [Feed hold]	Suspend program and MDI commands.	Auto, MDI (Including single block)
	Rapid traverse speed override key / [Rapid traverse override]	Override of rapid traverse speed.	Handwheel, incremental, jog, reset, auto, MDI (including single block and handwheel)
	Spindle override key / [Spindle override]	Override of spindle speed.	
	Spindle control key / [Spindle CW/CCW rotation]	Control CW rotation, CCW rotation, and stop of spindle.	Handwheel, incremental, jog
	Manual control of axis feed key / [Axis feed]	1) Control movement and direction of axes under jog or incremental mode; 2) Select handwheel control axis under handwheel mode; 3) When an axis is pressed under jog mode, the axes run as per feedrate. When the Rapid traverse key is pressed at the same time, the axis run as per rapid traverse speed.	Handwheel, incremental, jog

	Machine control key / [Machine control]	Manual control of auxiliary actions of machine tool	Next tool, tool loosening and tightening, tool change allowed, cooling manual, magazine commissioning, CW rotation of tool arm, CW rotation of magazine. CCW rotation of magazine.	Jog
			Machine tool lighting, lubrication, rear row flush, machining blowing	Handwheel, incremental, jog, reset, auto, MDI (including single block and handwheel)
			Handwheel precutting, protective door	Auto
	Machine control extension key / [Machine control]	Manual control of auxiliary actions of machine tool.		Set by machine tool manufacturer 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]	In case of an emergency, the system and the machine tool immediately enter the halt state, and all outputs are turned off.		
	Spindle override key / [Spindle override]	Override of spindle speed.		Handwheel, incremental, jog, auto, MDI (including single block, handwheel simulation)
	Feedrate knob / [Feedrate]	Feedrate override.		Jog, auto, MDI, reset
	System power-on / [Power-on]	Control power-on of CNC device.		Handwheel, incremental, jog, reset, auto, MDI (including single block

	System power-off / [Power-off]	Control power-off of CNC device.	and handwheel)

Note:

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

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

In text descriptions in the subsequent chapters, key name will be replaced with key symbol.

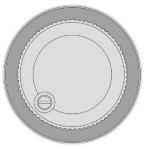
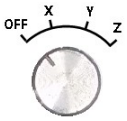
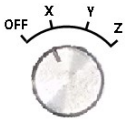
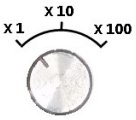
2.3 Handheld Unit

1. Handheld unit structure

Handheld unit consists of manual pulse generator, coordinate axis option switch, magnification option switch, pulse enable switch, and emergency stop switch. The structure diagram is shown below (specific appearance and shape should be subject to actual model of order)

2. Function definition of keys of handheld unit

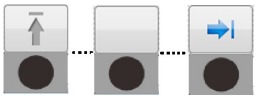
Key	Name/symbol	Functional description	Working mode at effective state

	Handwheel / [Handwheel]	Control movement of machine tool. (When handwheel function is valid, it can control the machine tool to move based on the programmed path).	Handwheel
	"MPG enable OFF" switch / [Enable OFF]	When the switch is turned to “OFF”, all switches and keys except the emergency stop button on the handheld unit are invalid.	Handwheel
	Axis option switch /[X]\[Y]\[Z]\[4TH]	When the switch is turned to the axis selection h except “OFF”, all switches and keys on the handheld unit are valid.	Handwheel
	Handwheel magnification switch / [Incremental magnification]	Tthe movement distance of the machine tool is 0.001mm/0.01mm/0.1mm as the handwheel rotates one graduation, or "Manual axis feed key" is pressed once.	Handwheel
	Emergency stop button / [Emergency stop]	When the handwheel is valid, in case of an emergency, the system and the machine tool immediately enter halt state, and all outputs are turned off.	Handwheel, incremental, jog, reset, auto, MDI

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 the function menu is selected by soft key.

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

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

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

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

7) Function sets of this system are at most a 4-level menu structure, and the function soft keys marked with "≫" on the right can be used to find lower-level menus. To return to the previous menu, use the "↑" key.

8) For configuration of soft keys of menus at all levels, the standard version of this system has set personalized display interface or menu according to actual needs. For special needs, users can also configure by themselves.

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 then input data or parameters.

10) When the man-machine dialog box does not exit, function sets cannot

be switched by function keys.

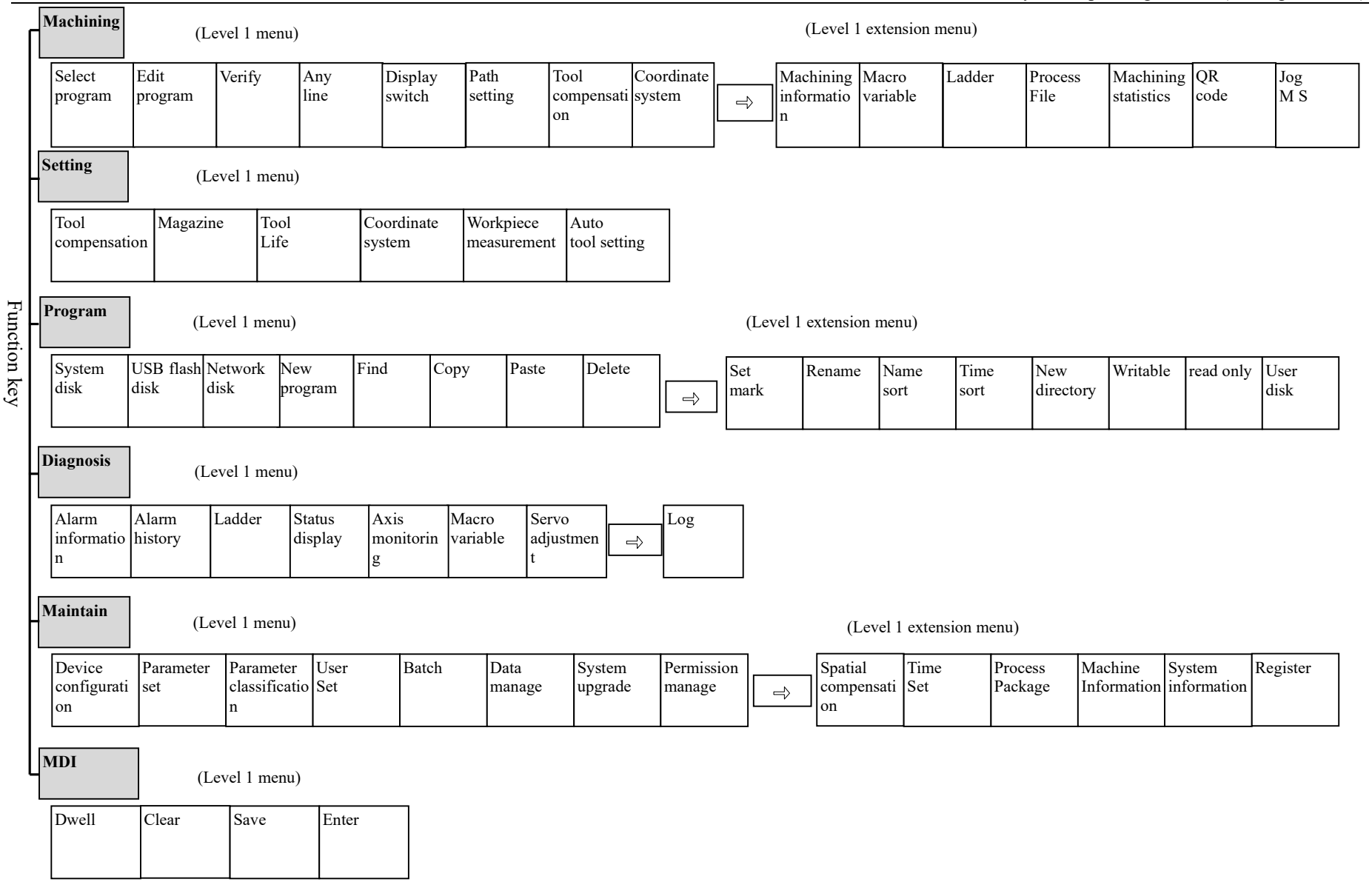
11) Exit mode of man-machine dialog box:

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

3.1.2 Function Menu Structure

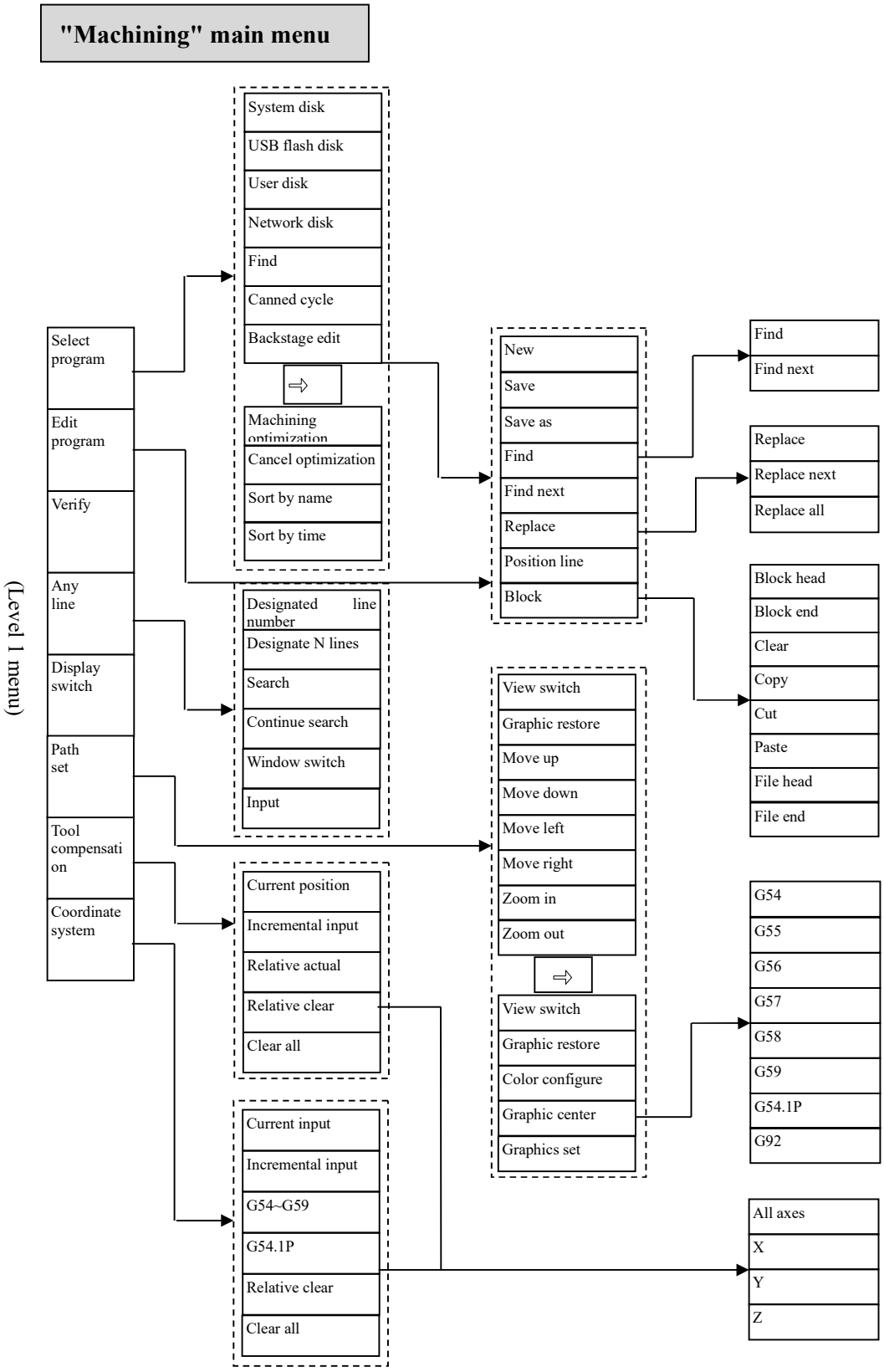
The menu tree is the basic structure diagram of the standard version of the system. The increase, decrease, sorting or position of the function menu will vary depending on the user's permission, parameter settings and the machine tool manufacturer. For details, refer to the specification provided by the machine tool manufacturer.

1) Level 1 menu of function sets

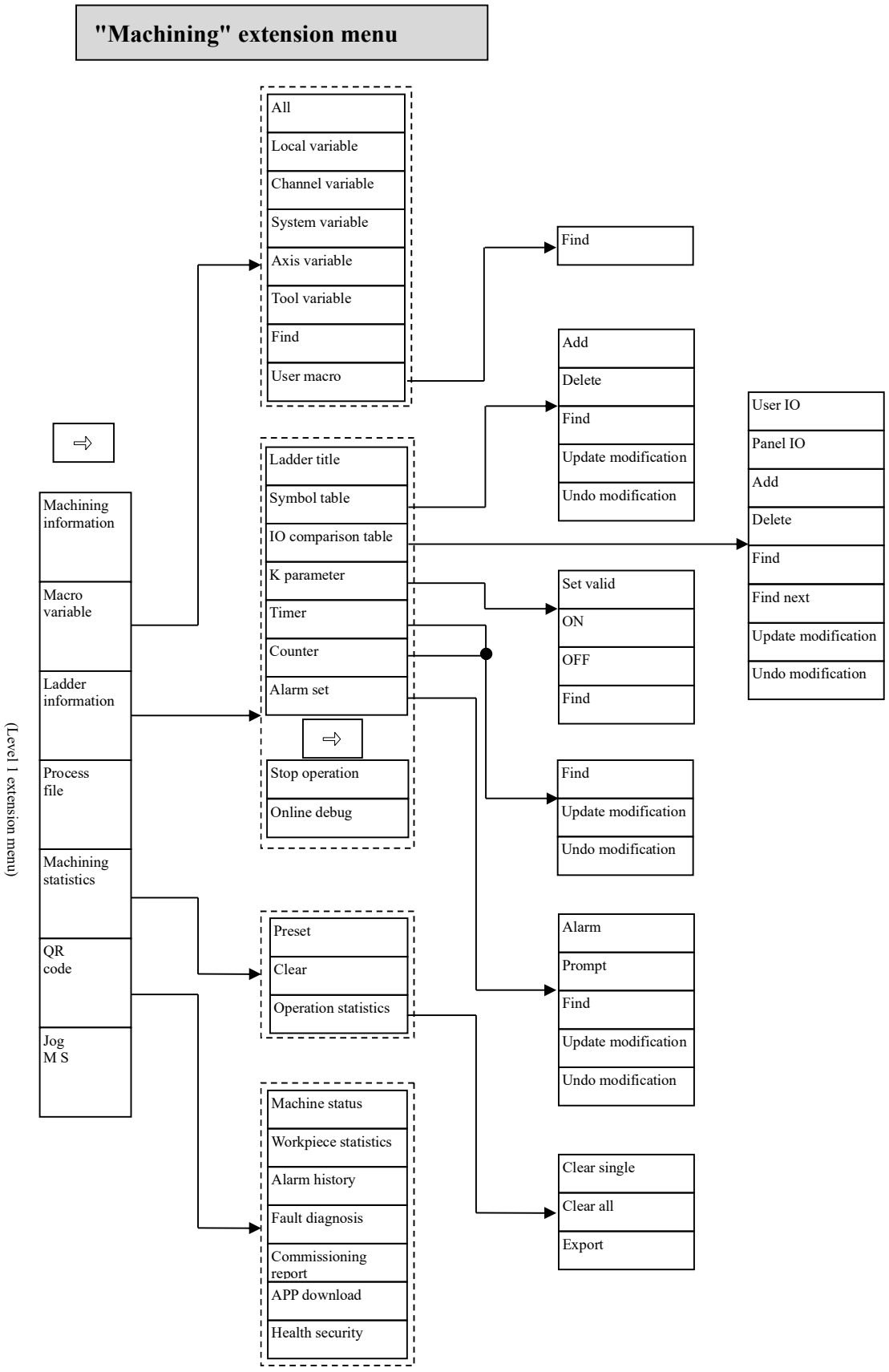

Function key
Machining
(Level 1 menu)
Select program
Edit program
Verify
Any line
Display switch
Path setting
Tool compensation
Coordinate system
⇨
(Level 1 extension menu)
Machining information
Macro variable
Ladder
Process File
Machining statistics
QR code
Jog M S
Setting
(Level 1 menu)
Tool compensation
Magazine
Tool Life
Coordinate system
Workpiece measurement
Auto tool setting
Program
(Level 1 menu)
System disk
USB flash disk
Network disk
New program
Find
Copy
Paste
Delete
⇨
(Level 1 extension menu)
Set mark
Rename
Name sort
Time sort
New directory
Writable
read only
User disk
Diagnosis
(Level 1 menu)
Alarm information
Alarm history
Ladder
Status display
Axis monitoring
Macro variable
Servo adjustment
⇨
Log
Maintain
(Level 1 menu)
Device configuration
Parameter set
Parameter classification
User Set
Batch
Data manage
System upgrade
Permission manage
⇨
(Level 1 extension menu)
Spatial compensation
Time Set
Process Package
Machine Information
System information
Register
MDI
(Level 1 menu)
Dwell
Clear
Save
Enter

