HNC-8 System Operation Manual

V2.42

Introduction

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

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

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

Note

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

Contents

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 system includes HNC-848. This manual is based on HNC-848 panel type. In case of discrepancies, please refer to the specification provided by the machine tool manufacturer.

1.1 Basic Operation

HNC-848 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 reference point return". During operation of CNC maching, 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.
Reference point return	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-848 CNC device is furnished with 6 function keys "MACH, SET, PROG, DGN, MAINT 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
MACH	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)*.
SET	Functions of tool setting	Tool setting (coordinate system, workpiece measurement, automatic tool setting), tool compensation setup*, tool life management
PROG	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
DGN	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
MAINT	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 series, user-defined key is often set as the MDI function.

1.3 Basic Display Interface

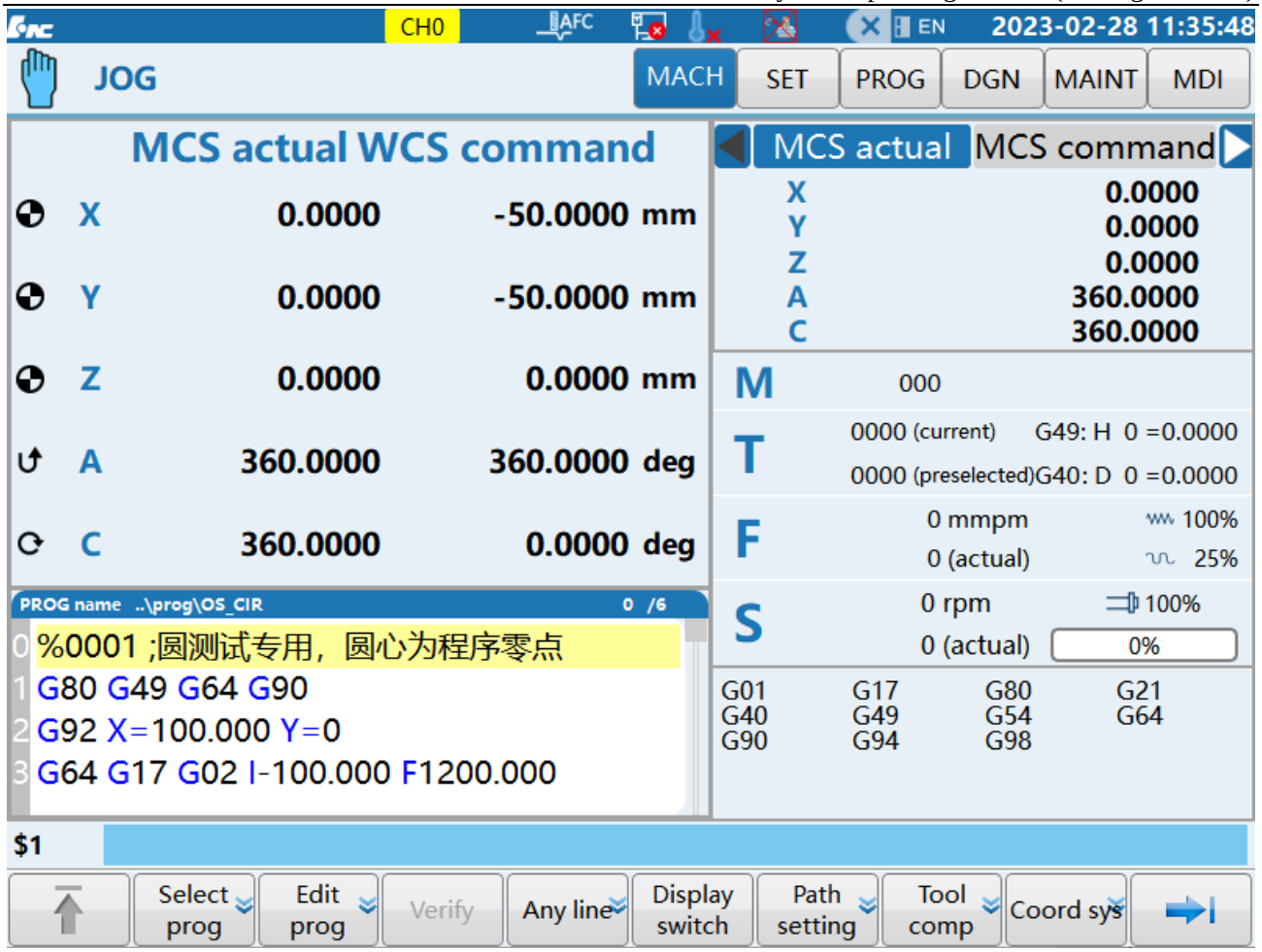
HNC-848 system can realize different application functions through function keys and function soft keys, and display corresponding interfaces. The display interface of this system mainly includes machining display interface, program selection and editing interface, machining setup interface, parameter 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 realize command input, parameter setup and other operations.

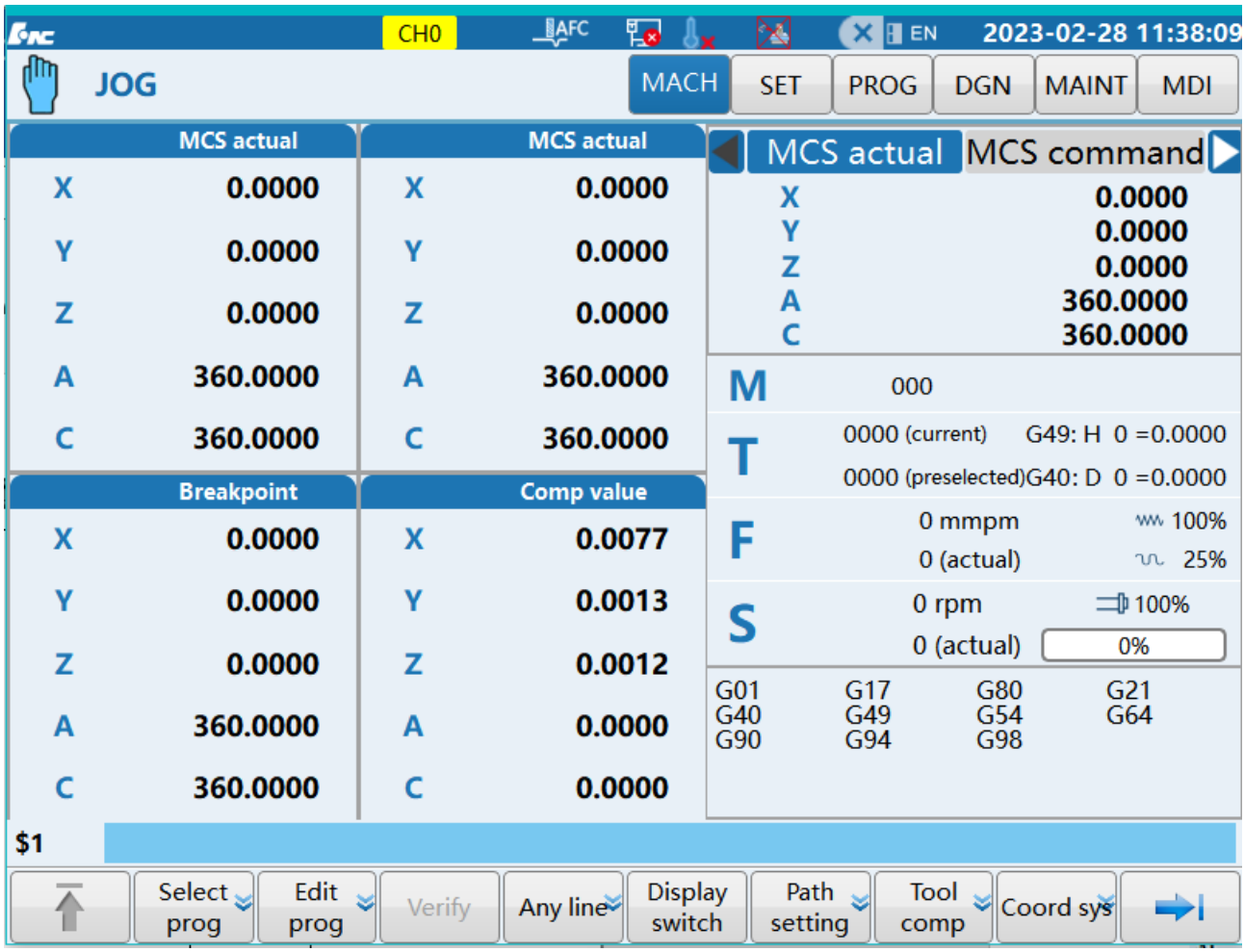
All interfaces are briefly introduced based on HNC-848 standard configuration.

1.3.1 Machining Display Interface

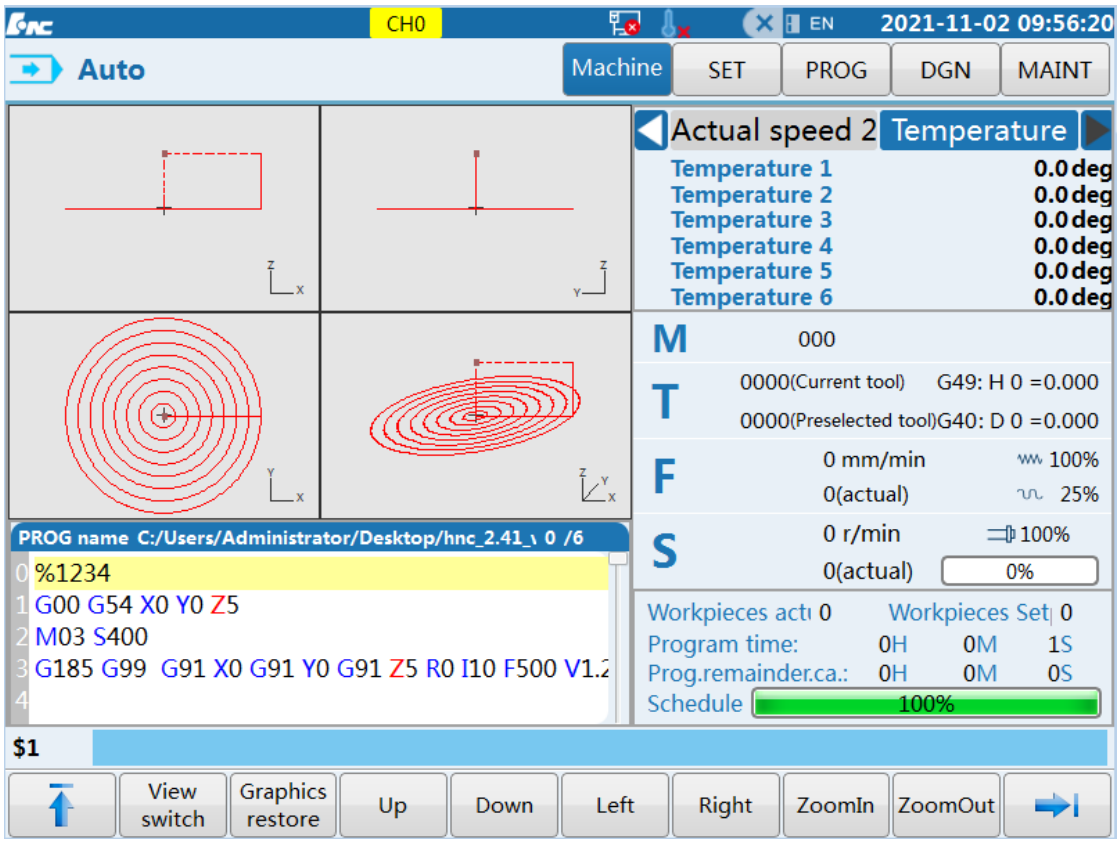
The machining display interface enables the operator to observe the machining process and has 4 display forms: big character coordinate + program, joint coordinate, graphics path + program, and program. These 4 interface can be switched through 『Switch display』 soft key.