2) Menu structure of "Machining" function set

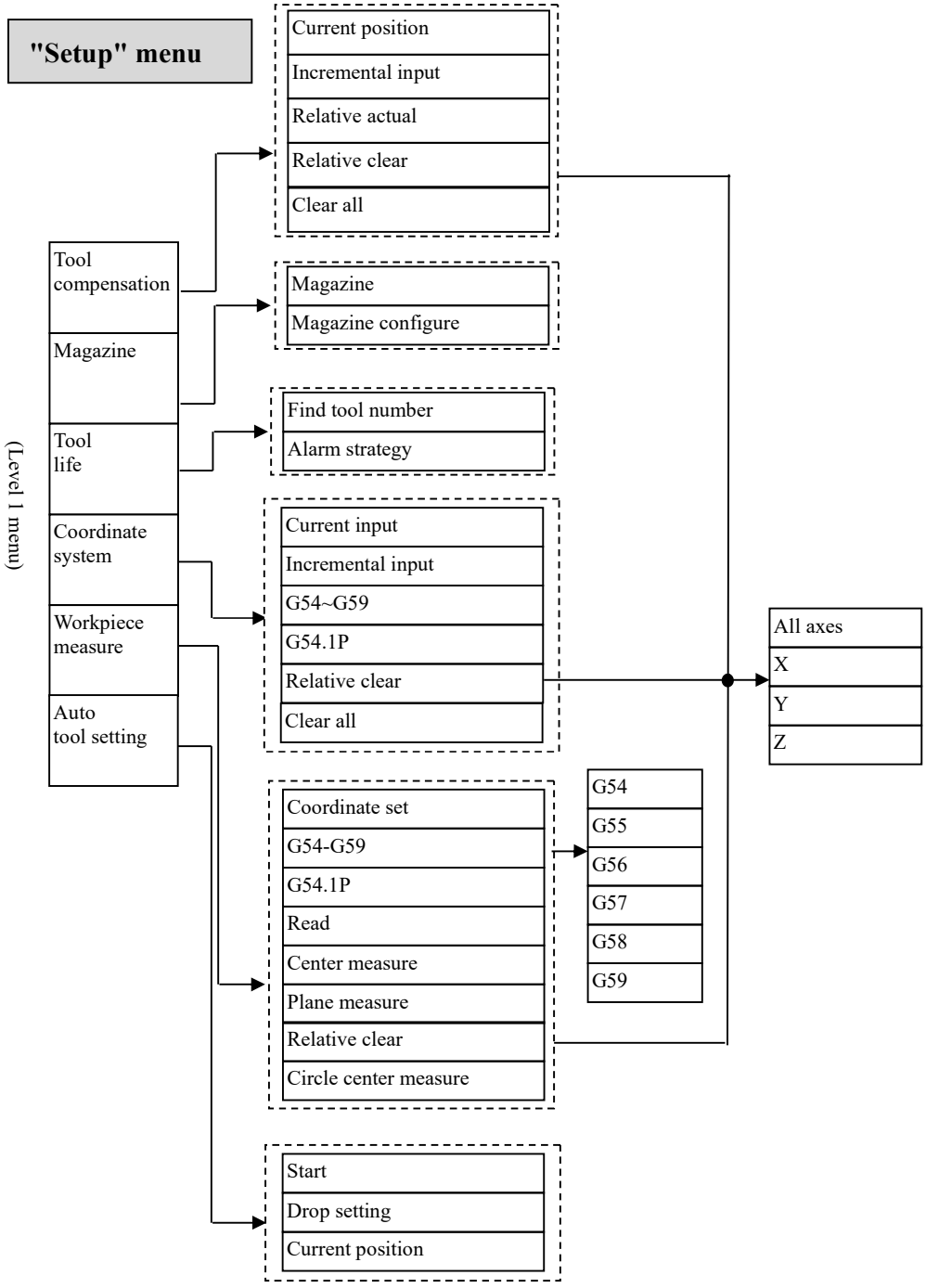
(1) "Machining" main menu



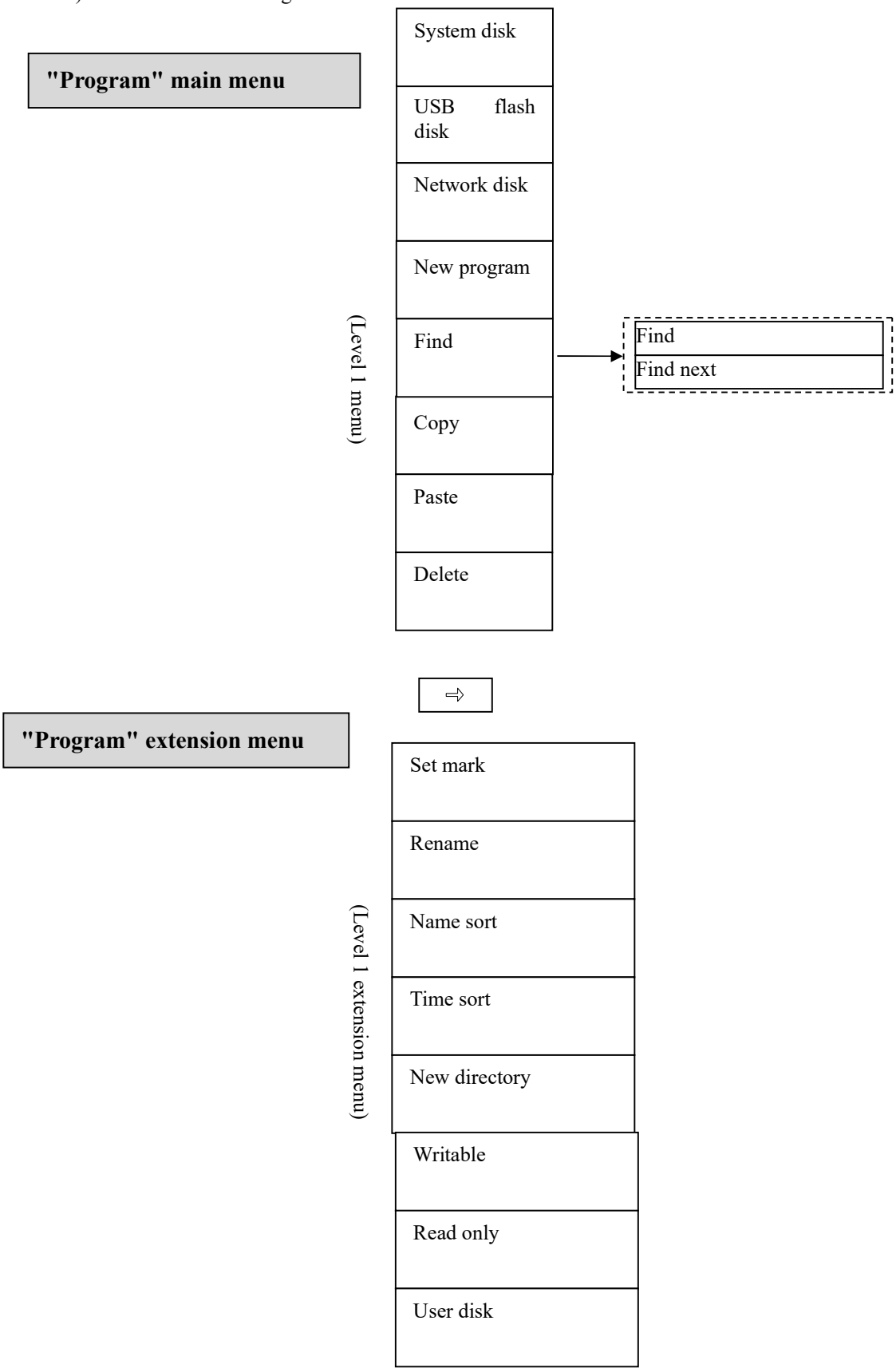
(2) "Machining" extension menu



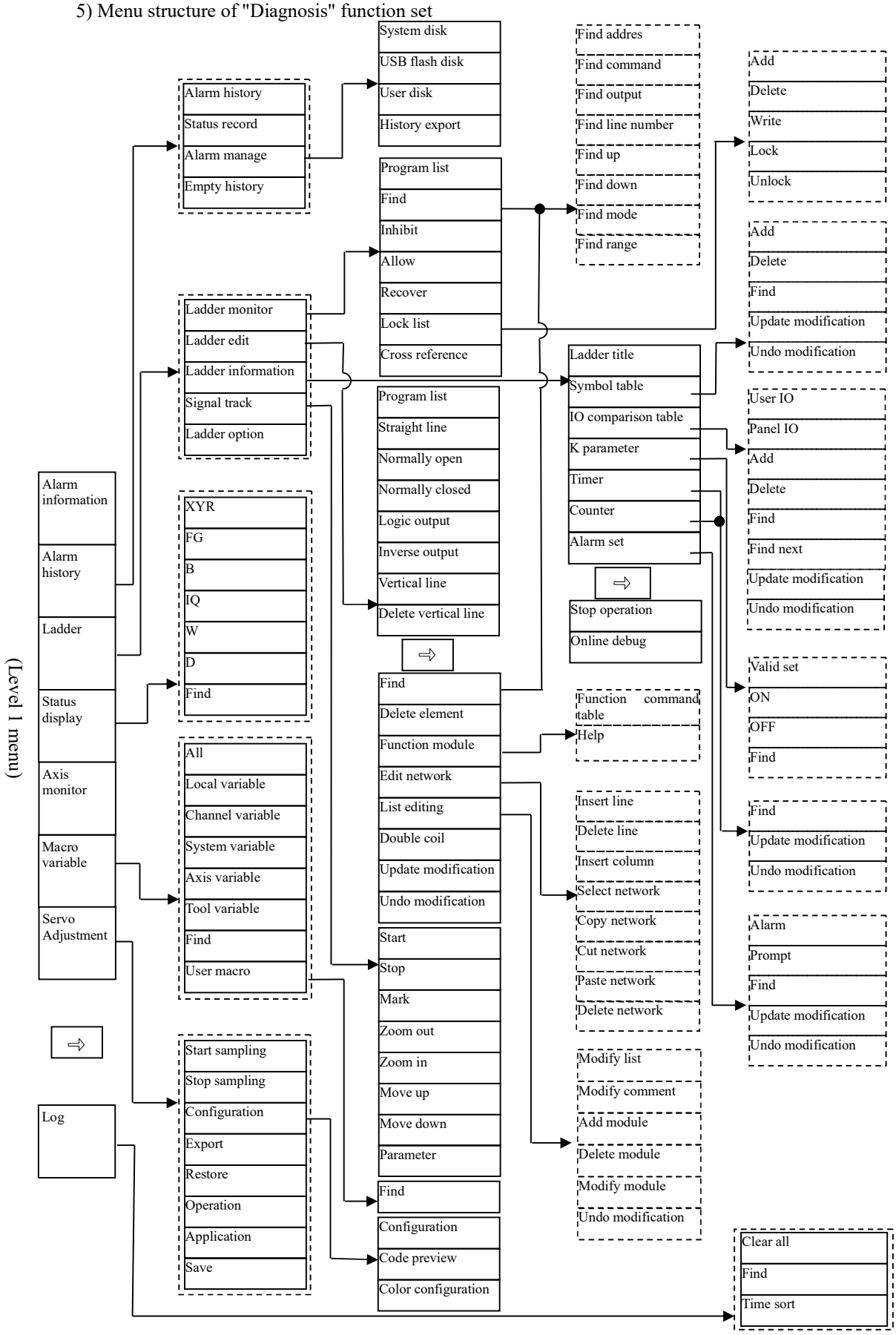
3) Menu structure of "Setup" function set



4) Menu structure of "Program" function set

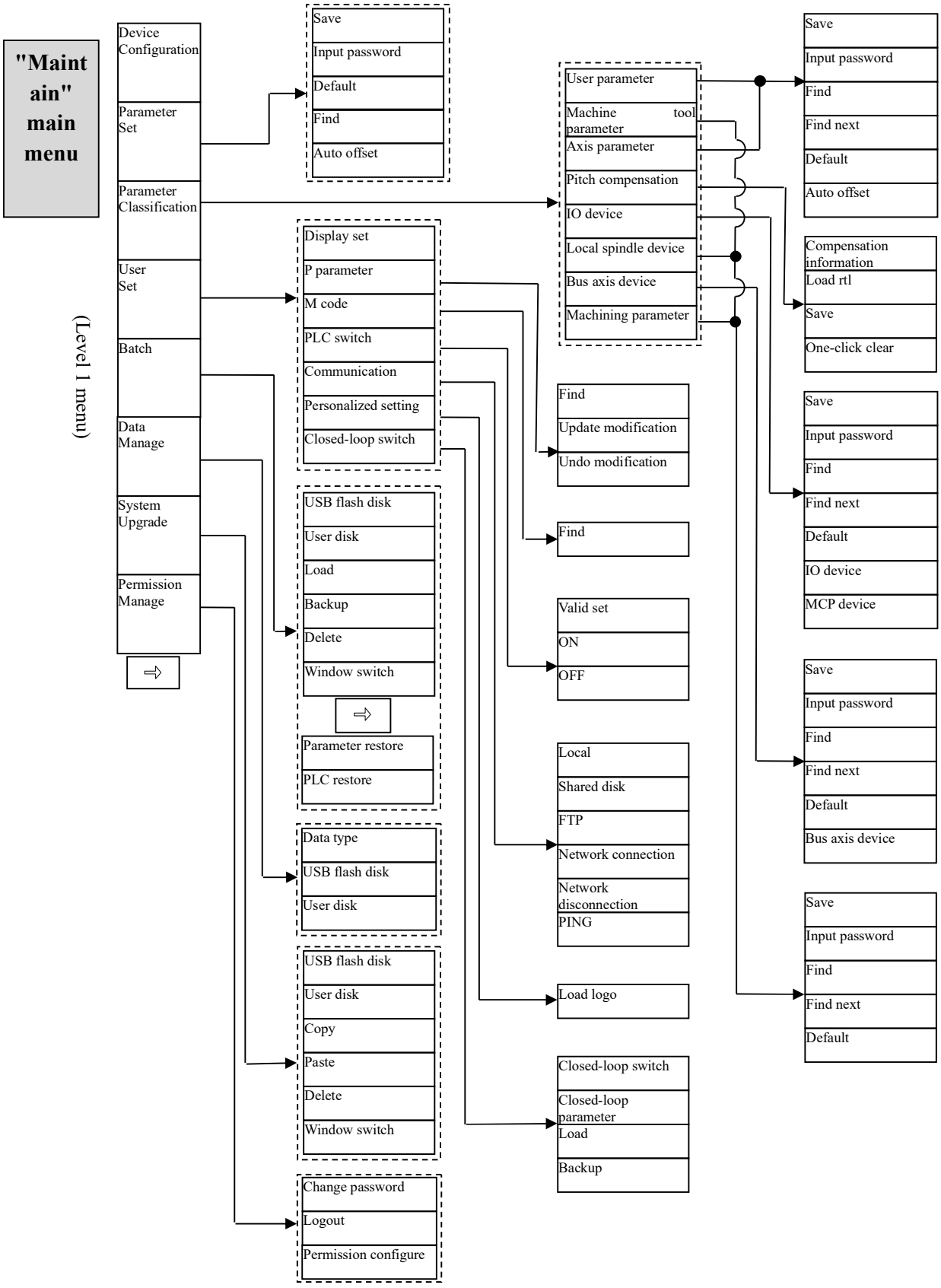


5) Menu structure of "Diagnosis" function set

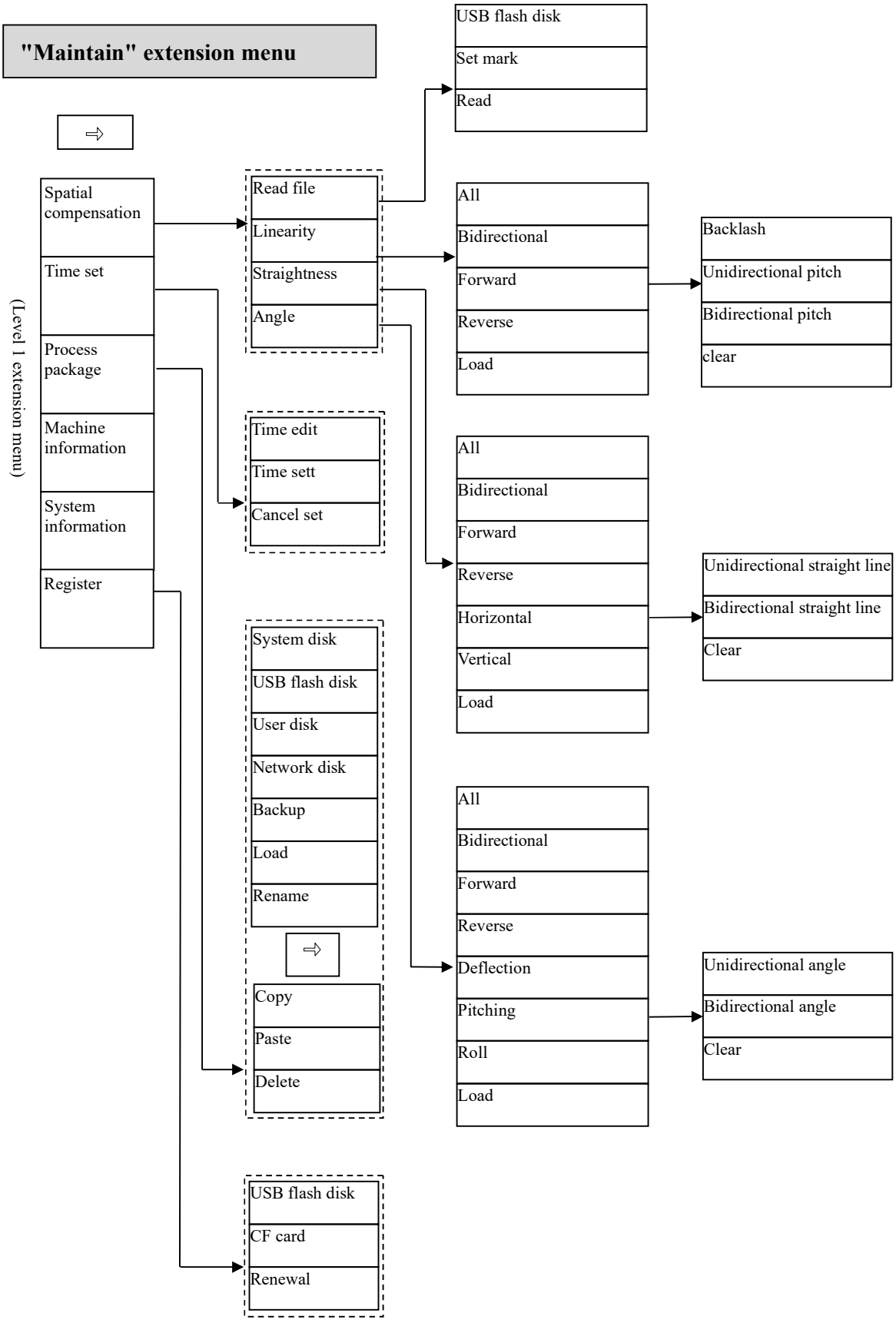


6) Menu structure of "Maintain" function set

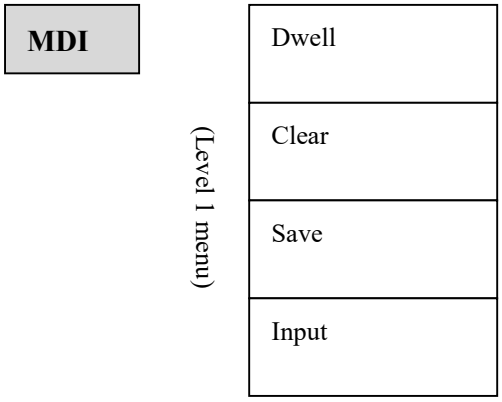
(1) Level 1 main menu of "Maintain"



(2) Level 1 extension menu of "Maintain"



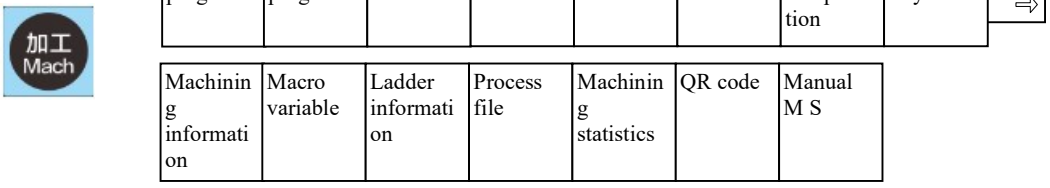
7) Menu structure of "User-defined (MDI)" function set



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 required for parts machining and is compatible with some functions of function set "Setup", "Program" and "Diagnosis", which greatly reduces interface switching. The operations that can be conducted under the function set include select machining program, select editing program, edit new program, verify program, tool setting (coordinate setup, tool compensation setup), any line, parameter configuration, coordinate display, graphic display, machining information display and user macro query, etx. Level 1 main menu and level 1 extension menu of "Machining" function set are shown below.



Select program: Select a program from the target disk (system disk, USB flash disk, user disk and network disk) and load it as the 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 the current machining program. A running program cannot be edited.

Verify: Enable this function under "Auto" or "Single block" working mode to quickly verify current loading program, and detect alarms of programming and grammar during program running.

Any line: Designate programs to run from any line under "Auto" working mode. Specific operations of this function are introduced in 7.2.3 of this manual.

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

Path setup: Used to set view switching, graphic restoration, path color, graphic center, graphic scaling setup of the programmed path.

Tool compensation: Compensation values such as tool length, length wear, radius and radius wear can be set under the sub-interface of this function. Jog tool compensation input mode and automatic measurement input mode are introduced in detail in Chapter 8 of this manual.

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

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

Macro variable: Local variable, channel variable, system variable, axis variable, tool variable and user macro variable of the system can be displayed and queried.

Ladder information: Query and modify title information, symbol table, IO comparison table, K parameter, timer, counter and alarm setup in the ladder diagram.

Process file: Store and view process card of machining program.

Perform statistics on the workpiece required, completed quantity, and accumulated quantities, as well as the current running, cumulative running, and current cutting time.

QR code: Generate QR code of machine tool status, workpiece statistics, alarm history, fault diagnosis, commissioning report, APP download and health security information for scanning and viewing using mobile APP.

Manual MS: When there is no command F (sub-feed)/S (spindle speed), set it using this function, and save it as system modal amount. If it has been set in the program, use the set value as the modal amount.

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--Machining mode, alarm message, prompt message and main function set display area

(2) Area--Coordinates and graphics display window: Coordinate and graphics path display area.

(3) Area---G code display area: Preview or display code 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.

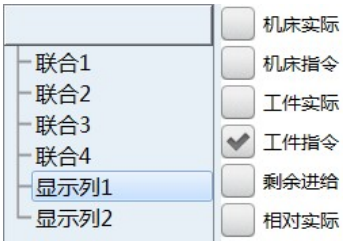
(8) Area--Machining information area: Display G modal, program progress and workpiece statistics during machining.

3.2.1.2 Switching of graphics and G code area display

显示
切换

For switching of graphics and G code areas (2) and (3) display , press 『Display switch』 soft key under the main menu interface of this function set, and 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 of coordinate graphics display



For big character display setup of coordinates and graphics display area (2), press 『User setup』 soft key under the "Maintain" function set interface to enter the lower-level menu, press 『Display setup』 soft key to enter the lower-level 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 "User setup" in 3.6.8).

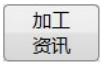
3.2.1.4 "Joint coordinate" display setup of coordinate graphics display area

For joint coordinate display setup of coordinates and graphics display area (2), press 『User setup』 soft key under the "Maintain" function set interface to enter the lower-level menu , press『Display setup』 soft key to enter the lower-level menu, select "Joint 1-4", and set 4 coordinate contents in the "Joint coordinate" interface (for details, refer to "User setup" in 3.6.8) .

3.2.1.5 Switch 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 "Machining" function extension menu interface to switch G modal, machining quantity and other information.

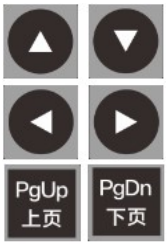
3.2.2 "Select Program" Sub-Interface

Main function of "Select program" sub-interface includes: select machining program, select editing program, edit program, and create new programs. The existing programs in system disk, USB flash disk and network disk can be selected.

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



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

preview program;

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

Note: If an 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 current machining program

- Press 『Select program』 to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys of 『System disk』,『USB flash disk』,『User disk』 and 『Network disk』, and enter the 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 underneath.
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the program file name;
- Press「Enter」to load the selected program as current machining program, and revert to the previous menu and interface. After that, parts can be processed.

3.2.2.3 Exit file directory

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

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

3.2.2.4 Edit current machining program in the backstage

Current machining program cannot be edited when the program runs, but it can be edited using backstage edit function under non-running status.

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

- Press 『Backstage edit』 soft key to enter the editing interface to edit 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 the 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 edit" sub-interface.

3.2.2.5 Edit other programs than current machining program in the backstage

- Press "Program select" soft key to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys of 『System disk』, 『USB flash disk』,『User disk』 and 『Network disk』, and 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 edit』 soft key to enter the editing interface 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 a program in the "Backstage edit" interface, the loading status of current machining program is not affected

3.2.2.6 Edit and creates new programs in the backstage

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

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 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 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 create and edit 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.

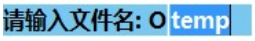
3.2.3.1 Edit current machining program

- Press 『Edit program』 soft key under the "Machining" function set and the cursor is in the editing area of current machining program for editing 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: 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 『New』 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 multiple program

blocks. It defines the initial block and the final block of multiple program blocks 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 status. There are 4 program editing status: Under "Machining" function set, edit and create program "Backstage edit" function; under "Machining" function set, edit current machining program of "Program edit" 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 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" Sub-interface

The "Verify" sub-interface is mainly used to quickly check programs, and at this time the machine tool does not run.

Verifying program is valid under auto mode and single block mode. After pressing 『Verify』 soft key, the working mode display changes from "Auto" to "Verify".

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

3.2.4.1 "Verify" runs

- Load programs under auto mode;
- After pressing 『Verify』 soft key, the working mode changes to "Verify";
- Press [Cycle start] to verify programs. (Feedrate override can control the verification speed)

3.2.4.2 Exit "Verify"

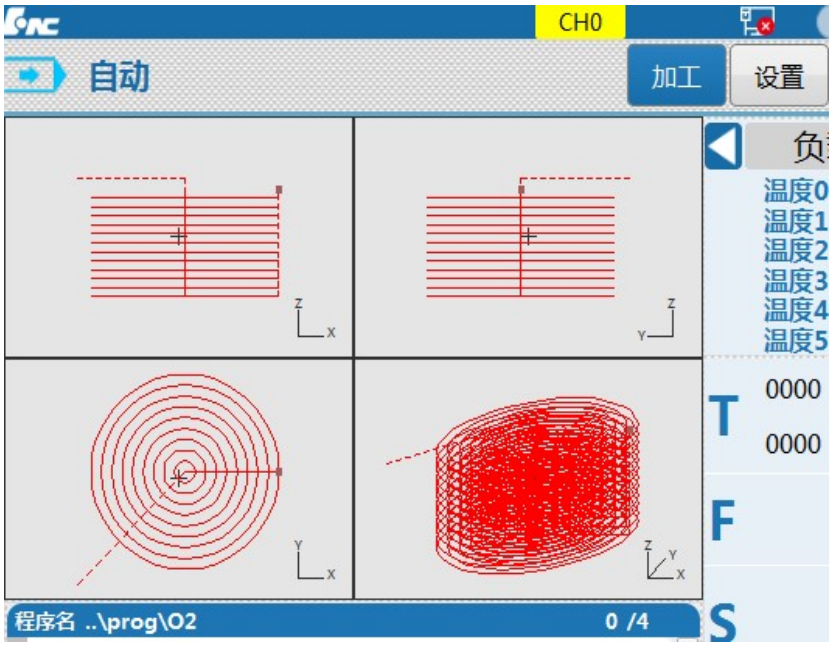
- After the program runs correctly, exit the verification status automatically;
- If verification is not conducted correctly or misoperation occurs, press 「Reset」 to exit the verification status.

3.2.5 "Path setup" Sub-interface

Select program machining path display interface through cyclic switching of "Display switch" soft key in the level 1 menu under machining function set. View switching, graphic restoration, color configuration, graphic center and graphic setup can be conducted in the 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.



3.3 "Setup" Function Set Interface and Basic Operation

3.3.1 "Setup" Function Set Interface and Function

"Setup" function set integrates operation functions of tool setting. Operations that can be conducted under the function set include tool setting operations such as coordinate system setup, tool compensation setup *, automatic tool measurement and workpiece measurement as well as tool life management.

This function set interface is recommended to be used as the main tool setting operation interface. "Machining" function set interface should be used as supplementary tool setting interface during machining (namely coordinate and tool compensation value modification).

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

Tool compensation	Magazine	Tool Life	Coordinate system	Workpiece measurement	Auto tool setting

- Tool compensation: Tool length compensation, radius compensation and radius wear compensation values can be set under this function sub-interface, and tool length compensation value can be set through operations under 『Auto tool setting』 soft key.

  Function and operation of "Tool compensation" under "Setup" function

set are the same as those of "Tool compensation" under "Machining" function set.

- Magazine: The magazine function interface displays tool number and machining mode. The magazine tool configuration function interface displays magazine type and magazine capacity.
- Tool life: This function can be used to set tool life management strategy.
- Coordinate system: This function can be used to set values of the workpiece coordinate system through direct input, current value input and increment input mode or save measured coordinate values through operations under 『Workpiece measurement』 soft key.

  Function and operation of "Coordinate system" under "Setup" function set are the same as those of "Coordinate system" under "Machining" function set.
- Workpiece measurement: This function is used for center measurement, plane measurement and circle center measurement of workpiece. The measurement results are stored in G54-G59 and extension coordinate system. For specific operating steps, please refer to "Tool setting and machining setup'.
- Auto tool setting: This function can be used for automatic tool length measurement under the three applications including single tool single workpiece, single tool multiple workpieces and multiple tools multiple workpieces. The measured values are stored in the tool compensation table. For specific operating steps, please refer to "Tool setting and machining setup'.

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 "Tool Compensation" Sub-interface

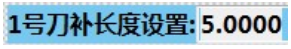
"Tool compensation" function is mainly used to set tool length compensation, length wear, radius compensation, and radius wear.

Tool compensation value can be inputted manually, or automatically through automatic tool measurement mode.

To simplify the operation, the system configures "Tool compensation" function under "Machining" set and "Setup" set with the same function and operation. This section is introduced with "Tool compensation" submenu under "Setup" set.

3.3.2.1 Direct input mode of tool length compensation

- Press 『Tool compensation』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Direction」 or 「PgUp/PgDn」 to move the cursor to tool length compensation of corresponding tool number;
- Press 「Enter」 to confirm, activate input status, and the input box gives a prompt message Input tool length compensation value of the selected tool number.
- 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 "Take effect from next tool changing or rerun", and exit the input status.

3.3.2.2 Input mode of tool length compensation of current position

Input mode of tool length compensation of current position is to take the actual machine coordinates when the tool nose touches the tool setting face of the workpiece as the tool length compensation value.

- Press 『Tool compensation』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Direction」 or 「PgUp/PgDn」 to move the cursor to tool length compensation of corresponding tool number;
- Under jog mode, move the tool nose to touch the tool setting face of the workpiece, press 『Current position』 to write actual machine position in tool length compensation value automatically.

3.3.2.3 Incremental input of tool length compensation

When there is tool length compensation value in the tool compensation table, increase or decrease tool length compensation using increment input mode.

- Press 『Tool compensation』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Direction」 or 「PgUp/PgDn」 to move the cursor to tool length compensation of corresponding tool number;
- Press 『Incremental』 to activate the input box;

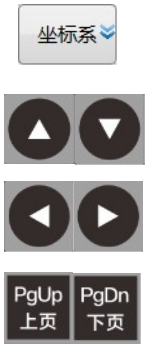
- When a positive value is input, the tool length compensation is increased; When a negative value is input, the tool length compensation is reduces.
- Press 「Enter」 to confirm and complete modification of tool length compensation.

3.3.2.4 Relative actual input of tool length compensation

When the relative movement distance of tool is used as tool length compensation, input tool length compensation using relative actual input mode.

- Press 『Tool compensation』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Direction」 or 「PgUp/PgDn」 to move the cursor to tool length compensation of corresponding tool number;
- Before input, press 『Relative clear』 soft key to clear relative coordinate value of Z axis;
- Move the tool in Z direction under jog mode, the movement distance is displayed on the relative actual coordinates of Z axis;
- Press 『Relative actual』 soft key to input relative actual coordinates of Z axis to tool length compensation.

3.3.3 "Coordinate System" Sub-interface



For ease of operation, the system configures "Coordinate system" function both under "Machining" set and "Setup" set with the same function and operation. This section is introduced with "Coordinate system" submenu under "Setup" set.

The coordinate values 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 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.

3.3.3.1 Direct input of coordinate value

This function can be used to input known workpiece zero coordinates of 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 「Up and down cursors」 to select the coordinate system of area 1 or 3;
- Press 「Left and right cursors」 or 「PgUp/PgDn」 to select and set coordinate system;
- Press 「Enter」 to activate the input box;
- Input the workpiece zero coordinate in the input box;
- Press 「Enter」 to validate input, and exit the dialog box.
- To abandon the input, press 「Reset」to abandon the input and exit the input box

3.3.3.2 Current value input

After tool setting is completed and the tool moves to the workpiece zero, this function can be used to set the machine 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 the 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 as the the selected workpiece zero;
- Press 「N」 or 「Reset」 to abandon setup and exit the input box.

3.3.3.3 Incremental value input

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』 to activate the input box;
- Input the incremental value of the coordinates in the input box;
- Press 「Enter」 to confirm and exit the input box;
- To abandon input, press 「Reset」to invalidate the input, and exit the input box

3.3.4 "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 revolution.

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.

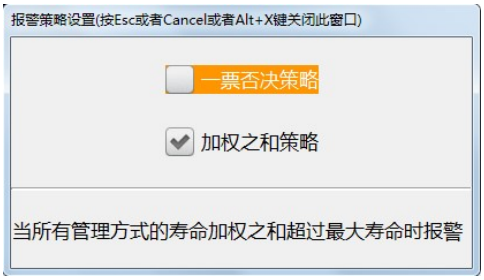
3.3.4.1 Tool life benchmark setup

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

3.3.4.2 Tool life alarm strategy setup

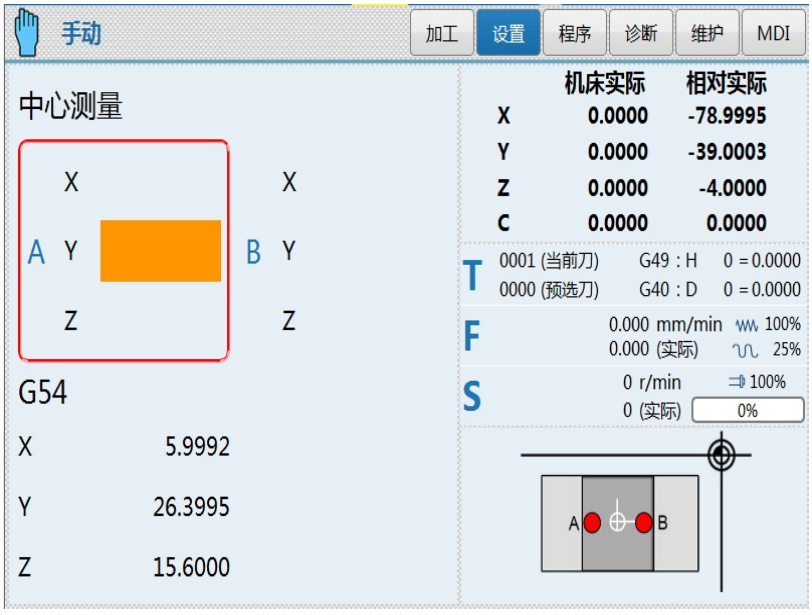
- Press『Tool life』soft key under the "setup" interface to enter the sub-interface;

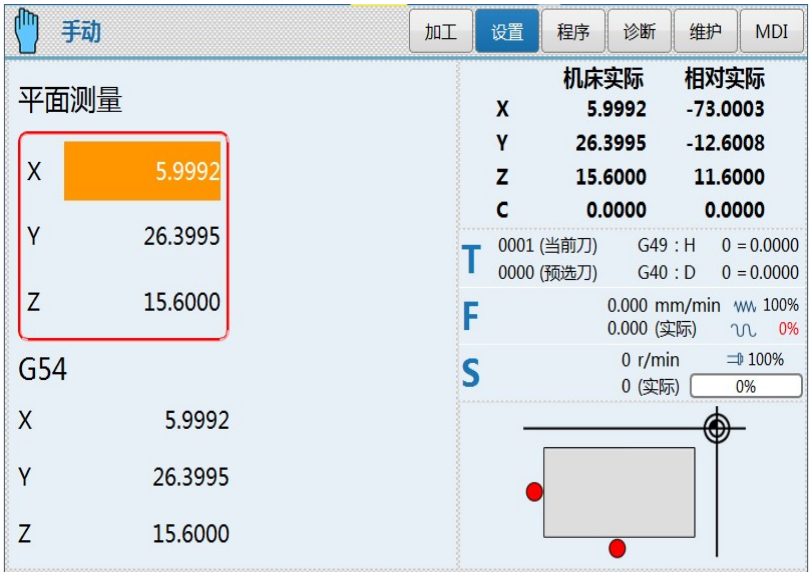
- Press 『Alarm strategy』 soft key, and the strategy selection window will pop up (as shown below)
- Press 「Cursor」 to select the alarm strategy;
- Press 「Enter」 to confirm and exit the selection window.

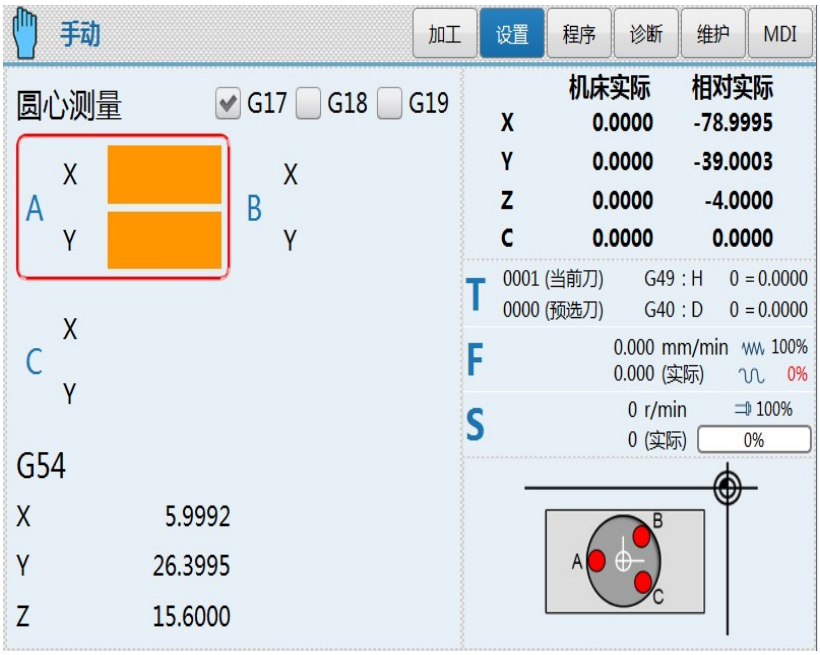


3.3.5 "Workpiece Measurement" Sub-interface

Press 『Workpiece measurement』 under the extension menu of "Setup" function set to enter the "Workpiece measurement" default sub-interface (as shown below), namely "Plane measurement" sub-interface.







The Standard configuration supports three manual measurement functions: center measurement, plane measurement and circle center measurement, under the "Workpiece measurement" interface. When some appropriate measuring instruments and softwares are configured, the plane calibration, single point measurement, bevel edge measurement, rectangle measurement and special-shaped circle measurement can be realized.

3.3.6 "Automatic Tool Setting" Sub-interface

Press 『Auto tool setting』 under the extension menu of "Setup" function set to enter the "Auto tool setting" default sub-interface (as shown below).

The "Automatic tool setting" sub-interface supports three automatic measurement functions of tool length including single tool single workpiece measurement, single tool multiple workpieces measurement and multiple tools multiple workpieces measurement. To satisfy thethree different types of applications, the storage location of measured tool length value is different. For single tool single workpiece measurement, the tool length is saved in the selected workpiece coordinate system, and Z drop value is filled in external zero offset. For single tool multiple workpieces measurement, the tool length is saved in external zero offset, and Z drop value is filled in the selected workpiece coordinate system. For Multiple tools multiple workpieces measurement, the tool length value is saved in the tool compensation table, and Z drop value is filled in the selected workpiece coordinate system.

3.3.7 "Manual MS" Sub-interface

Press 『Manual MS』 under the extension menu of "Setup" function set to enter the "Manual MS" setup window (as shown below).

The "Manual MS" function interface supports manual definition of F/S information. When F/S is not defined in the program, F/S value can be defined in the interface and saved as modal. When F/S has been defined in the program, the defined value should be saved as modal.

3.4 "Program" Function Set Interface and Basic Operation

3.4.1 "Program" Function Set Interface and Function

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

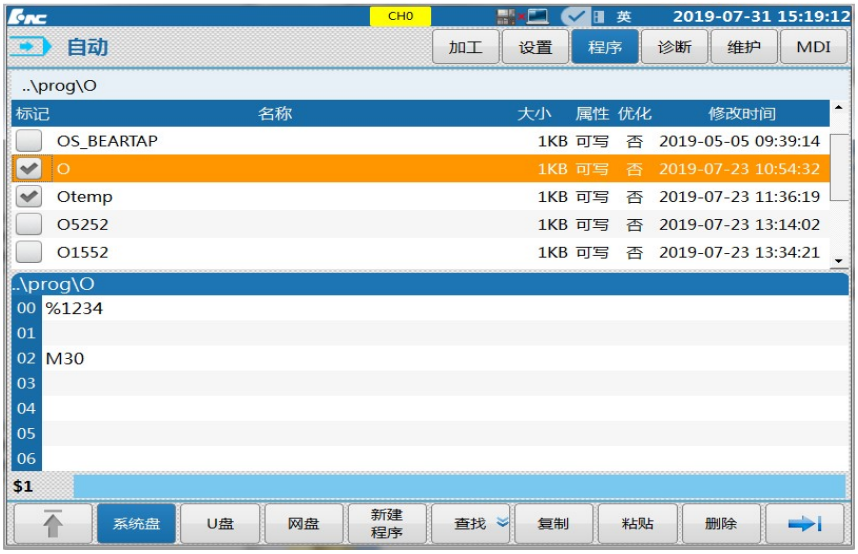
System Disk	USB flash disk	Network disk	Create a new program	Find	Copy	Paste	Delete ⇨
Set mark	Rename	Name sort	Time sort	New directory	Writable	read only	User disk

- 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.
- Set mark: Mark programs in the source disk in order to copy or paste multiple programs.
- Rename: Rename programs in the source disk.
- 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.