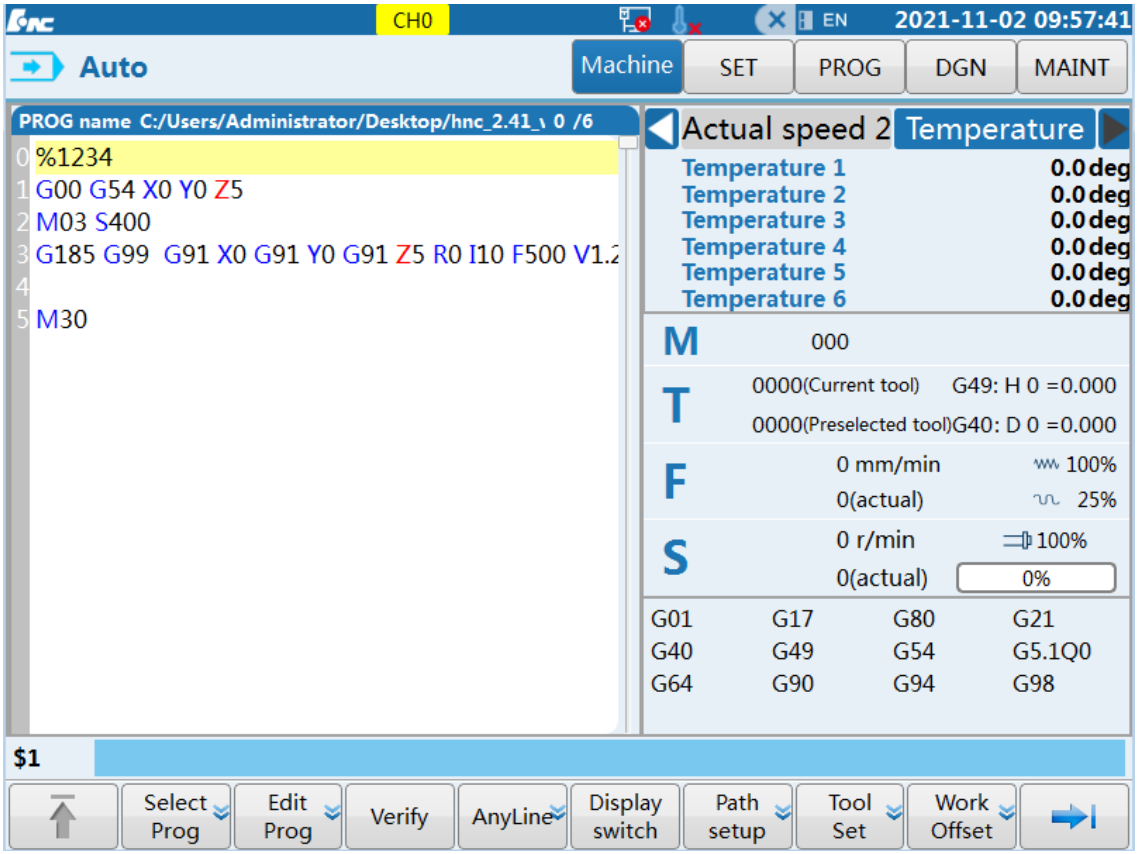
Big character coordinate + program display interface



Joint coordinate display interface



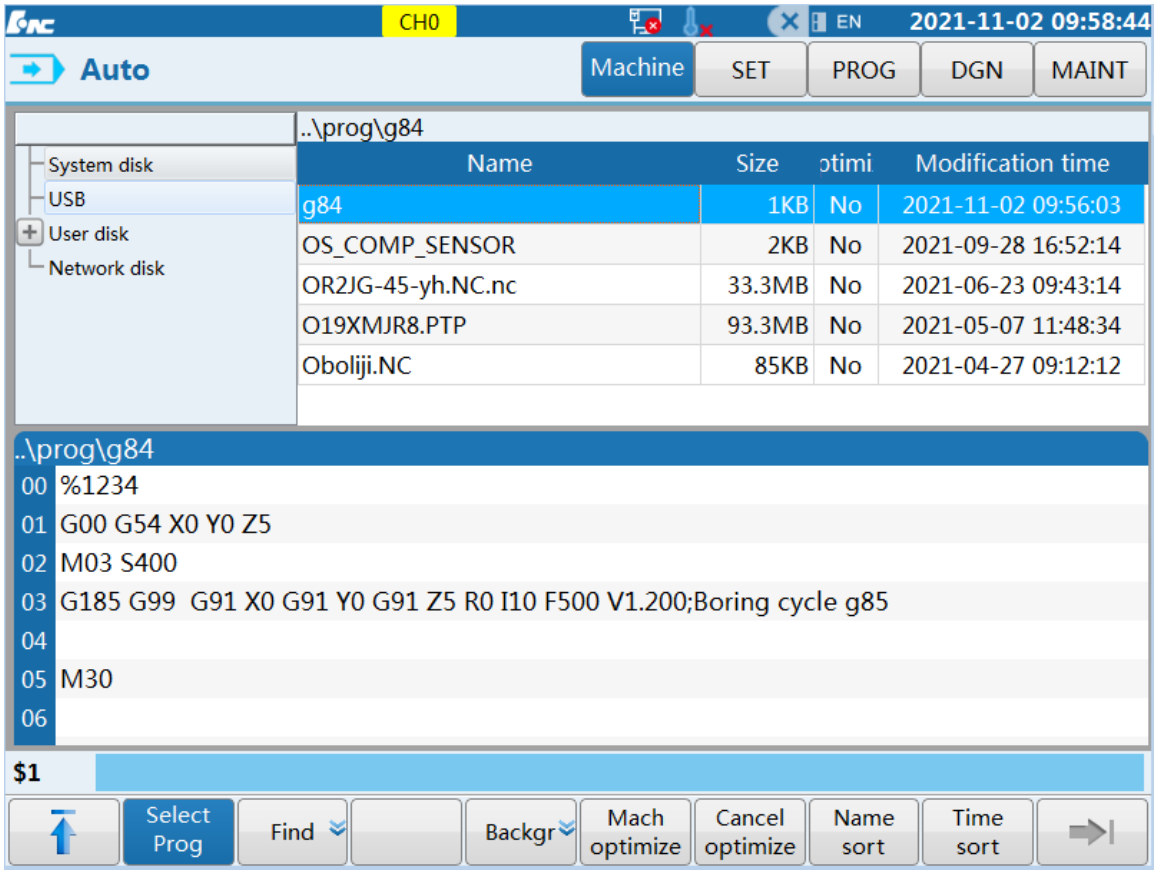
Graphics path + program display interface



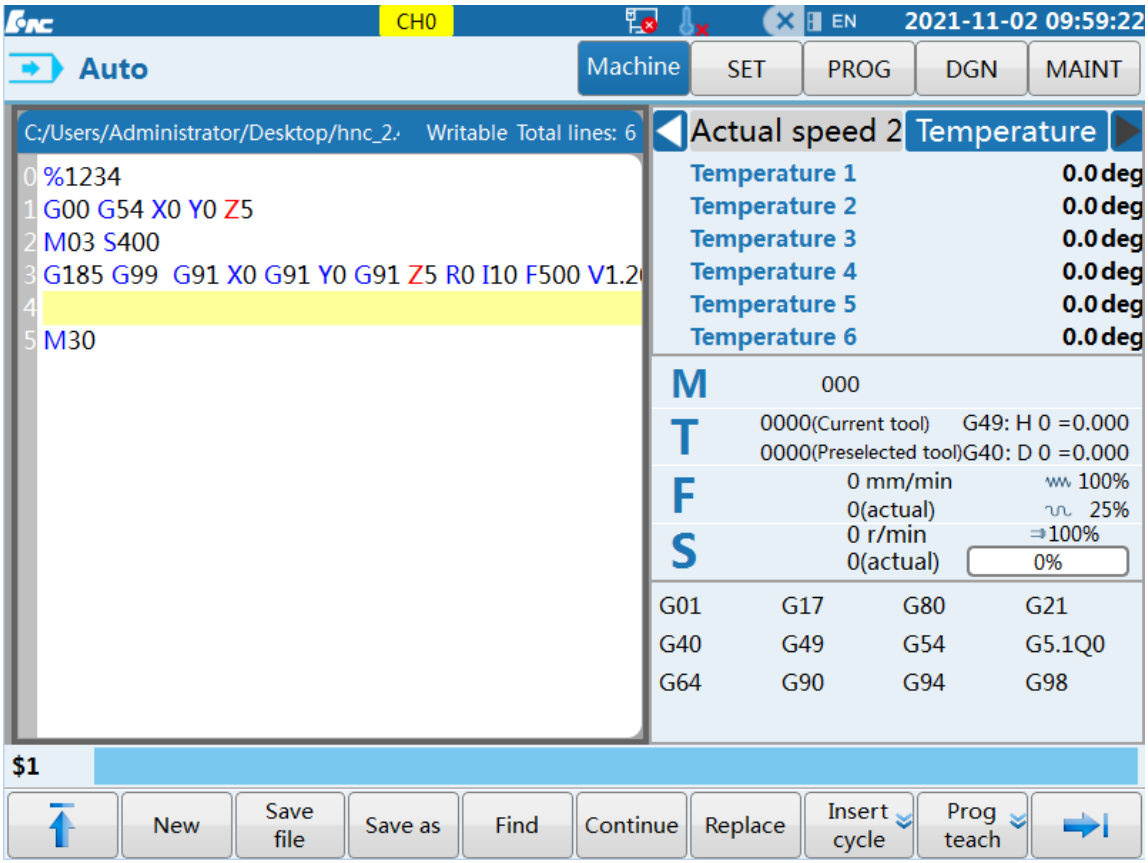
Program display interface

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.