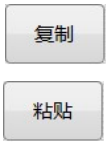
3.4.2 Management Of Files in System Disk, USB Flash Disk and Network Disk

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 search 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 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 a 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 『Create』 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, and a prompt message Save succeeds will be given.

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 the input and the original program is renamed as a new program.

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 requested.

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 and the dialog box gives a prompt message: "Please input a directory name";
- Input a directory name such as HCNC, and 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 tool commissioning. 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 monitor	Macro variable	Servo adjustment	⇨

Self check	Log	Fault video	Screw load	QR code

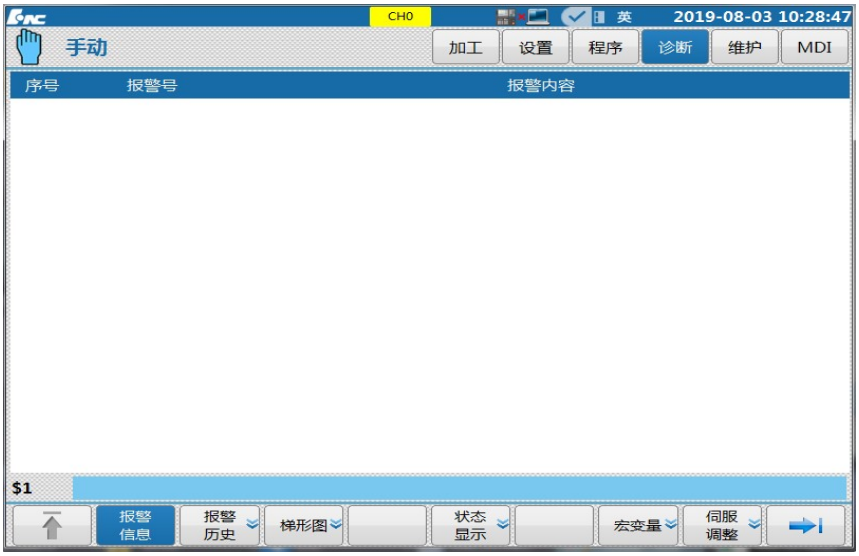
- Alarm message: Display current alarm message
- Alarm history: Save information of recent alarms and export historical alarms to USB flash disk and user disk using "History export" soft key in the lower menu.
- Ladder: This interface is used for monitoring and editing of PLC as well as the PLC module setting and query.
- Status display: Display and view the status of all registers.
- Axis monitor: 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: Optimize servo parameters to improve the positioning and repeatability accuracy in the movement process the response speed of machine tool, thus improving machining quality and efficiency.
- Self-check: Diagnose the change of health index during operation of machine tool; used for detecting assembling and commissioning

consistency.

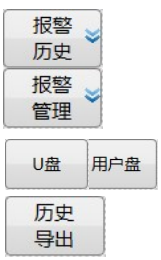
- Log: Record all contents such as system fault diagnosis, machining information, file modification, panel operation and system event during operation of machine tool. This function can be regarded as the black box of CNC system.
- Fault video: Record fault related data 10s before the fault occurs. The 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, upload them to cloud data center and realize full life circle management of machine tool.

Intelligent functions and interfaces such as "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

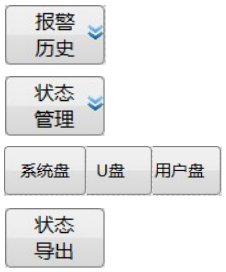


- Press 『Alarm history』 soft key under the "Diagnosis" function set default interface;
- Press 『Alarm history』 soft key under the 『Alarm history』

sub-interface;

- Press 『Alarm manage』 soft key;
- Select 『USB flash disk』 and 『User disk』 soft keys;
- Press 『History export』 soft key to export corresponding information to the selected disk.

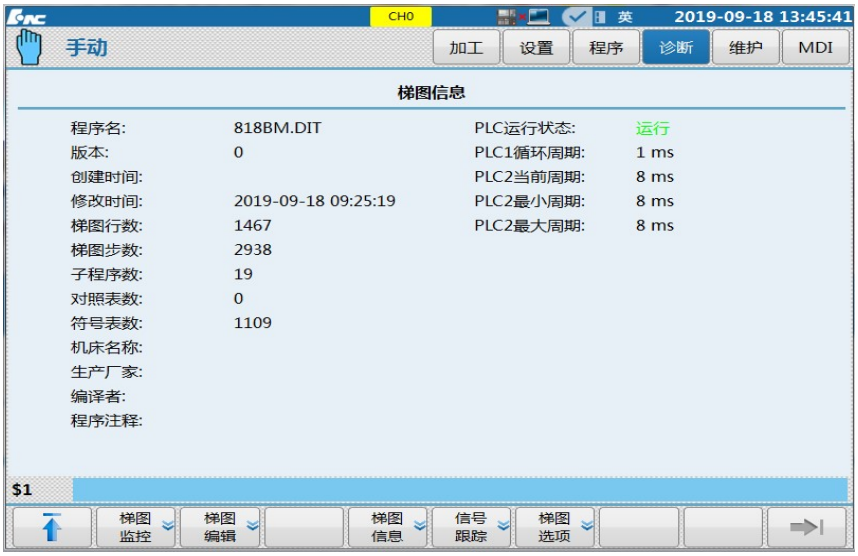
3.5.3 Status Record Export



- 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 3.6.5.

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, negation 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, search, 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.

梯图信息

- Press 『Ladder』 soft key under the "Diagnosis" default interface;
- Press 『Ladder information』 soft key to enter the sub-interface and the lower-level menu (as shown below).

1. Ladder diagram title

梯图标题

Record line number, 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. While configuring different machine tools, PLC needs not to be modified for the change of IO point. Different XY registers correspond to IO registers and IO point location remains unchanged in PLC.

4. K parameter

Function and P parameter are the same, but 0 or 1 point location can be modified directly in this interface.

5. Timer and counter

Used to monitor status of timer and counter in monitoring program.

6. Alarm setup

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

3.5.4.4 Ladder signal tracking

This function can be used to track the value change of registers through sampling.

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

For example, sample “Run allowed R10.0”and a blinking point R72.6, fill in the two registers in the corresponding box, click to start sampling, and 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. For example, the figure below is a reduction of the sample of the above.

Press Move up and Move down to move the sample where the cursor is located up or down one line. For example, move R72.6 down one 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 to 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-variable』 soft key to display status 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 set	Parameter classification	User set	Batch	Data manage	System upgrade	Permission manage	⇨

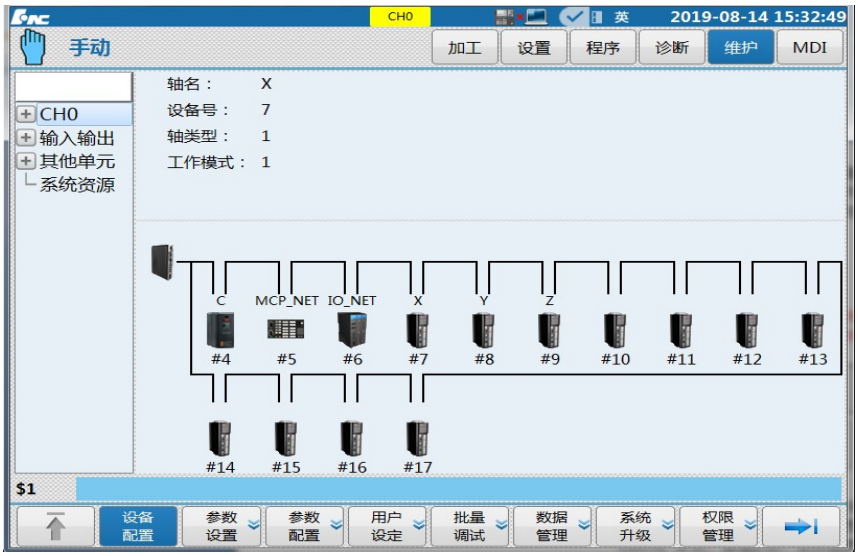
Spatial compensation	Time set	Process package	Machine information	System information	Register

- Device configuration: View number of drive, I/O, panel and other hardware devices, and connection sequence of bus;
- Parameter set: This interface includes all system parameters including NC parameter, machine user parameter, channel parameter, coordinate axis parameter, error compensation parameter, device interface parameter and data table;
  - ✧ NC parameter: Common parameters of CNC system.
  - ✧ Machine user parameter: Common parameters relating to machine tool and users.
  - ✧ Channel parameter: Common parameters of channels.
  - ✧ Axis parameter: Related parameters of logical axis (electronic gear ratio, acceleration/deceleration time constant, and so on)
  - ✧ Error compensation parameter: Set related parameters of error compensation of logical axis (such as backlash compensation type of axis 0)
  - ✧ Device interface: Parameters of interface of connection between physical device and CNC system (such as device type and device ID)
  - ✧ Data table: Data table storing corresponding compensation values of error compensation parameters.
- Parameter configuration: the common user parameters in this interface, which are classified according to application type for ease of setting;

Parameters under this interface include user parameter, axis parameter, pitch compensation, IO device, local spindle device, bus axis device and function parameter;

- User set: Setup relating to user application. This interface includes display setup, P parameter, M code, PLC switch, communication setup and personalized setup;
- Batch: Loading and backup of PLC, parameter, canned cycle, G code and other files;
- Data manage: Loading and backup of various types of data;
- System upgrade: System upgrade and backup;
- Permission manage: To set the administrators of different permissions. Different permissions have certain influence on the structure of interfaces and menus;
- Spatial compensation: Spatial error compensation setup;
- Time set: System time setup;
- Process package: Load or back up process package file;
- Machine information: Edit or display machine tool information;
- System information: Display information of the system;
- Register: 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 classification』 and 『Parameter set』 to select parameter set;
- Press 「Left and right cursors」 to move the cursor and select parameter lower classification column or parameter setup column

2) Parameter input activation

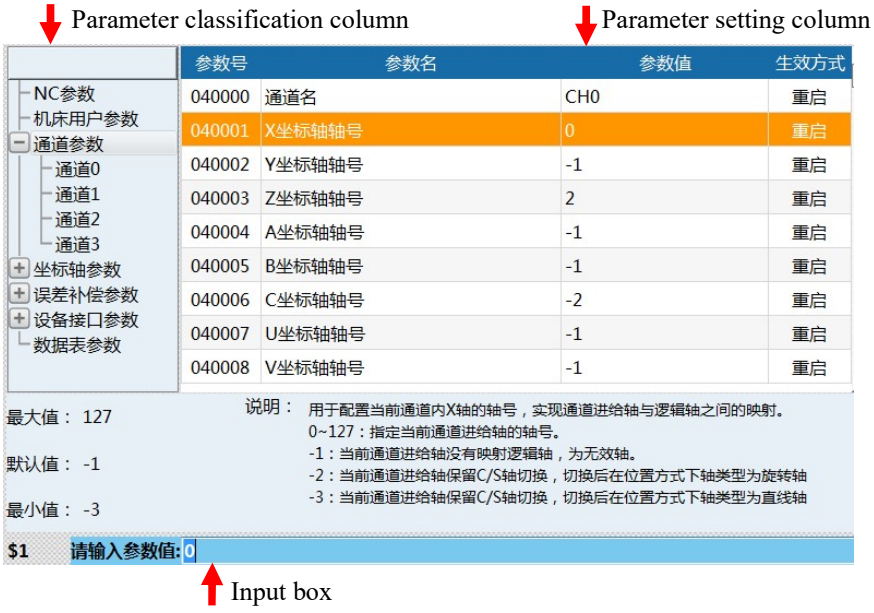
- When the cursor moves to parameter classification column, press 「Enter」 to open categories of the selected parameter
- When the cursor moves to parameter setup column, press 「Enter」 to activate the input box

3) Parameter input

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

4) Exit input

- To abandon the input after input box is activated, press 「Reset」 to exit the input, and the original value is maintained.



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) Immediate validation of parameters

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

2) Save to take effect parameters

- After parameters are inputted in the input box, press 「Enter」 to confirm and a prompt message "Setup succeeds, save to take effect" will be given;
- Press『Save』or『↑』soft key and the input box gives a prompt message "Whether to save the modification? (Y/N)”；
- Press 「Y」 or 「Enter」 to confirm, a prompt message "Setup succeeds" will be given and parameters take effect;
- Press 「N」 to abandon save and restore the original value.

3) Reset to take effect parameters

- After parameters are inputted in the input box, press 「Enter」 to confirm and a prompt message "Setup succeeds, reset to take effect" will be given;
- Press『Save』or『↑』soft key and the input box gives a prompt message "Whether to save the modification? (Y/N)”；
- Press 「Y」 or 「Enter」 to confirm, a prompt message "Save parameters successfully, please press Reset" will be given;
- Press 「Reset」 to confirm, a prompt message "Reset succeeds" will be given and parameters take effect;
- Press 「N」 to abandon save and restore the original value.

4) Restart to take effect parameters

- After parameters are inputted in the input box, press 「Enter」 to confirm and a prompt message "Setup succeeds, reset to take effect" will be given;
- Press『Save』or『↑』soft key and the input box gives a prompt message "Whether to save the modification? (Y/N)”；
- Press 「Y」 or 「Enter」 to confirm, a prompt message "Save parameters successfully, please power off and restart" will be given;
- After the system is powered off, restart it to validate parameters.

Note:

- Parameter setup and modification are limited, input a password of corresponding permission while setting and modifying parameters.

3.6.4 "Parameter Classification" Sub-interface

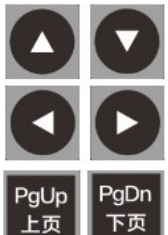
The "Parameter classification" sub-interface can be used to set device interface parameters such as axis, MCP and IO as well as user parameter, machine parameter and machining parameter. Pitch compensation function is also under this function set.

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 and import, refer to 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.

3.6.4.1 Direct input of parameter classification 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 the machine tool manufacturer permission password "HOD";
- Press 「Enter」 to confirm the input;



- Press 「Cursor」 or 「PgUp/PgDn」 to select the value on the right of parameter name;\
- Press 「Enter」 to activate the value input status;
- Input the value to be set such as “1”;
- Press 「Enter」 to confirm, and a prompt message "Setup succeeds, save to take effect" will be given;
- Press 『Save』 soft key and a prompt message "Save the modification or not" will be given;
- Press 「Y」, a prompt message "Save successfully" will be given and parameters will be validated immediately.

3.6.4.2 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";
- Press 「Cursor」 to move the cursor to the value setup area of "Starting point", "Compensation interval", "Backlash compensation type", "Compensation points", "Initial number of data table" and "Backlash value";
- Press 「Enter」 to activate the value input status;
- Input the corresponding value in the input box (initial number of data table is often 71000)
- Press 「Enter」 to confirm the input;
- Press 『Save』 soft key and a prompt message "Parameters have been saved" will be given.

3.6.5 Classification and Switching of Management Permission

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 with 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. Permission login password is HIG.

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

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

"Operator": Machining operation. It has permissions of editing tool compensation data and selecting program. There is no password need for this permission.

Operation function \ Permission type		System administrator	System 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 error 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 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
Process package		Yes	Yes	Yes	No	No
Register		Yes	Yes	Yes	No	No
Alarm history		Yes	Yes	Yes	Yes	No
Ladder diagram	Ladder monitoring	Yes	Yes	Yes	Yes	No
	Ladder editing	Yes	Yes	Yes	No	No
	Ladder 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
Clear log		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 setup	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 the permission of "Operator" or "Workshop administrator" as the default permission through parameter 000359 in NC parameter table, and other permissions can be switched through entering a password after startup. Permission switching is shown as below:

- Press 〖Maintain〗 function key to enter the default interface of the "Maintain" function set
- Press 〖Permission manage〗 soft key to enter the "Permission manage"

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;
- If the machine tool manufacturer permission is selected, please enter: “HOD”;
- Press 「Enter」 to confirm the input. Then permission is modified successfully.

3.6.6 System Upgrade

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

After permission setup, 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 on the upper part of the interface
- Press 「Left and right cursors」 to select the required items; (BTF is to upgrade all items)
- Press 「Enter」 to confirm;
- For backup, select "Backup" (the default backup target disk is the user disk);
- Press 『Window switch』 soft key to select the upgrade package file

source selection window on the lower part of the interface (the default upgrade source disk is the 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 the upgrade

Note:

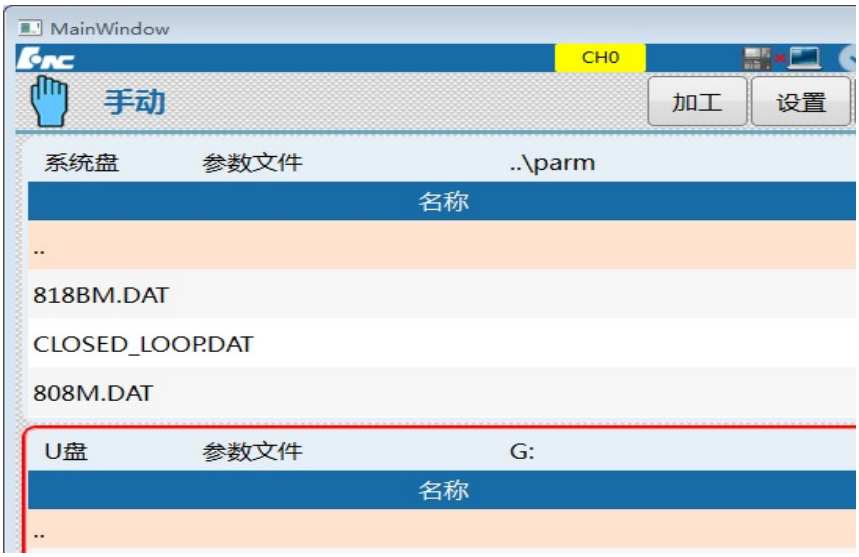
- Upgrade can be conducted with system permission by HNC CNC technical personnel .
- System upgrade must be conducted under "Emergency stop" status
- Upgrade patch file must be .BTF file and file name is suffixed with .BTF
- The default backup disk is the user disk, pay attention to the size of user disk during backup

3.6.7 Data Management

In the data management interface, single file of parameters, PLC, canned cycle, log, compensation, oscilloscope can be loaded/backed. This section takes the example of loading/backing up system parameter files. The operation steps of loading and backing up other files (except the error compensation file) 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 on 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, and the input box gives a prompt message "Whether to load the selected file?"
  - Press 「Y」 to load the data file;
  - Press 「N」 or 「Reset」 to abandon 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 the system 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 "Whether to back up the selected file?"
  - Press 「Y」 to back up data file;
  - Press 「N」 or 「Reset」 to abandon backing up data file.

Note Power failure is not allowed during backup or loading.

3.6.8 User Setup

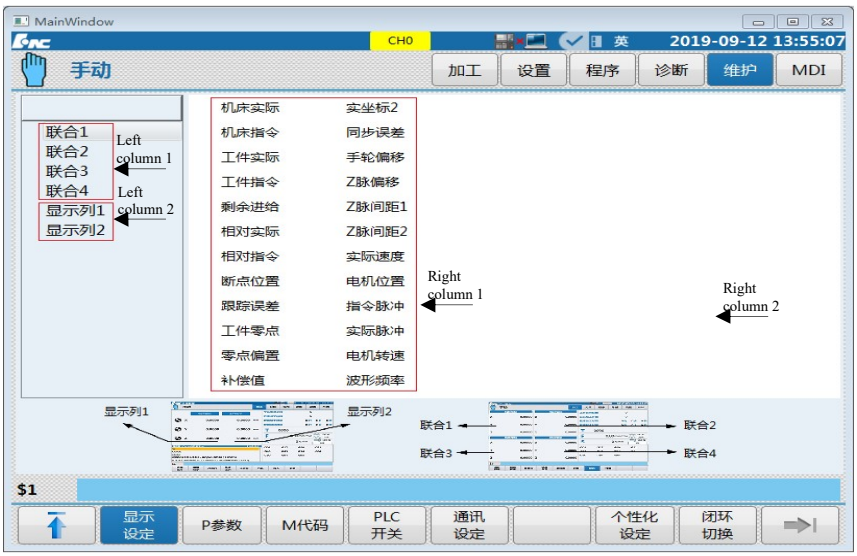
User setup is used for control switches of some common display and PLC so that users can set different functions based on different needs.