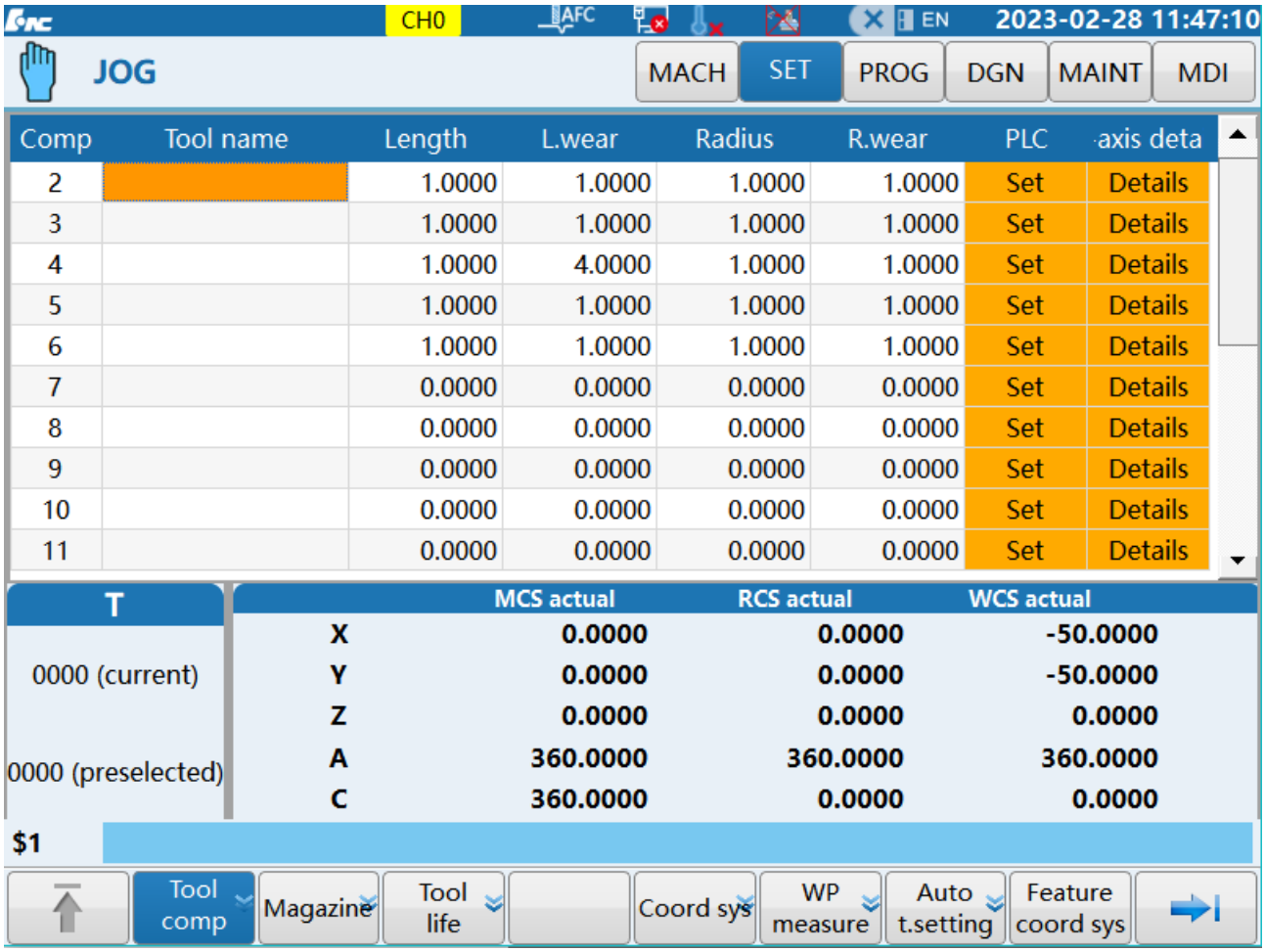


Program selection interface



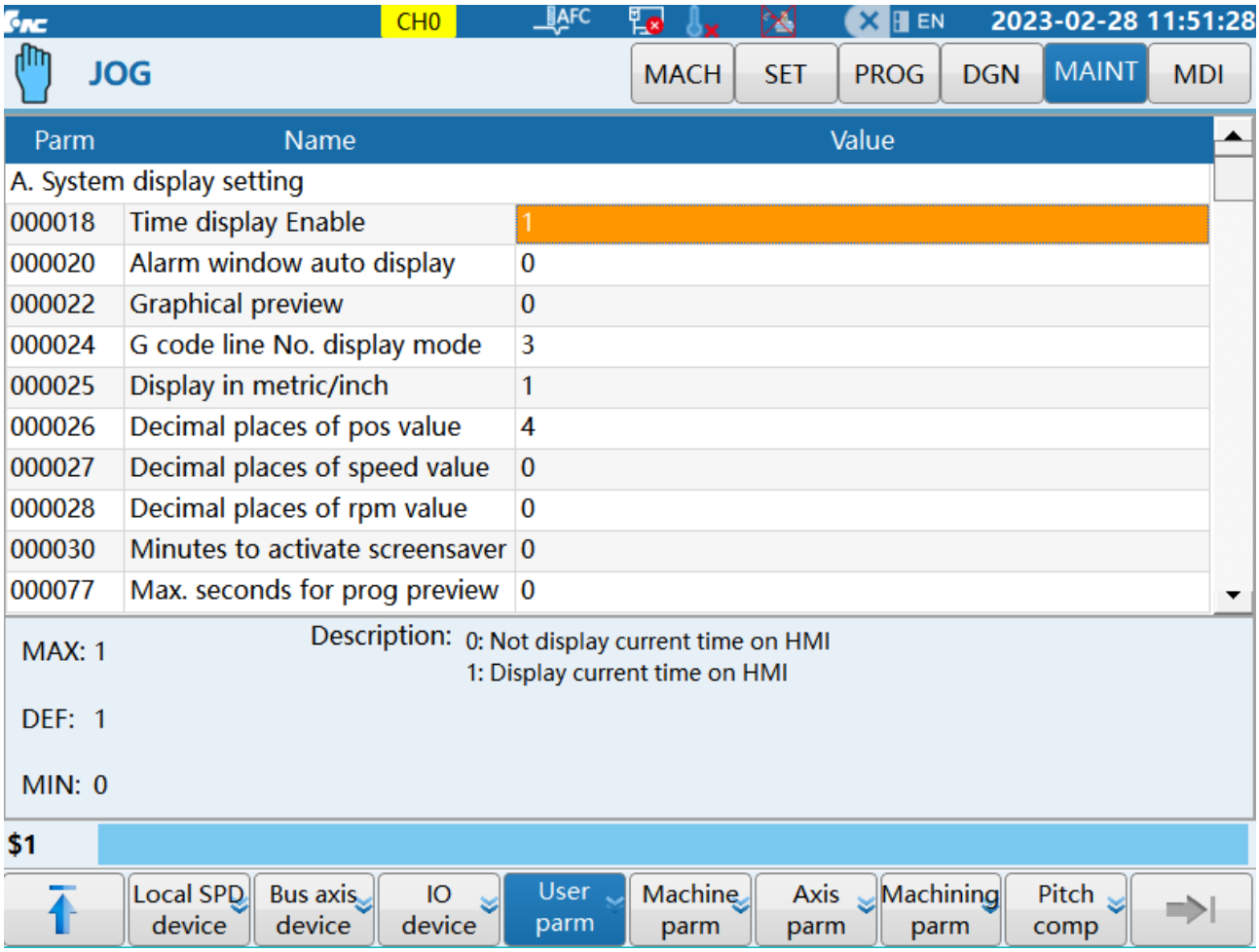
Program editing interface

1.3.3 Machining Setup Interface



Machining setting interface

1.3.4 Parameter Setting Interface

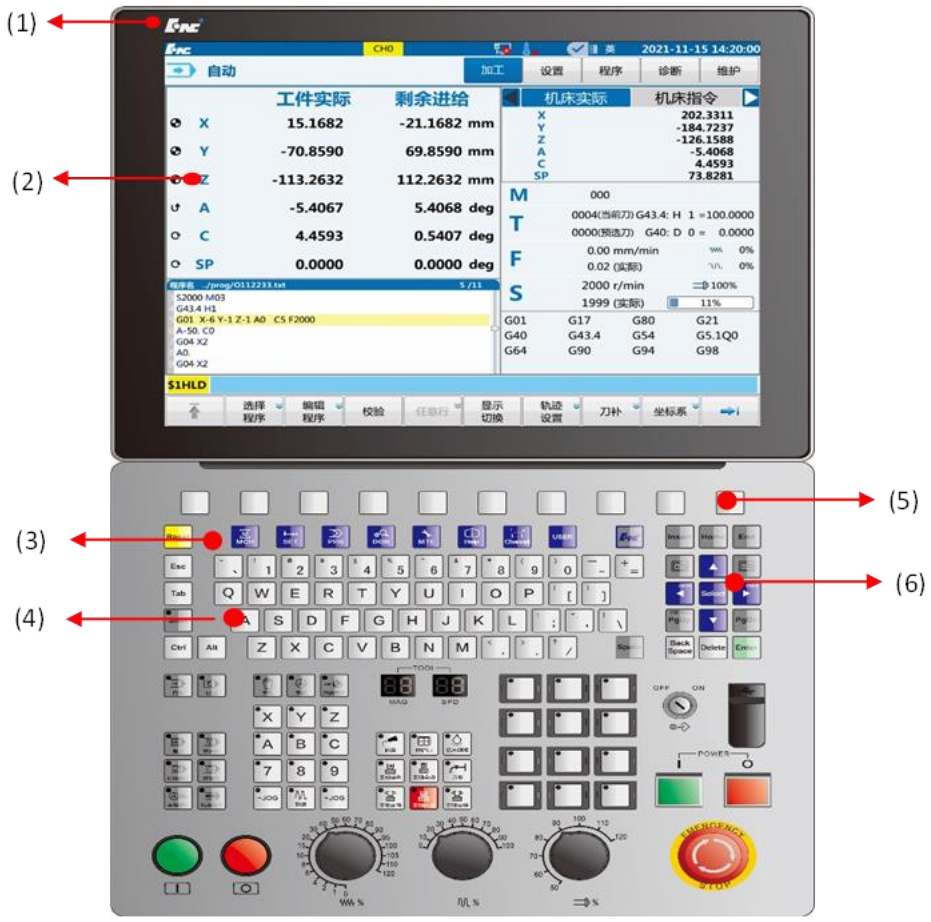


2 Operating Equipment

2.1 System Hosting Panel (NC Panel)

2.1.1 System Hosting Panel Zoning

HNC-848 system panel is 17 in. color LCD (resolution is 1280×1024). Panel zoning is shown below.



(1)---LOGO

(2)---Interface display interface area

(3)---Function button area

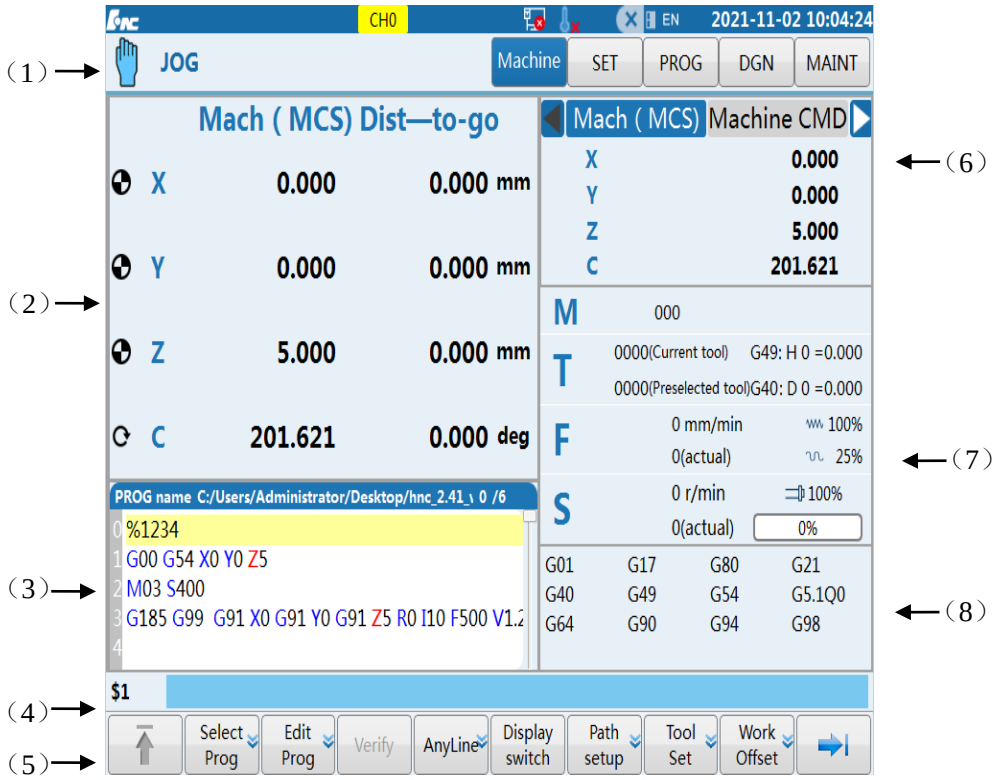
(4)---Number and character key area

(5)---Soft key area

(6)---Cursor area

2.1.2 Display Interface Zoning

The operation interface of HNC-848 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 Hosting Panel Keys

The hosting 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 button function

HNC-848 system has 6 function keys "Machining", "Setting", "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).

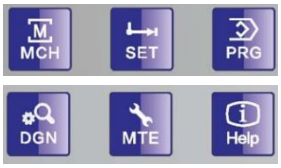
NC keyboard area

Soft key area

2.1.4 Function of Keys on MDI Keyboard

Key	Name/symbol	Functional description

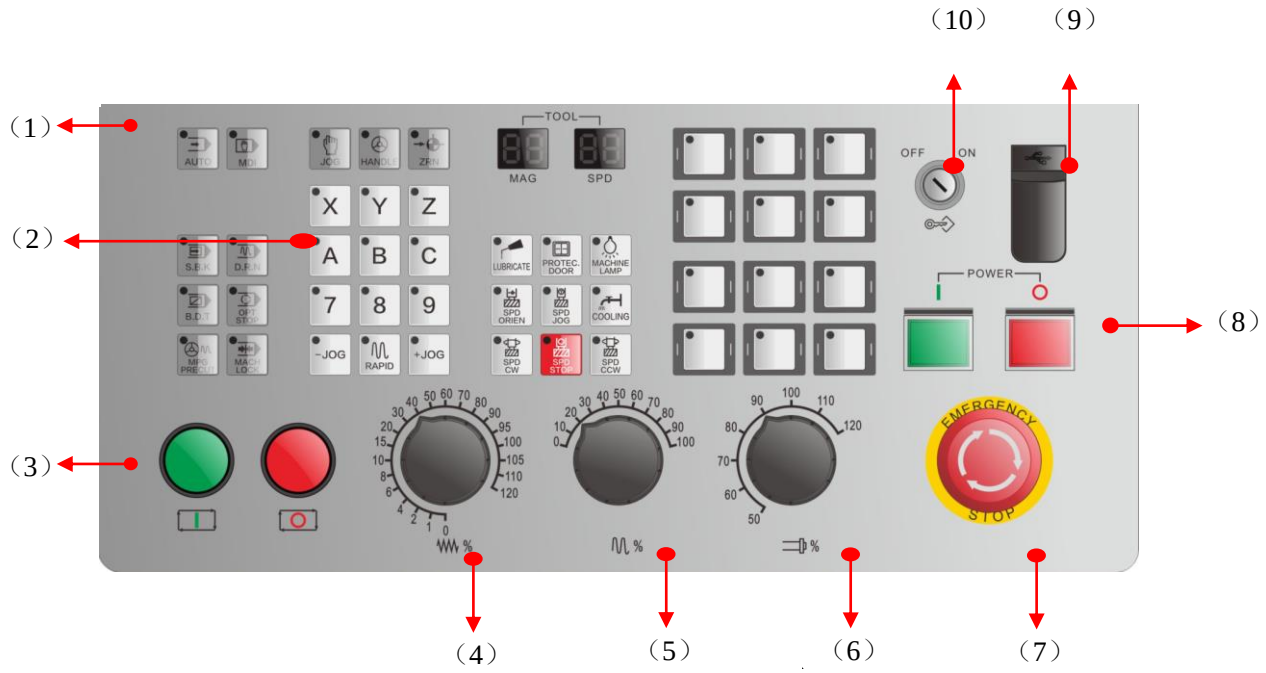
	Character key (letter, number, symbol)/ 「 “Letter” 」 (such as「Y」)	Input letters, numbers and characters. Every character key has upper and lower characters. When the shift key and the character key are pressed simultaneously, the upper character is input; otherwise, the lower character is input.
	Cursor shift key/ 「Cursor」	Control the cursor to move horizontally and vertically.
	Program name symbol key/ 「%」	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 down」	Switch previous and next pages in the same display interface.
	Function key/ 〖MACH〗 〖SET〗 〖PROG〗 〖DGN〗 〖MAINT〗 〖HELP〗	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. Help: Help contents of relevant operation
	Soft key/ 『↑』 『⇨』 『 “Function” 』	There are 10 unidentified keys below HNC-848 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 (MCP Panel)