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

3.6.8.1 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), the 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), the content 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)

3.6.8.2 Set "P parameter"

P parameter is a parameter relating to machining and system operation. All PLC subprogram switches and PLC determinations of system are set by P parameter, and P parameter 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 machine tool user parameter 010300 and correspond to the same memory address with these user parameters. 『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 parameter, not only should PLC switch be set, but also other relevant parameters and functions should be set. e.g.: When rotation function of mill spindle is enabled, not only should parameters of spindle rotation ON/OFF be set, but also spindle rotation parameters should be set; otherwise, the spindle cannot rotate.

3.6.8.3 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 set" 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 M00, M01, M02, M05, M30, M92 and M93 codes is cured of the system, of which the Pre/Post attribute cannot be changed.

M00: Program suspension

While executing M00 command, the execution of current program is suspended, and the system is at feed hold state, so that the operator can conducts the operations including size measurement of tool and workpiece, workpiece turn-around and manual speed change. Press [Cycle start] to continue to run the program;

M01: Optional stop

If this function key on the system panel lights up, the system suspends the execution of current program while executing M01 command, and at time tim the system is at feed hold state; so that the operator can conducts the operations including size measurement of tool and workpiece, workpiece turn-around and manual speed change. Press [Cycle start] to continue to run the program;

If the “Optional stop” key on the system panel does not light up, the system does not suspend the execution of current program while executing M01 command.

M92: Program suspension (wait for user's manual intervention)

While executing M92 command, the system suspends the execution of current program and waits for user's manual intervention before cycle start. The difference from M00 is that the user can manually intervene with axes, and move axes under "Jog" mode. Then, press "Cycle start" under "Auto " mode to continue running the current program.

Note: When M92 is used, The channel parameter 040059【Automatic breakpoint block number return】 must be set as 0, that is, the function of returning to the breakpoint position automatically after manual intervention is turned off. Otherwise, manual intervention is invalid.

M93: Program suspension (manual intervention is not allowed)

M93 command is equal to M00 command. Differnet from M92, user's manual intervention is not allowed when M93 suspends the program.

M02: Program end

M02 is often edited in the last block of the main program. When the system executes M02 code, spindle, feed and cooling of the machine tool stop, and the machining is completed.

After a program using M02 ends, to re-execute the program, users should recall and load the program and press [Cycle start].

M30: Program termination

M30 is often edited in the last block of the main program. When the system executes M30 code, spindle, feed and cooling of the machine tool stop and the machining is terminated, and then the system returns to program header automatically.

After a program using M30 ends, to re-execute the program, recall and load the program and press [Cycle start].

a) Pre or post

Pre: M code takes effect first when M code and G code are in the same line;

Post: G code takes effect first when M code and G code are in the same line;

Synchronization: M code and G code are executed simultaneously when they are in the same line.

The synchronous execution of M code and G00 rapid traverse positioning command is not included in the above three situations.

b) Scanning of any line

Yes: After the scanning mode of the channel parameter 040113 【Any line mode selection】 is turned on, the system scans the M code while using any line function.

No: While using any line function, the system does not scan the M code whether any line function is turned on or not.

c) G00 synchronization

Yes: M code and G00 code are executed simultaneously;

No: M code and G00 code cannot be executed synchronously.

In HCNC system, the synchronous execution of M code and G00 is used as a configurable attribute. If the system do not conduct cutting while executing G00 command, some designated M codes can be executed synchronously to improve efficiency, such as M03 spindle CW rotation, M04 spindle CCW rotation, and M08 cooling ON, etc. After M code is executed, the response conditions of M codes in PLC must trigger G2562.13 signal and notify the system of executing the next block of movement.

0

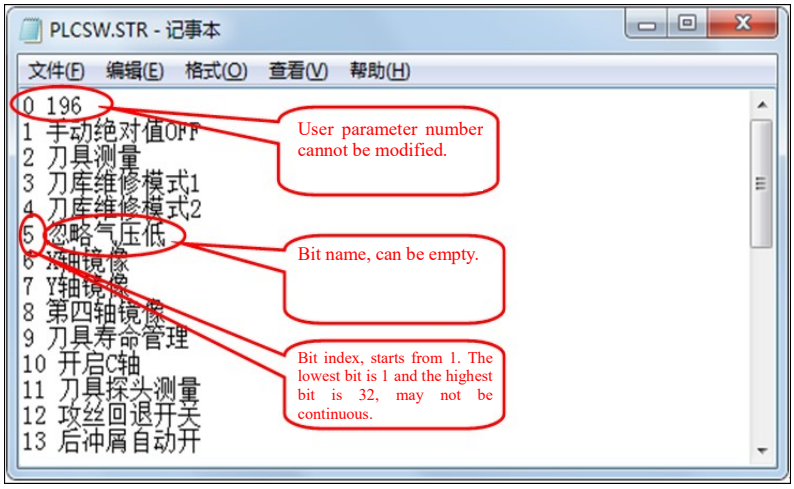
d) Spindle control

When M code is used to control CW rotation, CCW rotation and stop of spindle, it can be marked and prompted using this configuration, and it has

no effect on the actual attribute and action of M code. It is often used for marking M code of multiple channels and multiple spindles.

3.6.8.4 Set "PLC switch"

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



Note: Machine tool user parameter P196 of HNC-8 system is used as the PLC switch file by default. The setting is closely related to PLC function and cannot be modified without the guidance of CNC system commissioning personnel.

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 which can be operated only through pressing "Effective setup" menu;

ON: Set bit of focus as ON;

OFF: Set bit of focus as OFF;

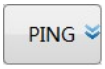
Setup result is saved in the designated user P parameter.

3.6.8.5 Communication setup

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

Note: 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

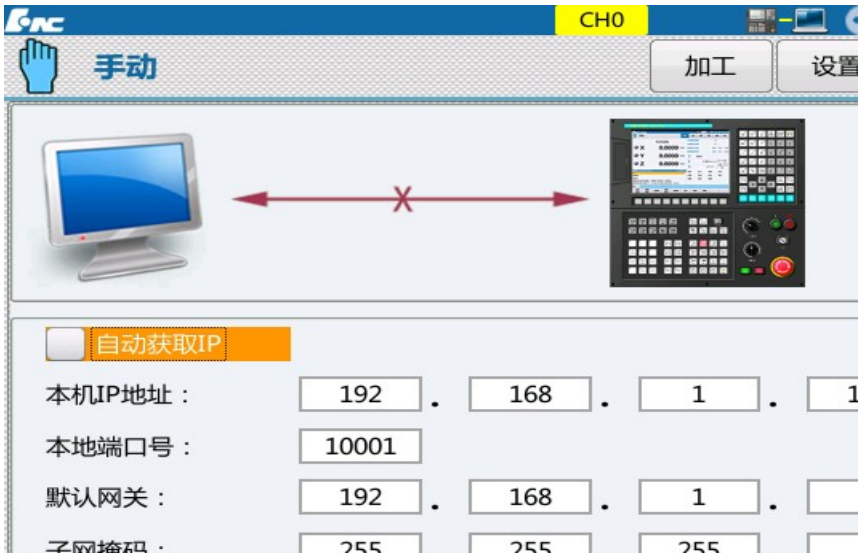


For communication between the upper computer of the CNC system and the computer, ip of the computer and ip of the CNC system should be set in the same block,

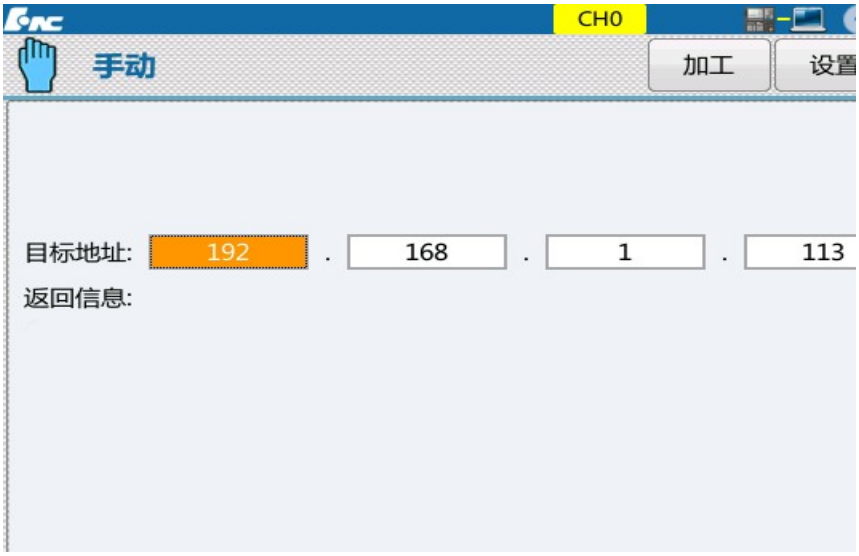
ip of the CNC system is 192.168.1.113 by default. ipv4 of the computer is set to 192.168.1.XXX at the time of connection, and default gateway and subnet mask of the computer are consistent with those of the CNC system. Specific steps:

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

- Press 『Local』 soft key to enter the "Local" connection sub-interface (as shown below);
- Move the cursor to where "Server IP address" is set, and set the 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 the "Local port number", "Default gateway" and "Subnet mask" of the computer.
- Connect Internet access of computer and that of system IPC using network cable. Pay attention not to connect to M3 or ETH port of system IPC.



- 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

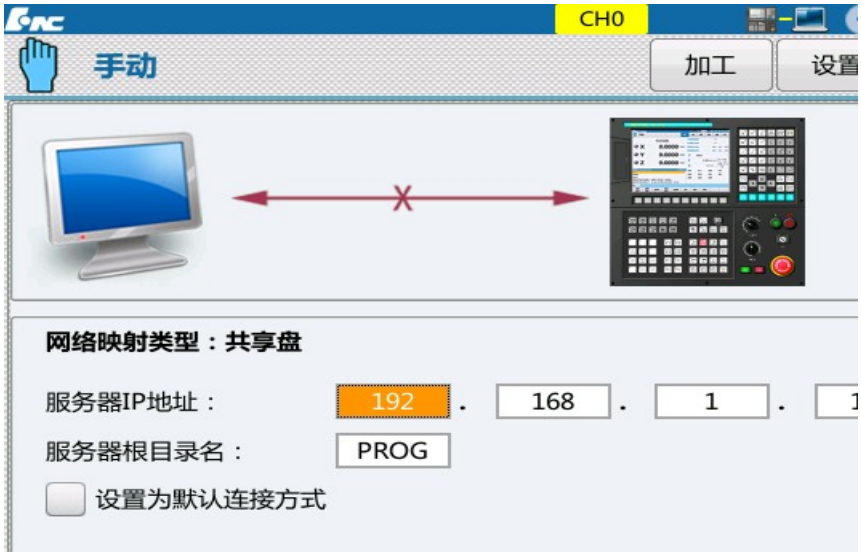
in PING interface and click on 『PING start』 soft key;

- 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『Communication setup』soft key under "Maintain" function set to enter the 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.

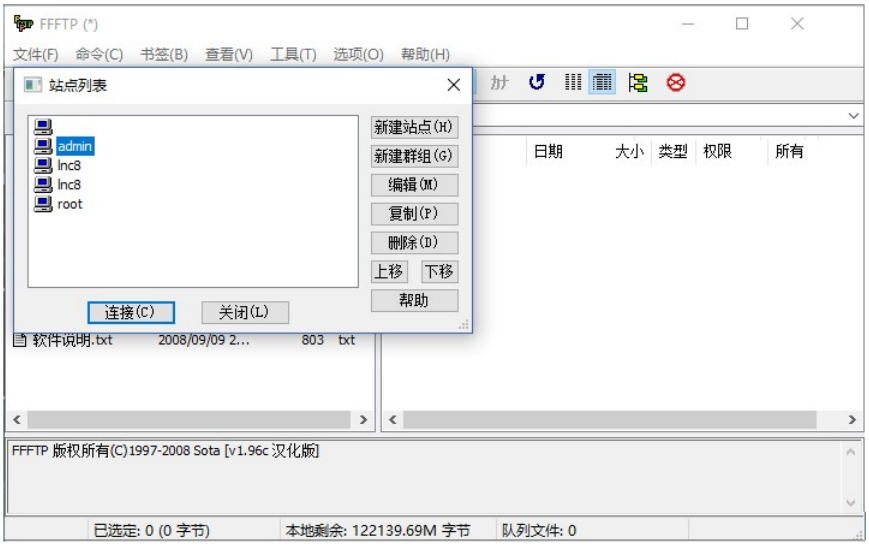
c) FTP

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

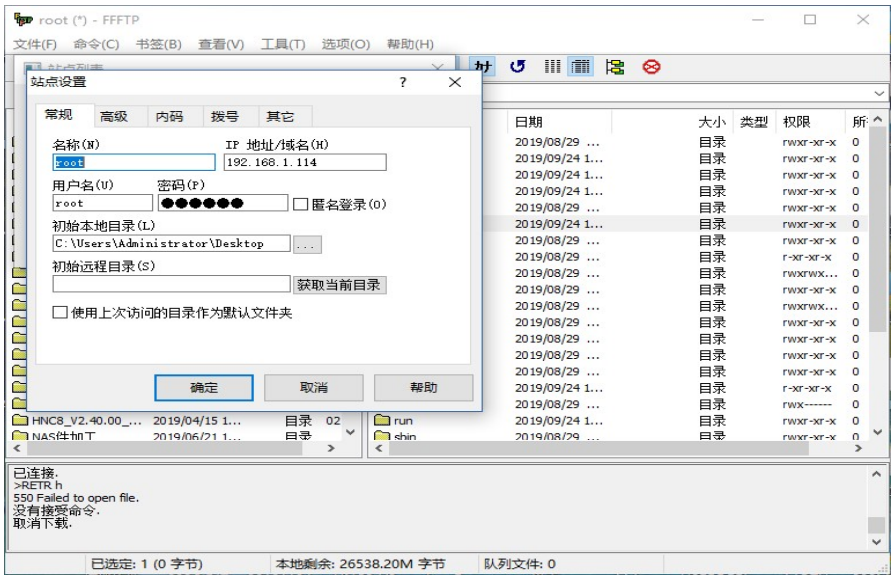
Before FTP is used, ensure smooth network according to the previous section.

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

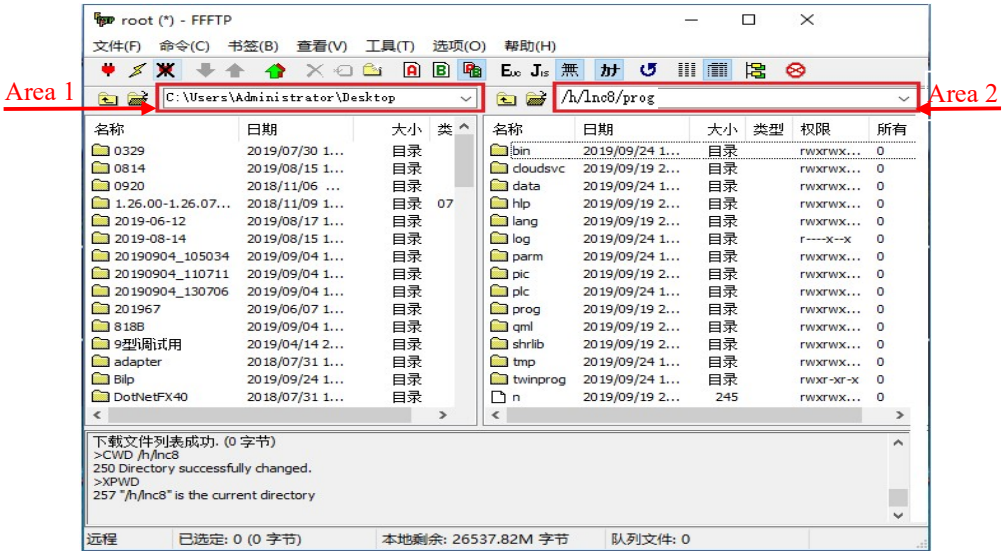
After ping the system and the computer, click on FFFTP icon and the interface is displayed as below:



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 as 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.

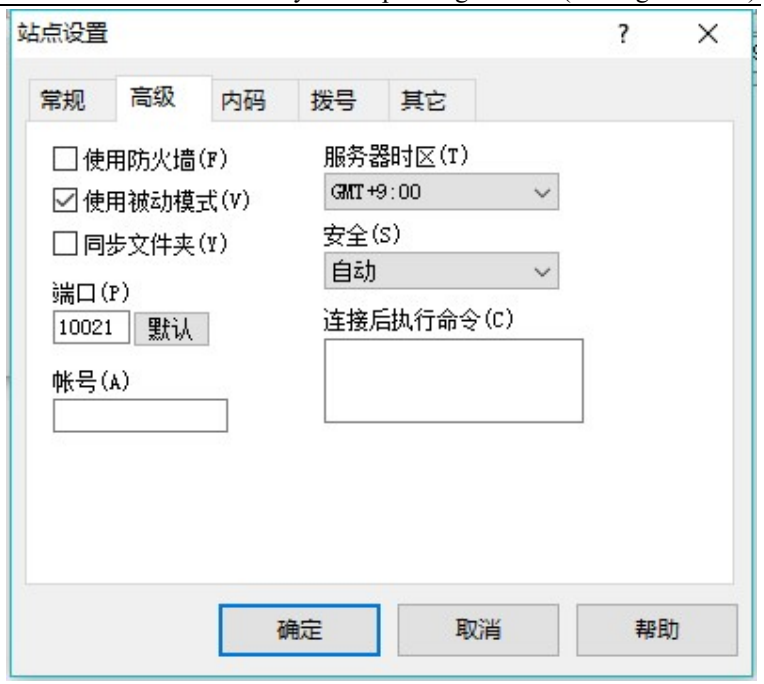


The transmission path of the system can be set in the above interface area 2 and system is often written in /h/lnc8/prog. The computer path can be set in area 1.

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

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

1. Whether the network parameter 00050 enables the network, and whether it is opened
2. Whether the computer and the system are connected can be pinged
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 PLC of system 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

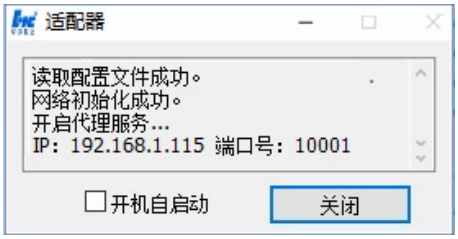
- The system and the computer must be connected, as shown in the second section.
- A suitable adapter is needed,

Setup steps for HCNC controller adapter software are as follows (it should be noted that 1.26 and 2.XX series adapters are different).

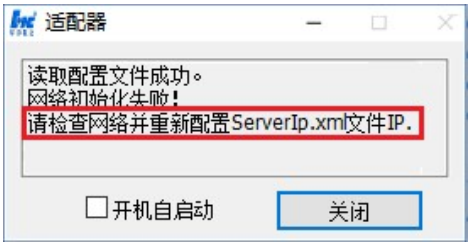
(1) Turn on the adapter and find ServerWindowD.exe and Serverlp.xml

(2) Open Serverlp.xml and replace LocalIP to IP of this computer, which is 192.168.1.115 in this example. The port number must be consistent with that on the system and should be edited and saved.