2.2.1 Operation Panel Zoning



(1)---Working mode selection button

(2) ---Feed axis movement control button

(3)---Cycle start/feed hold

(4)---Feedrate override switch

(5)---Rapid traverse magnification control button

(6)---Spindle magnification switch

(7)---Emergency stop button

(8)---Power supply on/off button

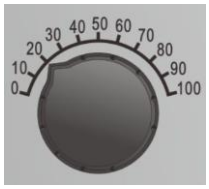
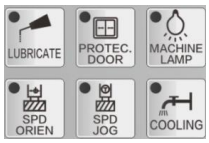
(9)---USB interface

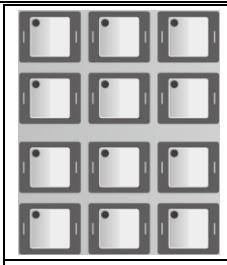
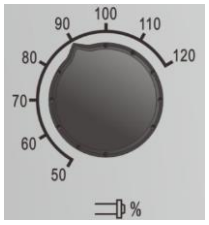
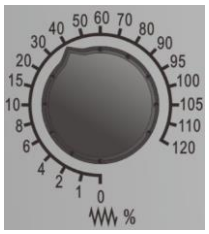
(10)---Edit lock

2.2.2 Definition of Operation Panel

This manual describes function and status of all keys based on standard PLC of HNC-848 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 /【Reference point return】	Select the reference point return mode key.	Reference point return
	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 /【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)
	Dry run Working mode key /【Dry run】	Select the dry run working mode	Auto
	Block skip /[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	Auto, MDI (Including single block)

		to continue running the subsequent programs. If this key is not pressed, consistently run the program.		
	Handwheel precut /【Handwheel】	1) Whether to enable the handwheel precutting function		Auto, MDI (Including single block)
	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, reference return, auto, MDI (including single block and handwheel simulation)
	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 axis runs 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	Machine tool lighting, lubrication, air blowing, spindle orientation, spindle jogging	Handwheel, incremental, jog, reference return, auto, MDI (including single block and handwheel simulation)
			Protective door	Auto

	Machine control extension key / [Machine control]	Manual control of auxiliary actions of machine tool.	Handwheel, incremental, jog, reference return, auto, MDI (including single block and handwheel)
	Program protection switch / [Program protection]	Protect program from being modified arbitrarily.	Handwheel, incremental, jog, reference return, auto, MDI (including single block and handwheel simulation)
	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.	Jog, auto, MDI, reset
	Feedrate knob / [Feedrate]	Feedrate override.	Jog, auto, MDI, reference return
	System power-on / [Power-on]	Control power-on of CNC device.	Handwheel, incremental, jog, reference return, auto, MDI (including single block and handwheel)
	System power-off / [Power-off]	Control power-off of CNC device.	

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 return key, and continued 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 enabling 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

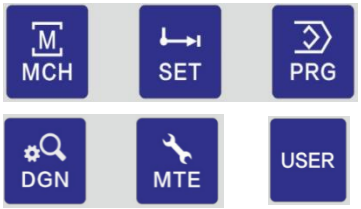
Key	Name/symbol	Functional description	Working mode at effective state
	Handwheel / [Handwheel]	Control movement of machine tool. (When handwheel function is valid, it can control the machine tool to move based on the programmed path).	Handwheel
	"MPG enable OFF" switch / [Enable OFF]	When the switch 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]\[4] \[5]\[6]	When the switch is turned to the axis selection position except “OFF”, all switches and keys on the handheld unit are valid.	Handwheel
	Handwheel magnification switch / [Incremental magnification]	The 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	Handwheel

		once.	
	Emergency stop button / [Emergency stop]	When the handwheel is valid, in case of an emergency, the system and the machine tool immediately enters halt state, and all outputs are turned off.	Handwheel, incremental, jog, reference return, 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.