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

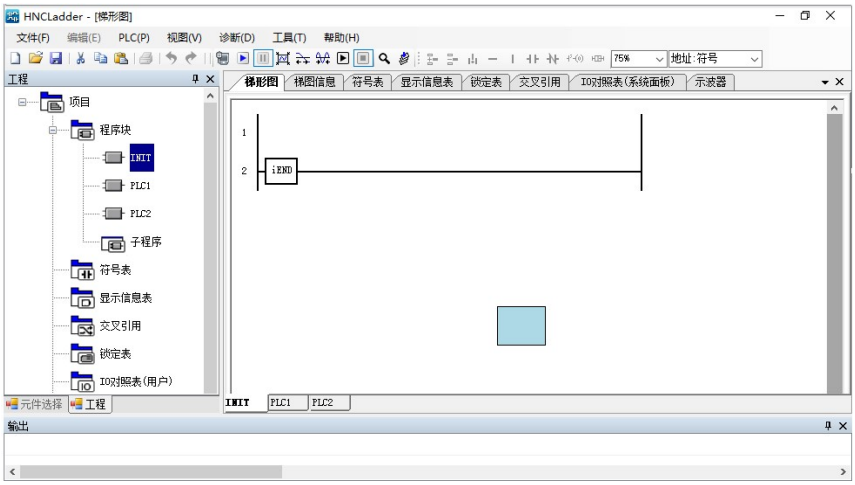


If Serverlp.xml is not modified, an alarm will be given as shown in the following figure



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

(5) Open the ladder diagram editing tools of 2.0, and click Network connection under setup



Correctly fill in IP of the computer and IP of the system and click to connect

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

3.6.8.6 Personalized setup

The system version can be realized through personalized setup: Language setup, resolution setup and skin setup. Press 『Personalized setup』 soft key under "Maintain" function set to enter the personalized setup sub-interface, as shown below. The modification should be validated after restart.

1. Language setup

Language setup can modify system interface language among Chinese, English and Russian.

2. Resolution setup

Resolution setup can modify resolution as 800*600 or 1024*768

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

3. Skin setup

The skin both in black and blue are supported. Subsequent versions will support gray as well.

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

"Process package" of this system is shown below:

- Backup

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

- Load

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

- Rename

  Users can 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 the network disk).

Note:

- The directory of process package is parm, and process file can be renamed.
- Naming rule for XML file: CB_*.XML. “CB” and “XML” is in capital form. 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 in 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 machine tool status is normal; (2) Check whether power supply voltage is consistent with the requirements; (3) Check whether connection is correct and secure.		

SN	Operation steps	Key	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, inspect 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	Key	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 Protectiona And Release

4.3.1 Overtravel Protection

There is a travel limit switch at each end of the travel of servo axis, which is used to prevent from damaging the linear axis mechanism arising from collision. When the mechanism touches the travel limit switch, the hard 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 setup 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 give an alarm.		

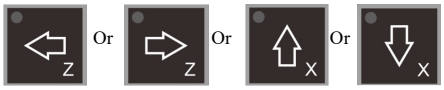
SN	Operation steps	Key	Description
1	Press 【Jog】 or 【Handwheel】		● Set effective working mode
2	Press [Overtravel release] and [Axis feed]		● Press [Overtravel release] and [Axis feed] simultaneously ● Select [Axis feed] in the reverse direction of overtravel axis

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 the 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 state bar changes to "Normal", which means the normal work is restored and operation can be resumed.

4.3.3 Over Travel 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 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 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

When the machine tool is running, in case of danger or an emergency, press "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 "Emergency stop" (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, re-execute returning to the reference point in order to ensure correctness of coordinate position.

Note:

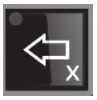
- Press **"Emergency stop"** to reduce electric shock of device before power-on and power-off.

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]	 Or  Or 	● [Axis feed] key* in the specified stipulated direction

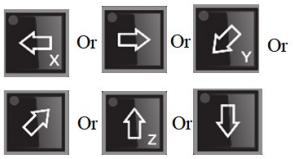
Note:

- When the machine tool is configured with the absolute encoder motor, the system needs not return to the reference point;
- While returning to the reference point, [Axis feed] key is determined according to “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 the reference point;
- After all axes return to the reference point, as long as the servo drive device does not give an alarm during operation, the reference point return doesn’t need to perform when other alarms are issued (including pressing 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 to move coordinate axis of the machine tool manually.

Operation name	Move coordinate axis in jog mode	Working mode	Joh
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
	Press [Axis feed]		● If the key is released, feed stops

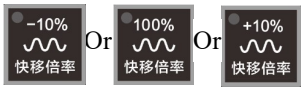
Note:

- Set the default speed by “Slow speed Jog speed” parameter (X: 100032, Y: 101032, Z: 102032) in coordinate axis parameters;
- Press all [Axis feed] keys, 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

This function can move the coordinate axis rapidly and continuously. Press 【Jog】 working mode, [Rapid traverse override] and [Rapid traverse]+[Axis feed] on the control 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:

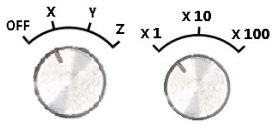
- Set the default speed by “High speed jog speed” parameter (X: 100033, Z: 102033) in coordinate axis parameter;
- Based on 100% of default speed, increase and decrease rapid traverse override rate as per 10%;
- Under other modes than "Manual" mode, the [Rapid travers] key is invalid.

5.4 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.

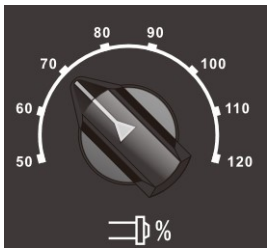
Operation name	Move the coordinate axis by handwheel	Working mode	Handwheel
Basic requirements	(1) The need for continuous accurate movement of machine tool		

SN	Operation steps	Key	Description
1	Press 【Handwheel】		● Set effective working mode
2	Select [Axis selection] and [Magnification] of handheld unit		● For the axis selection,Select X, Y and Z axis or OFF (no axis selection) ● The product of magnification and 0.001mm
	Swing [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;
- 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.

5.5 Manual Control Of Spindle

<table>
<tr><th>SN</th><th>Operation name</th><th>Start operation</th><th>Stop operation</th><th>Description</th><th>Effective working mode</th></tr>
<tr><td>1</td><td>Spindle rotation CW</td><td>Press [Spindle rotation CW] key
  </td><td>Press [Spindle stop] or [Reset] key
 
  </td><td rowspan="3">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.</td><td rowspan="3">Handwheel, incremental, jog</td></tr>
<tr><td>2</td><td>Spindle rotation CCW</td><td>Press [Spindle rotation CCW] key
  </td><td>Press [Spindle stop] or [Reset]</td></tr>
<tr><td>3</td><td>Spindle stop</td><td>Press [Spindle stop]</td><td>Press [Reset]</td></tr>
<tr><td>6</td><td>Spindle speed magnification</td><td colspan="2">Rotate [Spindle override] knob
  </td><td>1. Override range: 50%-120%.</td><td>Handwheel, incremental, jog, auto, MDI</td></tr>
</table>

5.6 Other Manual Operations

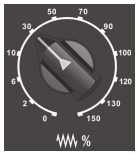
SN	Operation name	Start operation	Stop operation	Description	Effective working mode
1	Next tool	Press [Next tool] 	Stop after the next tool is changed	Press this key to control the magazine tool to change the next tool.	Jog
2	Tool change	Press [Tool change is] 	Press this key again	Press this key, and the system allows tool changing.	
3	Tool clamping/release	Press [Tool clamping/release] 	Press this key again	When the tool change is allowed, press this key to control loosening and tightening of spindle tool.	
4	Magazine commissioning	Press [Magazine] 	Press this key again or reset key	K2.0 is turned on for K parameter, press this key, and the system enters magazine commissioning state.	
5	Magazine rotation CW	Press [Magazine CW] 	Stop after rotating a tool position	The magazine rotates CW a tool position as this key is pressed once.	
6	Tool arm rotation CW	Press [Tool arm CW] 	Stop after rotating CW an action	Under magazine commissioning state, the tool arm rotates CW an action.	
6	Machine 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 to cyclically control ON or OFF of lubricating pump. It is OFF by default.	
8	CW rotation of chip removal	Press [Chip removal CW] 	Press this key again	Press this key to control the CW rotation or stop of chip removal motor cyclically. It is OFF by default.	
8	CCW removal of chip removal	Press [Chip removal CCW] 	Press this key again	Press this key to control the CCW rotation or stop of chip removal motor cyclically. It is OFF by default.	
9	Blowing ON/OFF	Press [Blowing] 	Press this key again	Press this key to control the start or stop of blowing cyclically. It is OFF by default.	

10	Manual cooling ON/OFF	Press [Manual cooling] 	Press this key again	Press this key to control the start or stop of cooling pump cyclically. It is OFF by default. Start/stop cannot be controlled through M8 and M9 codes.	
11	Auto cooling Start/stop	Press [Auto cooling] 	Press this key again or M9 code	Press this key to control the start or stop of cooling pump cyclically. It is OFF by default. Start/stop cannot be controlled through M8 and M9 codes.	Auto

5.7 Speed Override

5.7.1 Feedrate Override

Under auto mode or MDI operation mode, when the feedrate specified by F command is too high or too low, rotate the feedrate override switch to adjust the feedrate in the program. Override range: 0%-120%.



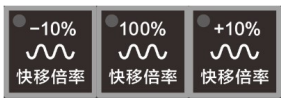
If the feedrate is not designated in the automatic operational program and "Manual MS" function is not enabled, the system runs as per the set value of channel parameter "Default feedrate" 040030. If "Manual MS" function is enabled, the system runs as per the speed configured in "Manual MS" and the feedrate override in the operational process is valid.

Under manual continuous feed mode, the manual feedrate can be adjusted by this switch, and the system runs as per the set value of axis parameter "Low-speed jog speed" 100032 (X axis), 101032 (Y axis) and 102032 (Z axis).

Note:

- When this knob is turned to 0, the feedrate override is 0, and rapid traverse override rate changes to 0 temporarily. If the feedrate override is changed to a non-zero value, the rapid traverse override recovers to the original value;
- Feedrate override is invalid during rigid tapping machining, the program runs with the original speed;

5.7.2 Rapid Traverse Speed Override



When a program runs under 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, the jog 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 rapid traverse block in jog rapid traverse, G00 rapid traverse and canned cycle, G28, and G29 can increase and decrease by 10% by "Rapid traverse magnification", and the override range is 0%-100%.

Note:

- As requested by some users, rapid traverse override of some systems is set as follows: When rapid traverse override is set as 0%, the actual rapid traverse is 2%. Actual rapid traverse override is 0% only when the feedrate override is also 0.

6 Program Edit and Management

6.1 Program Search

HNC-808DiM system has the program search function under "Machining" function set and "Program" function set, but usage of programs found under the 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 searched under "Program" function set are used for program management including program copy, paste and delete and program transmission among different disks,etc.

6.1.1 Machining Or Editing Program Search

6.1.1.1 Direct search

Operation name	Machining or editing program search	Working mode	Auto, single block, jog
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』 and 『USB flash disk』, and so on		● 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 found programs can be used for two purposes
5.1	Press 「Enter」		● The found programs are used for loading machining program and running
5.2	Press 『Backstage edit』		● The found programs are used for entering program editing status

6.1.1.2 Search programs under different disks by "Find" function

Operation name	Search of machining or editing program		Working mode	Auto, single block, jog
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』 and 『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	
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」		● Searched programs are used for loading machining program and running	
7.2	Press 『Backstage edit』		● 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
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』	选择程序	● "Select program" sub-interface, level 2 menu
3	Press 『System disk』 and 『USB flash disk』, etc.	系统盘 U盘 网盘	● Select system disk, USB flash disk, network disk and user disk
4	「Cursor」 or 「PgUp/PgDn」	PgUp 上页 PgDn 下页 ▲ ▼	● Move the cursor to the selected file directory name
5	Press 「Enter」		● Confirm and open the directory
6	Press 『Find』	查找	● Prompt: Enter a file name
7	(Enter a file name)	---	● e.g.: Onc123
8	Press 「Enter」	Enter 确认	● Complete searching programs and move the cursor to the program to be searched; ● The found programs can be used for two purposes
9.1	Press 「Enter」	Enter 确认	● The found programs are used for loading machining program and running.
9.2	Press 『Backstage edit』	后台编辑	● The found programs are used for entering program editing status

6.1.2 Search Of Management Program (to Be Transmitted and Deleted)

6.1.2.1 Direct search

Operation name	Search of management (copy, paste) program	Working mode	Auto, single block, jog
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〗	程序 Porg	● Default interface, main menu
2	Press 『System disk』 and 『USB flash disk』, etc.	系统盘 U盘 网盘	● Select system disk, USB flash disk, network disk and user disk
3	Press 「Cursor」 or 「PgUp/PgDn」	PgUp 上页 PgDn 下页 ▲ ▼	● 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	Search of management (copy, paste) program	Working mode	Auto, single block, jog
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』 and 『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	Search of management (copy, paste) program	Working mode	Auto, single block, jog
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』 and 『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 Edit

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

Operation name	Create new programs under "Machining" function set	Working mode	Auto, single block, jog
Basic requirements	The new program name cannot be the same as 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 such as O321

5	Press 「Enter」		● Confirm the new file name and enter the editing area ● To rename, a prompt message will be given and a request for reentering a program name will be given
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 machining program

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

Operation name	Create new programs under "Machining" function set	Working mode	Auto, single block, jog
Basic requirements	The new program name cannot be the same as 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	Select 『Select program』		● "Select program" sub-interface, level 2 menu
	Press 『Backstage edit』		● Enter the "Backstage edit" sub-interface, level 3 interface
3	Press 『New』		● Prompt: Please enter a file name
4	(Enter a program name)	---	● Input program name such as O321
5	Press 「Enter」		● Confirm the new file name and enter the editing area ● To rename, a prompt message will be given and a request for reentering a program name will be given
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

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
Basic requirements		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
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
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』	保存文件	● Program has been saved

Note:

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

6.2.2.2 Editing and modification of non-loading program in the backstage

Operation name	Editing and modification of non-loading program in the backstage	Working mode	Auto, single block, jog
Basic requirements	Existing programs to be modified	Display interface	3.2.2 "Select program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖Machining〗	加工 Mach	● Default interface, main menu
2	Press 『Select program』	选择程序	● "Select program" sub-interface
3	Press 「Cursor」	◀ ▶ ▲ ▼	● Select current programs to be edited and modified
4	Press 『Backstage edit』	后台编辑	● Enter program editing status
5	(Edit program)	---	● Edit and modify existing loading programs
6	Press 『Save』	保存文件	● Program has been saved

Note:

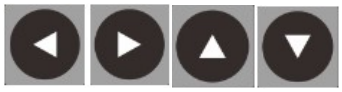
- When current loading program is selected in the "Select program" sub-interface, the current loading program also can be edited through "Backstage edit" function.

6.2.3 Save as

"Save as" function of this system is to integrally and quickly copy the program of current editing status. Thus, the 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 edit" sub-interface in order to realize the saving.

6.2.3.1 Save program as "Current loading program"

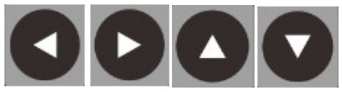
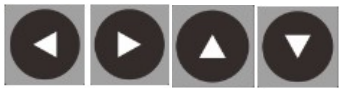
Operation name	Save program as "Current loading program"	Working mode	Auto, single block, jog
Basic requirements	The saved file name cannot be the same as the existing program names	Display interface	3.2.3 "Edit program" sub-interface

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 all disks can be selected
4	Press 「Cursor」		● Move the cursor to the selected target disk or file directory name
5	Press 「O」		● Activate the file name input box
6	(Input the saved file name)		● If renamed, the original program will be covered
7	Press 「Enter」		● 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 "Non-loading program" as

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

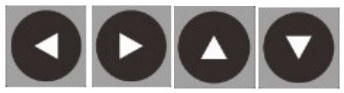
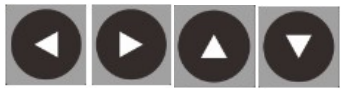
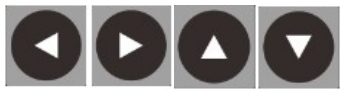
SN	Operation steps	Key	Description
1	Press 〖Machining〗		● Default interface, main menu

2	Press 『Select program』	选择程序	● "Select program" sub-interface
3	Press 「Cursor」		● Select the current program to be saved
4	Press 『Backstage edit』	后台编辑	● 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 all disks can be selected
6	Press 「Cursor」		● Move the cursor to the selected target disk or file directory name
7	Press 「O」		● Activate the file name input box
8	(Input the saved file name)		● If renamed, 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 Program Block

Operation name	Copy and paste of program block	Working mode	Auto, single block, jog
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	● This system enters program editing status in 4 ways: ✧ Create "new program" function under "Machining" function set ✧ "Edit program" function under "Machining" function set ✧ "Backstage edit" function under "Machining" function set ✧ Create "New program" function under

			"Program" function set
2	『Block』		● Enter "Block " 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	『Block copy』		---
8	「 Cursor 」 or 「PgUp/PgDn」		● Move the cursor to where the current program or other programs are pasted
9	『Paste』		● Paste succeeds
10	『Save』		● Exit and save programs

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
Basic requirements	Existing programs can be searched	Display interface	3.4.4 "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 search program of "Program" function set
3	Press 『→』		● Enter "Program" set, level 1 extension menu

4	Press 『Rename』	重命名	● Prompt: Enter a new file name
5	(Rename a file)	---	● Enter a new file name
6	Press 「Enter」	Enter 确认	● Confirm the new file name ● Prompt: Rename the old file as a new file

6.3.2 Copy and Paste File Directory and Program

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

SN	Operation steps	Key	Description
1	Press 〖Program〗	程序 Porg	● 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 search program of "Program" function set
3	Press 『→』		● Enter "Program" set, level 1 extension menu
4	Press 『Copy』	复制	● Prompt: Select target disk of paste
5	(Select target disk or directory)	---	● Move the cursor to the target disk or the file directory according to 6.1.2 search program of "Program" function set
6	Press 『Paste』	粘贴	● 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
Basic requirements	Programs to be deleted can be searched	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 search program of "Program" function set
4	Press [Delete]		● Prompt: "Conform 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
Basic requirements	Programs to be deleted can be searched	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 search program 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

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 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
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” ● Address word of the new file name is O and needs not inputted
6	Press 「Enter」		● Confirm the input, file name is Onc123 ● The cursor enters the editing area
7	(Edit program)	---	● Edit program and complete
8	Press 『Save file』		● The newly edited program is loaded as the machining program immediately ● A prompt message File has been saved will be given

Note:

- After the new program is saved under "Machining" function set, it can be loaded as current machining program automatically
- The new program cannot be loaded as 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
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 loading program)	---	● Select programs to be loaded as "Current machining program" ● Search programs as per 6.1.1
6	「Enter」		● Loading is completed

7.1.2 Program Run

Operation name	Program run	Working mode	Auto
Basic requirements	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	(Safety inspection)	---	● Complete deceleration and lock
5	Press [Cycle start]		● Run a program automatically

Note:

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

7.1.3 Program Verify

Operation name	Program verify	Working mode	Auto, single block
Basic requirements	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』		● Working mode is displayed as "Verify" ● The『Verify』 soft key is highlighted.
5	Press [Cycle start]		● Exit the verification after automatic operation ● Press [Reset] to exit the verification

7.1.4 Program Graphics Simulation

Operation name	Program graphics simulation		Working mode	Auto, single block
Basic requirements	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, and the interface is switched ● Select “Graphics+ program” interface	
5	Press [Cycle start]		● Conduct automatic operation and realize graphics simulation	

Note

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

7.2 Automatic Operation Control

7.2.1 Single-block Operation

Operation name	Single-block operation		Working mode	Single block
Basic requirements	Complete 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 machining program as per 7.1.1
5	Press [Cycle start]		● Press Cycle 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 	● Block skip operation function is set only under jog, handwheel or incremental mode
2	Press [Block skip]		● Program blocks with the 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 automatic operation.

Note:

- If program blocks with the skip symbol "/" are executed after [Block skip] is pressed, the system skips over this line of commands, and directly executes the next line of commands.
- If the [Block skip] is not pressed, the system still executes this line of commands in order.

7.2.3 Execute from Any Line

Operation name	Execute 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 『Any line』		● Enter the "Any line" submenu
4	Press 『Specify line number 』 and 『 Specify N number』	 Or 	● The indicator light lights off ● Suspend operation
5	(Input line number)	---	● Input a value, such as 8
6	Press 「Enter」		● Confirm the input ● The cursor moves to the line before the inputted line ● Move the cursor to the selected any line by 「Cursor」
7	Press [Cycle start]		● Start to run from the specified 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 modal before the target line is inherited except Z axis mode;

  2: Full scanning mode. The modal before the target line is 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 machine, 040114=211 means X/Y axis moves to the right position and then Z axis reaches the right position.

For milling machine, 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 instruction 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 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 its execution ● 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 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 its execution ● If the 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 name	Dwell	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 the program)	---	● The program is running
4	Press [Feed hold]		● The indicator light lights off ● The operation is suspended

5	Press [Cycle start]		● Continue the operation

Note:

- During thread machining, feed hold cannot be validated immediately, and it is not validated until thread instruction 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 the program)	---	● The program is running	
4	Press [Feed hold]		● The indicator light lights off ● The operation is suspended	
5	Press 【Jog】		● To close MST manually	
6	(Disable M and S functions)	---	● Disable MST manually	
7	Press [Emergency stop]		● Terminate the operation ● Reset	

7.3 MDI Operation

"MDI" function of HNC-8 system has two forms,

- MDI key is the working mode key【MDI】 of MCP panel
- MDI key is the function set key 〖MDI〗 of NC panel

The【MDI】key of HNC-808DiM system is on MCP panel. 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】	 or 	● 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】	 or 	● 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 mode 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 operation ● Reset is not allowed 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 interface is switched, the edited MDI program cannot be cleared

Note:

- When the parameter 000371 is set as 0, The MDI is the key mode of MCP panel; when it is set as 1, it is the key mode of NC panel.
- If【MDI】key is on the NC panel, “MDI” is the function set key. There is an independent interface for the MDI working mode. If the working mode is switched, the interface changes accordingly.
- If〖MDI〗key is on the MCP panel, “MDI” is the working mode key. This function is valid under auto mode or single block mode.

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 precutting	Working mode	Auto
Basic requirements	The machining preparation is in the state of automatic program running	Display interface	See Chapter 3 "Machining" function set interface

SN	Operation steps	Key	Description
1	Press 【Auto】		● This function can be executed under automatic 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. ● The machine tool axis does not move at this moment
4	(Rotate the handwheel)		● If the handwheel rotates clockwise, the machine tool axis moves forward with the program ● If the handwheel rotates counterclockwise, the machine tool axis moves backward with the program
5	(Check tool position)	---	● Visually inspect correctness of tool position
6	Press [Handwheel simulation]		● Function is released and the indicator light lights off ● The machine tool continuously runs the subsequent programs until shutdown

7.5 Machining Information Query

Operation name	Machining information query	Working mode	Auto, jog, handwheel, reference point return
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 statistics』		● Enter the "Machining statistics" sub-interface ● The system displays number of processed parts and information related to system operation time
4	Press 『Preset』		● Set total number of processed parts needed, etc.
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

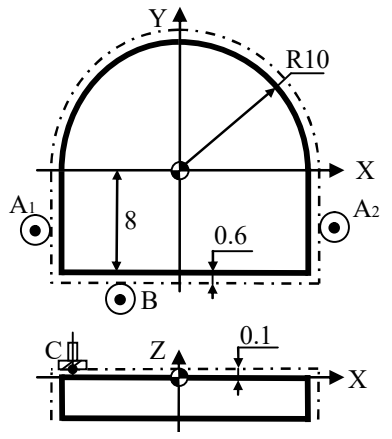
Manual tool setting is completed mainly in "Coordinate system" and "Tool compensation" sub-interfaces. For ease of user operation, the system set two sub-interfaces in both "Machining" and "Setup" function sets to reduce interface switching. The operating steps for both are basically the same.

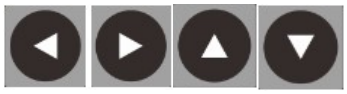
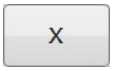
For some specific parts, the system can simplify manual tool setting through "Workpiece measurement" function in "Setup" function set. This function includes "Center measurement", "Plane measurement" and "Circle center measurement" and is mainly used to set coordinates of X, Y and Z axes automatically.

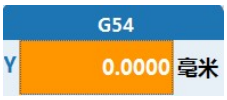
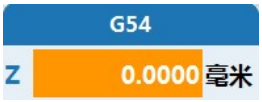
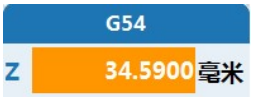
With the popularity of CNC machine tools, the application of tool setter has become more widespread, and automatic tool setting has been increasingly used. The system is configured with "Auto tool setting" function in "Setup" function set, which can be used for automatic setup of Z axis coordinates and tool length compensation.

8.1 Manual Tool Setting

Manual tool setting is taken as an example under "Setup" function set in this quarter.

<table>
<tr><td>Operation name</td><td colspan="2">Manual tool setting</td><td>Working mode</td><td>Jog, handwheel</td></tr>
<tr><td>Display interface</td><td colspan="2">3.2 "Coordinate system" and "Tool compensation" sub-interfaces under "Setup" function set</td><td colspan="2" rowspan="2">零件对刀示意图
 
</td></tr>
<tr><td>Basic requirements</td><td colspan="2">1. Peripheral allowance is uniform and is 0.6mm;
2. Allowance of the upper surface is 0.1mm;
3. Straight edge of the blank is basically parallel to the coordinate axis;
4. Roughing tool and finishing tool are adopted, and the tool diameter is 10mm;
5. When A1 and A2 are the tool setting on X axis, the tool just touches the workpiece;
6. When B is the tool setting on Y axis, the tools just touches the workpiece;
7. When C is the tool setting onZ axis, the tool just touches the workpiece.</td></tr>
<tr><td>SN</td><td>Operation steps</td><td>Key</td><td colspan="2">Description</td></tr>
</table>

1	Press 〖Setup〗		● Enter the "Setup" function interface
2	Press 『 Coordinate system』		● 『Coordinate system』 is the soft key of level 1 menu;
3	Select coordinate axis and tool setting axis (X axis)		● Select X axis coordinates of G54 using cursor as below: 
4	Move the tool to A1 point with handwheel		● The tool just touches the left edge of workpiece (precutting mode);
5	Press 『 Relative clear』		● Switch the interface to "Relative clear" sub-interface;
6	Press 『X』		● Relative clear of X axis, and "Relative actual" coordinate is displayed as 0; 
7	Move the tool to A1 point with handwheel		● After avoiding the workpiece, the tool just touches the right edge of workpiece; ● Read "Relative actual" coordinate value. If it is 21.2106 (error of precutting position value, cannot be greater than allowance value); at this time it is displayed as 
8	Move the tool to the point of "Relative actual" 10.6053 of X axis		● Move to the A1A2 midpoint, namely half of "Relative actual" value 21.2106/2=10.6053; ● The midpoint of A1A2 is the workpiece coordinate zero on C, and is displayed as  .
9	Press 『↑』		● Return to the previous "Coordinate system" sub-interface.
10	Press 『 Current input』		● Press 『Current input』 to set machine coordinate value of the tool as the workpiece coordinate zero on X. Replace the original value, and display the present value  .

11	Select coordinate system and tool setting axis (Y axis)		● Select Y axis coordinates of G54 using cursor as below: 
12	Move the tool to B point with handwheel		● The tool just touches the lower edge of workpiece (precutting mode)
13	Press 『Relative clear』		● Switch the interface to "Relative clear" sub-interface;
14	Press 『Y』		● Relative clear of Y axis, and "Relative actual" coordinate is displayed as 0;  。
15	Move the tool to the point of "Relative actual" 13.6 of Y axis		● The distance between the workpiece coordinate zero on Y and the tool is: The distance from the zero point to workpiece edge + allowance + tool radius 8+0.6+10/2=13.6; ● Displayed as: 
16	Press 『↑』		● Return to the previous "Coordinate system" sub-interface.
17	Press 『Current input』		● After tool setting of workpiece zero on Y is completed, record the machine coordinate value on Y axis of current tool in the coordinate system. 
18	Select coordinate axis and tool setting axis (Z axis)		● Select Z axis coordinates of G54 using cursor as below: 
19	Move the tool to C point with handwheel		● The tool just touches the upper surface of the workpiece (precutting mode)
20	Press 『Current input』		● Record the machine coordinate value on Z axis of current tool in the coordinate system .
21	Press 「Cursor」		● Returning to Z axis coordinates 

22	Press 『Incremental』	增量 输入	● The workpiece zero is at the distance equal to an allowance below the tool, -0.1mm. (The direction of the workpiece zero relative to the tool is opposite to the direction of the work coordinate system)
23	Input incremental value "-0.1"	--	● Increase "-0.1"mm based on the above value and confirm, it is displayed as 
24	After tool setting is completed, the zero of G54 is displayed	--	

8.2 Workpiece Measurement

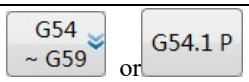
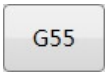
"Setup" function set supports the workpiece measurement function. This function supports center measurement, plane measurement and circle center measurement, and the measurement results are calculated and saved in the selected coordinate system.

8.2.1 Center Measurement

When the zero of the workpiece coordinate system is set in the symmetric central position of the workpiece, this tool setting mode is applicable. The system reads the coordinate value of machine tool in the same axial direction when the tool touches both ends of the workpiece, and then calculates the value of the zero of workpiece coordinate.

For vertical machine tools, the Z axis direction of the workpiece is not centered. Thus, move the tool to the workpiece zero(generally on the upper surface of the workpiece) during precutting, and keep the tool position unchanged while setting A and B points on the interface. At this time, press the 『Read measurement』 to read the machine coordinate value of the tool, and the point is set as the zero point value of the workpiece.

Operation name	Center measurement	Working mode	Jog, handwheel
Basic requirements	The system is allowed to run under jog mode and handwheel mode	Display interface	See Chapter 3 "Workpiece measurement" interface

SN	Operation steps	Key	Description
1	Press 〖Setup〗		● Enter the main interface of "Setup" function set.
2	Press 『Workpiece measurement』		● Enter the "Workpiece measurement" function interface.
3	Press 『Center measurement』		● Switch to the center measurement workpiece interface.
4	Press 『G54-G59』		● Select the type of coordinate system.
5	Press 『G55』		● Select the coordinate system.
6	Press left and right 「Cursor」		● Select the point A for tool setting.
7	Press up and down 「Cursor」		● Select the X coordinate axis;
8	Move the tool to the left size of the workpiece by handwheel		● The tool just touches the left edge of workpiece blank (precutting mode); ● At this time, select point A corresponding to the left side of the workpiece.
9	Press 『Read measurement』		● The cursor skips to B point automatically on X axis
10	Move the tool to the right size of the workpiece by handwheel		● After avoiding the workpiece, the tool just touches the right edge of workpiece blank; ● The error of precutting position cannot be greater than the allowance value. 
11	Press 『Read measurement』		● The cursor returns to A point automatically and the coordinate axis remains unchanged
12	Press up and down 「Cursor」		● Select Y coordinate axis;
13	Move the tool to the rear side of the workpiece by handwheel		● The tool just touches the rear edge of workpiece blank precutting mode); ● Then, select the A point corresponding to the rear of workpiece.

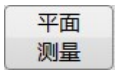
			X 15.5996 AY -28.8000 Z -8.8000
14	Press 『Read measurement』	读测 量值	● The cursor skips to B point automatically on Y axis
15	The tool moves to the front side of the workpiece by handwheel		● After avoiding the workpiece, the tool just touches the front edge of workpiece blank; ● The error of precutting position value cannot be greater than allowance value. X 19.1997 BY -111.6000 Z -8.8000
16	Press 『Read measurement』	读测 量值	● The cursor returns to A point automatically and the coordinate axis remains unchanged
17	Press up and down 「Cursor」		● Select Z coordinate axis;
18	The tool moves to the upper surface of the workpiece by handwheel		● The tool just touches the upper surface of the workpiece blank (precutting mode); ● Then, the upper surface of the workpiece is the Z axis zero point of the workpiece.
19	Press 『Read measurement』	读测 量值	● The cursor skips to B point automatically on (Z) axis; ● Keep the tool position unchanged and execute subsequent operations. X 15.5996 AY -28.8000 Z -8.8000
20	Press 『Read measurement』	读测 量值	● The cursor returns to A point automatically and the coordinate axis remains unchanged
21	Press 『Coordinate setup』	坐标 设定	● The system calculates measurement results and assign them to the selected coordinate system. G55 X 17.3997 Y -70.2000 Z -8.8000

8.2.2 Plane Measurement

This measurement mode sets the current coordinate value of the tool under the machine tool coordinate system as the zero point of workpiece coordinates. It is applicable to building the workpiece coordinate system with Z axis as the upper surface of workpiece and with the distance from X and Y axes to edges of the workpiece equal to a radius during plane

machining.

Operation name	Plane measurement	Working mode	Jog, handwheel
Basic requirements	The system is allowed to run under jog mode and handwheel mode	Display interface	See Chapter 3 "Workpiece measurement" interface

SN	Operation steps	Key	Description
1	Press 〖Setup〗		● Enter the main interface of "Setup" function set
2	Press 『 Workpiece measurement』		● Enter the default interface of "Workpiece measurement" function
3	Press 『 Plane measurement』		● Switch to the plane measurement function interface
4	Take X axis coordinates of workpiece		● Input the cursor on X axis coordinates ● Move the tool to a point on the edge of X axis of the workpiece in handwheel mode, and acquire coordinates of X axis based on the measured value
5	Take Y axis coordinates of workpiece		● Input the cursor on Y axis coordinates ● Move the tool to a point on the edge of Y axis of the workpiece in handwheel mode and acquire coordinates of Y axis based on the measured value
6	Take Z axis coordinates of workpiece		● Input the cursor on Z axis coordinates ● Move the tool to a point on the edge of Z axis of the workpiece in handwheel mode and acquire coordinates of Z axis based on the measured value
7	Select coordinate system		● Select the coordinate system to be set
8	Press 『 Coordinate setup』		● The system calculates measurement results and assign them to the selected coordinate system

8.2.3 Circle Center Measurement

This measurement mode determines the circle center through three points and sets the circle center as the zero point of workpiece coordinate. (A, B and C in the specification represent three points on the circle)

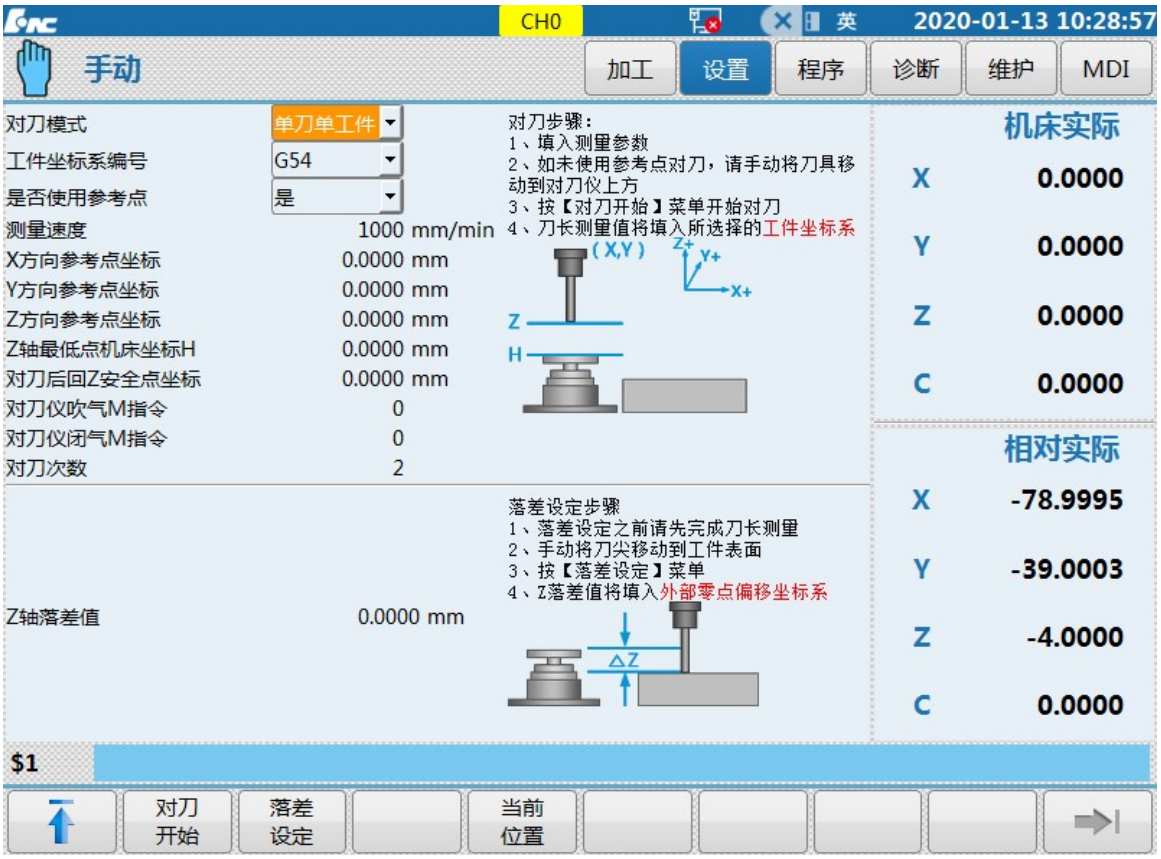
Operation name	Circle center measurement	Working mode	Jog, handwheel
Basic requirements	The system is allowed to run under jog mode and handwheel mode	Display interface	See Chapter 3 "Workpiece measurement" interface

SN	Operation steps	Key	Description
1	Press 〖Setup〗		● Enter the main interface of "Setup" function set
2	Press 『Workpiece measurement』		● Enter the default interface of "Workpiece measurement" function
3	Press 『Circle center measurement』		● Switch to the plane measurement function interface
4	Select coordinate plane		● Select and confirm arc plane using 「Cursor」 and 「Enter」; ● Display of coordinate axis of A, B and C points changes with plane setup.
5	Move the tool to A point of the circle	--	● Move the tool to A point manually (the tool touches the arc of workpiece); ● A, B and C points can be any unrepeated points on the arc and should be distributed evenly.
6	Press 「Cursor」		● Select A point setup display. 
7	Press 『Read measurement』 soft key		● Read the machine coordinates when the tool is at A point.
8	Read coordinates of the points B and C		● Repeat steps 5, 6 and 7.
9	Select coordinate system		● Select the coordinate system to be set.
10	Press 『Coordinate setup』		● The system calculates the measurement results and assign them to the selected coordinate system.

8.3 Automatic Tool Setting

"Automatic tool setting" submenu under "Setup" function set can measure tool length automatically. The "Auto tool setting" function includes single tool single workpiece measurement mode, single tool multiple workpiece measurement mode and multiple tools multiple workpiece measurement mode. The function interface is shown below. In the multiple tools

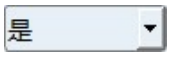
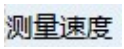
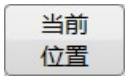
multiple workpiece measurement mode, the multiple tools can be replaced automatically and the measured value of tool length can be saved in the tool compensation interface.

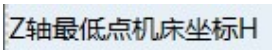
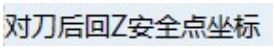


8.3.1 Single Tool Single Workpiece Measurement

Operation name	Single tool single workpiece measurement	Working mode	Jog, handwheel
Basic requirements	The machine tool is equipped with the environment of tool setter test	Display interface	"Auto tool setting" sub-interface

SN	Operation steps	Key	Description
I. Preparation for tool setting			
1	Press 〖Setup〗	设置 Set Up	● Enter the default interface of "Setup" function set and level 1 menu.
2	Press 『Auto tool setting』	自动 对刀	● Enter the "Auto tool setting" function interface.

3	Press 「Cursor」		● Move the cursor to "Tool setting mode". 
4	Press 「Enter」		● Activate "Tool setting mode" drop-down box.
5	Select "Single tool single workpiece"		● Select "Single tool single workpiece".
6	Press 「Enter」		● Confirm the selection of "Tool setting mode".
7	Select workpiece coordinate system		● Select the set workpiece coordinates corresponding to the tool; ● Perform the selection and confirmation as per the steps 3, 4, and 6 above.
8	Select whether to use the reference point		● When the position of tool setter has never been set, select and use the reference point; ● If "No" is selected, the steps of tool setter presetting will be skipped; ● Perform the selection and confirmation as per the steps 3, 4, and 6 above.
9	Input measurement speed		● Select suitable measurement speed according to Z axis height of the reference point, ● When the tool touches the tool setter, it is set as F50 by default; ● Perform the selection and confirmation as per the steps 3, 4, and 6 above.
II. Presetting of initial position for tool setting			
10	Move the tool to the preset position of tool setter manually	Handwheel or jog axis movement key	● "Coordinates of reference point in X, Y and Z directions" preset the initial position of tool setting. During automatic tool setting, all tools quickly reach this position from the safe point and then touch the tool setter at the measurement speed; ● On X and Y axes, try to make the tool in the center of tool setter; ● On Z axis, ensure all tools do not touch the tool setter while reaching this position.
11	Press 「Cursor」		● Move the cursor to "Reference point coordinates in X direction"
12	Press 『Current position』		● The system sets X axis position of current tool as the X axis position of the tool setter in the machine coordinate system; ● Then, the positions of Y and Z axes remain

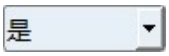
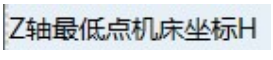
			unchanged; ● The value also can be set manually based on steps 3, 4 and 6.
13	Repeat steps 11 and 12	--	● Set reference point coordinates in Y and Z directions respectively.
III. Safe position presetting			
14	Press 「Cursor」		● Move the cursor to "Machine tool coordinates H of the lowest point on Z axis" 
15	Input machine tool coordinates H of the lowest point on Z axis	---	● The lowest position of spindle, ensuring all tools can touch the tool setter; ● The shortest tool can make the upper surface of the tool setter be pressed down within 5mm distance; ● Input mode: "Jog" and "Current position".
16	Press 「Cursor」		● Move the cursor to "Return to Z safe point after tool setting". 
17	Input tool setting and return to coordinates of Z safe point	--	● This position is where the tool reaches rapidly after tool changing; ● This position should ensure that all tools cannot touch the tool setter; ● Input mode: "Jog" and "Current position".
IV. Auxiliary action presetting			
18	Input the command for the tool setter to blow and hold the air		● Set if the tool setter has blowing function; otherwise, it is 0 by default.
19	Input tool setting times		● Refer to the slow-speed tool setting times. The collision of the tool with the tool setter at the first rapid drop is not counted.
V. Automatic tool setting and drop setup			
20	Press 『Start tool setting』		● Start tool setting; ● After tool setting, save the measured value in the workpiece coordinate system; ● If an alarm is given in the process of measurement, the measurement is stopped, and the measurement is performed again after check and debugging.
21	Move the tool nose to the position where Z axis of the workpiece coordinate system is 0 in handwheel mode	--	● "Drop setup" is to set the drop between the upper surface of tool setter and zero position of Z axis of workpiece coordinate; ● This operation requires that the tool reaches Z0 of workpiece coordinate accurately;

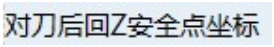
			● If the tool cannot reach Z0 of workpiece coordinate, treat after "Drop setup".
22	Press 『Drop setup』		● Drop is set when no machine tool runs. After calculation, drop of Z axis is set in the offset coordinates of external zero point; ● If the tool cannot reach the workpiece coordinates Z0 position, but coordinates of the tool on Z axis in the workpiece coordinate system can be accurately identified (such as "a"), input "-a" in incremental mode based on the external zero offset set.

8.3.2 Single Tool Multiple Workpiece Measurement

Operation name	Single tool multiple workpiece measurement	Working mode	Jog, handwheel
Basic requirements	The machine tool is equipped with the environment of tool setter test	Display interface	"Automatic tool setting" sub-interface

SN	Operation steps	Key	Description
I. Preparation for tool setting			
1	Press 〖Setup〗		● Enter the default interface of "Setup" function set and level 1 menu.
2	Press 『Auto tool setting』		● Enter the "Auto tool setting" function interface.
3	Press 「Cursor」		● Move the cursor to "Tool setting mode" column. 
4	Press 「Enter」		● Activate "Tool setting mode" drop-down box.
5	Select "Single tool multiple workpiece"		● Select "Single tool multiple workpiece". 
6	Press 「Enter」		● Confirm selection of "Tool setting mode".

7	Select whether to use the reference point		● When the position of tool setter has never been set, select the reference point; ● If "No" is selected, the steps of tool setter presetting will be skipped; ● Perform the selection and confirmation based on the steps 3, 4, and 6 above.
8	Input measurement speed		● Select suitable measurement speed according to Z axis height of the reference point; ● When the tool touches the tool setter, it is set as F50 by default; ● Perform the selection and confirmation based on the steps 3, 4, and 6 above.
II. Initial position presetting for tool setting			
9	Move the tool to the preset position of tool setter manually	Handwheel or jog axis movement key	● "Coordinates of reference point in X, Y and Z directions" presets the initial position of tool setting. During automatic tool setting, all tools quickly reach this position from the safe point and then touch the tool setter at the measurement speed; ● On X and Y axes, try to make the tool in the center of the tool setter; ● On Z axis, ensure all tools do not touch the tool setter while reaching this position.
10	Press 「Cursor」		● Move the cursor to "Reference point coordinates in X direction";
11	Press 『 Current position』		● The system sets X axis position of current tool as the X axis position of the tool setter in the machine tool coordinate system; ● Then, the position of Y and Z axes remains unchanged; ● The value also can be set manually as per steps 3, 4 and 6.
12	Set reference point coordinates in Y and Z directions	--	● Repeat steps 10 and 11 respectively
III. Safe position presetting			
13	Press 「Cursor」		● Move the cursor to "Machine tool coordinates H of the lowest point on Z axis" 

14	Input machine tool coordinates H of the lowest point on Z axis	---	● The lowest position of spindle, ensure all tools can touch the tool setter; ● The shortest tool can press the upper surface of the tool setter down within 5mm distance; ● Input mode: "Jog" and "Current position".
15	Press 「Cursor」		● Move the cursor to "Return to Z safe point after tool setting"; 
16	Input tool setting and return to coordinates of Z safe point	--	● This position is where the tool reaches quickly after tool changing; ● This position should ensure that all tools cannot touch the tool setter; ● Input mode: "Jog" and "Current position".
IV. Auxiliary action presetting			
17	Input the command for the tool setter to blow and hold the air		● Set if the tool setter has blowing function; otherwise, it is 0 by default.
18	Input tool setting times		● Refer to the slow-speed tool setting times s. The collision of the tool with the tool setter at the first rapid drop is not counted.
V. Automatic tool setting and drop setup			
19	Press 『Start tool setting』		● Start tool setting; ● After tool setting, save the measured value in the external zero point Z offset; ● If an alarm is given in the process of measurement, the measurement is stopped, and the measurement is performed again after check and debugging.
20	Select workpiece coordinate system		● Select the set workpiece coordinates corresponding to the tool; ● Select and conform as per the same steps as 3, 4 and 6.
21	Move the tool nose to the position where Z axis of the workpiece coordinate system is 0 in handwheel mode	--	● "Drop setup" is to set the drop between the upper surface of tool setter and zero position of Z axis of workpiece coordinates; ● This operation requires that the tool reaches Z0 of workpiece coordinate accurately; ● If the tool cannot reach Z0 of workpiece coordinate, treat it after "Drop setup".

22	Press 『Drop setup』		● Drop is set when no machine tool runs. After calculation, drop of Z axis is set in the selected workpiece coordinate system; ● If the tool cannot reach the workpiece coordinates Z0 position, but coordinates of the tool on Z axis in the workpiece coordinate system can be accurately identified, such as "a", input "-a" in incremental mode based on Z value of the set coordinate system.
23	Set workpiece coordinate such as G55 and G56	--	● Repeat steps 20-22.

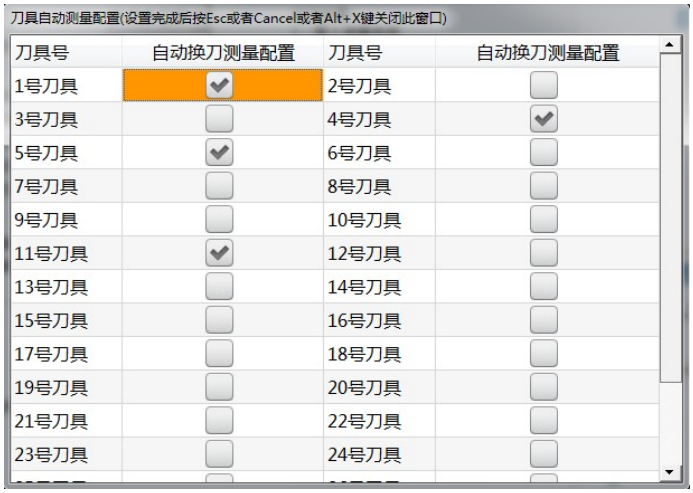
8.3.3 Multiple Tools Multiple Workpiece Measurement

Operation name	Multiple tools multiple workpiece measurement		Working mode	Jog, handwheel
Basic requirements	The machine tool is equipped with the environment of tool setter test		Display interface	"Auto tool setting" sub-interface
SN	Operation steps	Key	Description	
I. Preparation for tool setting				
1	Press 〖Setup〗		● Enter the default interface of "Setup" function set and level 1 menu.	
2	Press 『Auto tool setting』		● Enter the "Auto tool setting" function interface.	
3	Press 「Cursor」		● Move the cursor to "Tool setting mode". 	
4	Press 「Enter」		● Activate "Tool setting mode" drop-down box.	
5	Select "Multiple tools multiple workpiece"		● Select "Multiple tools multiple workpiece". 	
6	Press 「Enter」		● Confirm the selection of "Tool setting mode".	
7	Select whether to use the reference		● When the position of tool setter has never been set, select the reference point;	

	point		● If "No" is selected, skip over steps for presetting the position of tool setting gage; ● Select and conform as per the same steps as 3, 4 and 6.
8	Input measurement speed	测量速度	● Select suitable measurement speed according to Z axis height of the reference point; ● When the tool touches the tool setter, it is set as F50 by default; ● Perform the selection and confirmation as per the steps 3, 4, and 6 above.
II. Presetting of initial position for tool setting			
9	Move the tool to the preset position of tool setter manually	Handwheel or jog axis movement key	● "Coordinates of reference point in X, Y and Z directions" preset the initial position of tool setting. During automatic tool setting, all tools quickly reach this position from the safe point and then touch the tool setter gage at measurement speed; ● On X and Y axes, try to make the tool in the center of tool setter; ● On Z axis, ensure all tools do not touch the tool settier while reaching this position.
10	Press 「Cursor」		● Move the cursor to "Reference point coordinates in X direction"
11	Press 『 Current position』	当前 位置	● The system sets X axis position of current tool as X axis position of the tool setter in the machine coordinate system; ● Then, the positions of Y and Z axes remain unchanged; ● The value also can be set manually as per steps 3, 4 and 6.
12	Set reference point coordinates in Y and Z directions	--	● Repeat steps 10 and 11 respectively.
III. Safe position presetting			
13	Press 「Cursor」		● Move the cursor to "Machine tool coordinates H of the lowest point on Z axis" column. Z轴最低点机床坐标H
14	Input machine tool coordinates H of the lowest point on Z axis	---	● The lowest position of spindle, ensure all tools can touch the tool setter; ● The shortest tool can press the upper surface of the tool setter down within 5mm distance; ● Input mode: "Jog" and "Current position".

15	Press 「Cursor」		● Move the cursor to "Return to Z safe point after tool setting". 对刀后回Z安全点坐标
16	Input tool setting and return to coordinates of Z safe point	--	● This position is where the tool reaches rapidly after tool changing; ● This position should ensure that all tools cannot touch the tool setter; ● Input mode: "Jog" and "Current position".
IV. Auxiliary action presetting			
17	Input the command for the tool setter to blow and hold the air	对刀后回Z安全点坐标 对刀仪吹气M指令	● Set if the tool settere has blowing function; otherwise, it is 0 by default.
18	Input tool setting times	对刀次数	● Refer to the slow-speed tool setting times. The collision of the tool with the tool setter at the first rapid drop is not counted.
V. Automatic tool setting and drop setup			
19	Set tool number for automatic tool setting	自动换刀 选择刀号	● Select the pop-up of tool number list, and select the tool number required for the tool setting; ● Support up to 32 tools; ● Detailed list of tool number is shown below; ● Perform the setting and confirmation based on the steps 3, 4 and 6 above.
20	Press 『Start tool setting』	对刀 开始	● The selected tools touch the tool setter respectively to complete tool setting automatically; ● After tool setting, the measured value is saved in tool length compensation of tool compensation table, and tool compensation number corresponds to the selected tool number; ● If an alarm is given in the process of measurement, stop measurement and set the tools after inspection and commissioning.
21	Select workpiece coordinate system	G54	● Select the set workpiece coordinates corresponding to the tool; ● Perform the setting and confirmation based on the steps 3, 4 and 6 above.
22	Move the tool nose to the position where Z axis of the workpiece coordinate system is 0 under handwheel	--	● "Drop setup" is to set the drop between the upper surface of tool setter and zero position of Z axis of workpiece coordinate; ● This operation requires that the tool reaches Z0 of workpiece coordinates accurately; ● If the tool cannot reach Z0 of workpiece

	mode		coordinates, treat it after "Drop setup".
23	Press 『Drop setup』	落差 设定	● Drop is set when no machine tool movement. After calculation, drop of Z axis is set in the selected workpiece coordinate system; ● If the tool cannot reach the workpiece coordinates Z0 position, but coordinates of the tool on Z axis in the workpiece coordinate system can be accurately identified, such as "a", input "-a" in incremental mode based on Z value of the set coordinate system.
24	Set workpiece coordinates such as G55 and G56	--	● Repeat steps 21-23.



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. This function also can be used to modify default rotate speed of spindle.

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

Operation name	Manual MS	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 『Manual MS』		● F and S setup menus pop up
3	(Set machining F and S values)		● If there are no F and S in machining program, this value prevails ● Set the default S (rotation speed in jog mode)

Note:

- If there is no F/S value in the program, the value in the status bar can be validated immediately.
- If F/S has been set in the program, manual MS is invalid and data in manual MS will be modified by the program.

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 ●The 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 the 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 the upgrade file package ● The upgrade pacakge file name must be BTF file

9	Press 「Enter」		● Start system upgrade ● Do not power off before the upgrade is completed

Note

- **The upgrade should be conducted with the permission of system administrator, and system upgrade is often conducted by HCNC technical personnel;**
- **The system must not be powered off in the upgrade process.**

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 pacakge window on the upper part of the interface
4	Press 「Cursor」		● Select the backup item 
5	Press 「Enter」		● Confirm the selection 
6	Press 「Cursor」		● Select the backup item 
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:

- **When a file is backed up in the system disk, size of the system disk should be noted. Backup size of V2.40.00 software is about 200MB.**

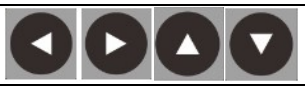
9.2 Batch Commissioning

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

Operation mode and object of "Batch commissioning" function are similar to those of "Data management" function. 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 file	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 the 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」		● A prompt message "File is successfully loaded, 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』		● 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 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 the backup package name " will be given
10	(Enter the backup package name)	---	● File name must be suffixed with .tar

11	Press 「Y」 or 「Enter」	V Y or Enter 确认	● Complete the backup and give a prompt message Backup succeeds

Note: Do not power off during backup or loading.

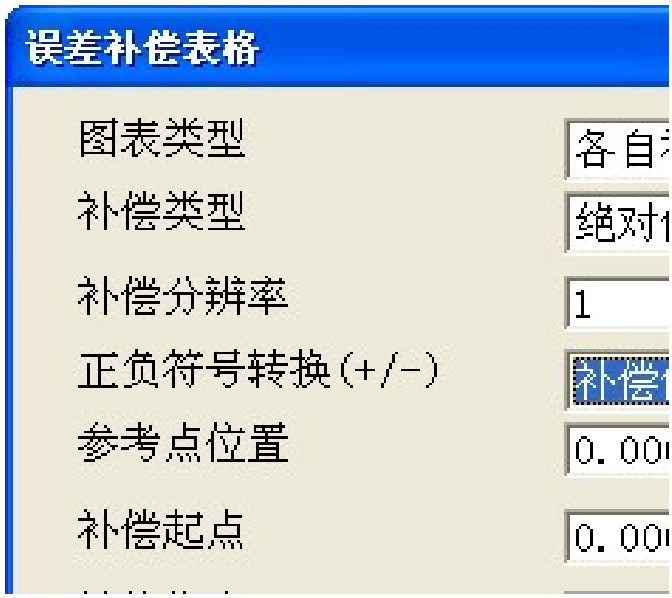
9.3 Pitch Error Compensation

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 the error of actual position and command position through increasing or decreasing actual displacement of machine tool.

The laser interferometer can measure error between the actual position and the command position of machine tool axis and generate error compensation data file. The existing HNC-8D-TU system only supports *.rtl file generated by Renishaw laser interferometer.

9.3.1 Generation Of Pitch Error Compensation Data File

Presently, the system supports direct import of error compensation data file (*.rtl) generated by Renishaw laser interferometer only. When Renishaw software generates the error compensation file, the 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 plus and minus 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" is the compensation interval and it must be a positive value.

9.3.2 Operation of Pitch Error Compensation Sub-interface

HNC-8D-TU system can enter this function sub-interface under "Machining" and "Maintain" function sets.

1) Under "Machining" function set, enter the pitch error compensation sub-interface

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 〖Machining〗		● "Tool compensation setup" sub-interface
2	Press 『→』		● Enter the extension menu
3	Press 『 Parameter setup』		● Enter the "Parameter setup" sub-interface
4	Press 『 Pitch compensation』		● Enter the pitch compensation sub-interface (as shown below)

2) Under "Maintain" function set, enter the pitch error compensation sub-interface

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

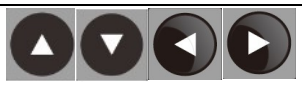
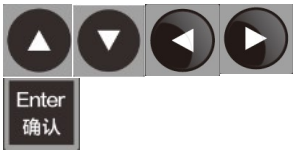
SN	Operation steps	Key	Description
1	Press 〖Maintain〗	维护 Mainte	● "Tool compensation setup" sub-interface
2	Press 『Parameter setup』	参数配置	● Enter the "Parameter setup" sub-interface
3	Press 『Pitch compensation』	螺距补偿	● Enter the pitch compensation sub-interface (as shown below)

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 tpitch error compensation 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 "Tool compensation setup" sub-interface

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」		● Use the arrow keys to select the setting options (the option in the area 1 of the above figure, see notes for specific options) ● Press "Enter" key to confirm the selection or setting
6	Press 『 One-click clear』		● If the pitch error compensation data is not imported for the first time, please press One-click clear to clear thread compensation data ● If thread 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 compensation data file of corresponding axis thread
8	Press 『 USB flash disk』/『User disk』		● Select the compensation data file disk
9	Press 「Cursor」		● Select the compensation data file
10	Press 「Enter」		● Press 「Enter」 to import the pitch error compensation data
11	Press 「Reset」		● After the pitch compensation data is imported successfully, press Reset to take effect
12	(Check thread compensation data)		● Check pitch error compensation type, start point, compensation poins, compensation interval and initial number of data table ● During unidirectional compensation type, check whether backlash compensation is enabled and the backlash value ● If the pitch error compensation data is imported incorrectly, execute from step 4 again

Remarks: Select 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 points: 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 (C)	Nonfreezing at 0- +45
Temperature variation	<1.1°C/min
Relative humidity	90%RH or lower (non-condensable) Normal condition: 75% or smaller Short-term (within a month): No more than 95%
Storage temperature (C)	Nonfreezing at -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 it is aimed at:

- 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 earth wire; otherwise, personal injury or device damage may be caused and exceptional operation of device may be caused.

10.3 Power Supply

Power of HNC-808Di-TU **CNC device** 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 screen is set 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, the user 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 used after left unused for a long time, remove dust and dry it. Then, check connection and grounding, power on for a period of time and ensure the system is faultless before restart.