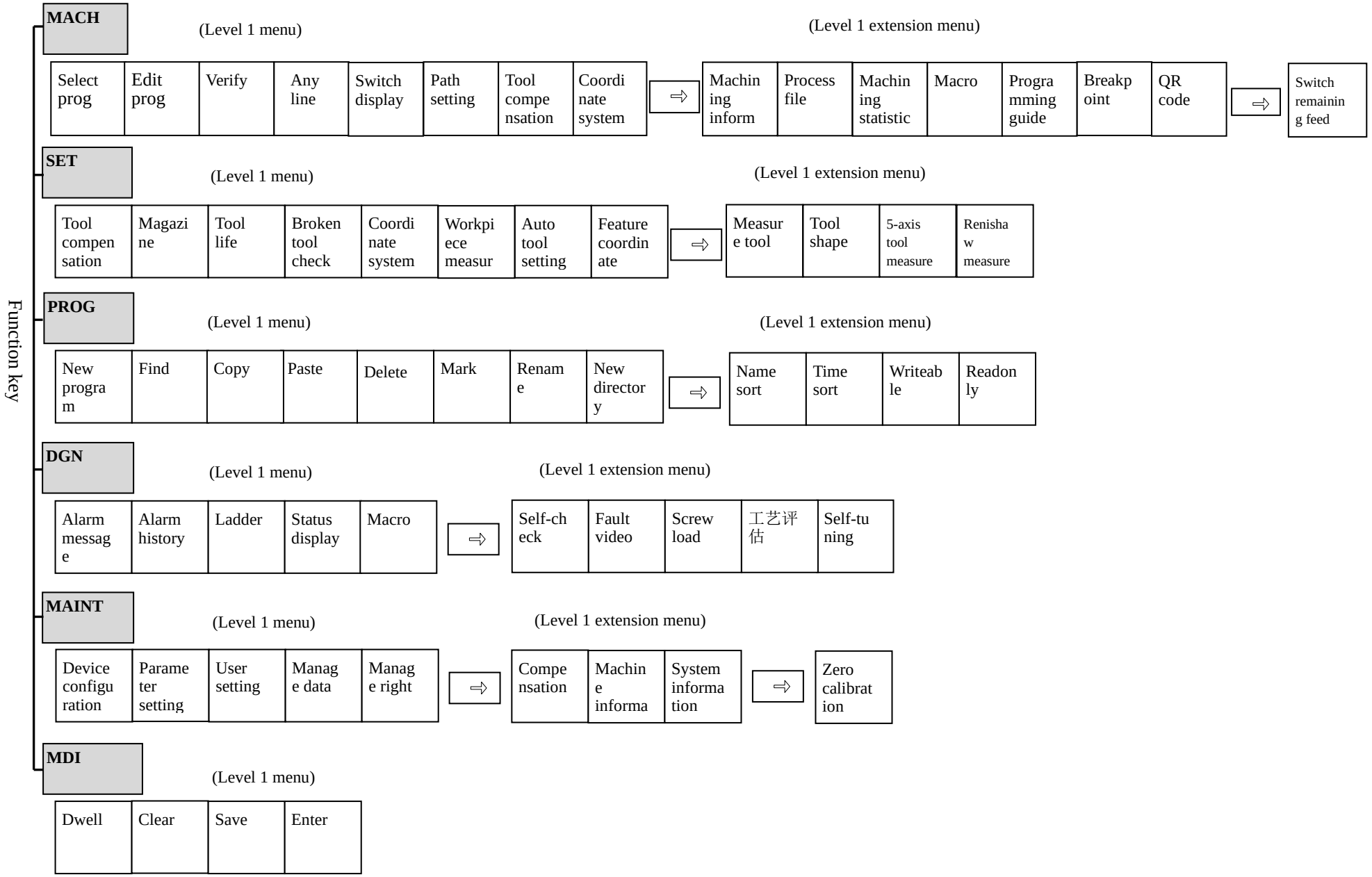
11) Exit mode of man-machine dialog box:

- 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

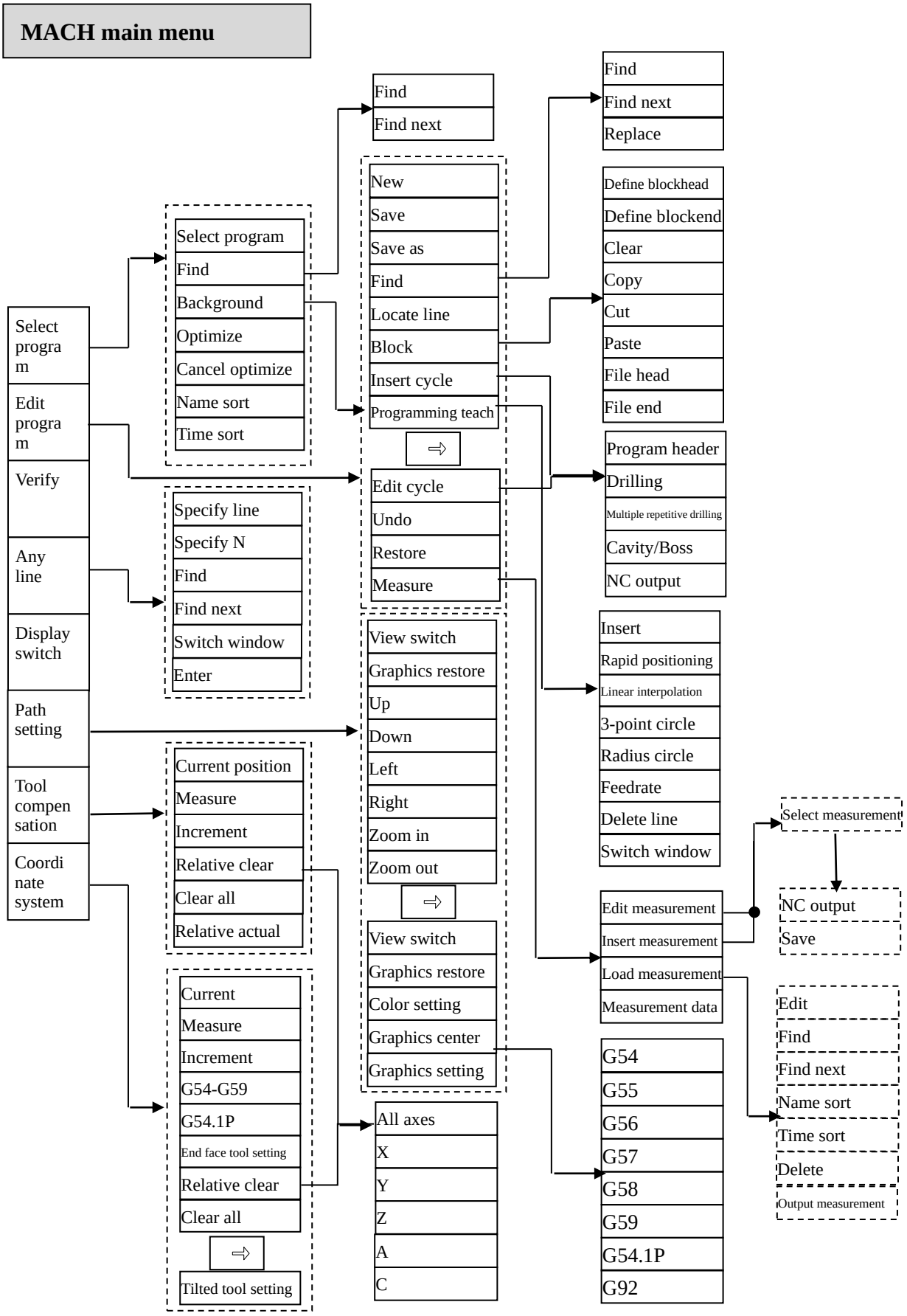
The menu tree is the basic structure diagram of the standard version of the system. The increase, decrease, sorting or position of the function menu will vary depending on the user's permission, parameter settings and the machine tool manufacturer. For details, refer to the specification provided by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

1) Level 1 menu of function sets

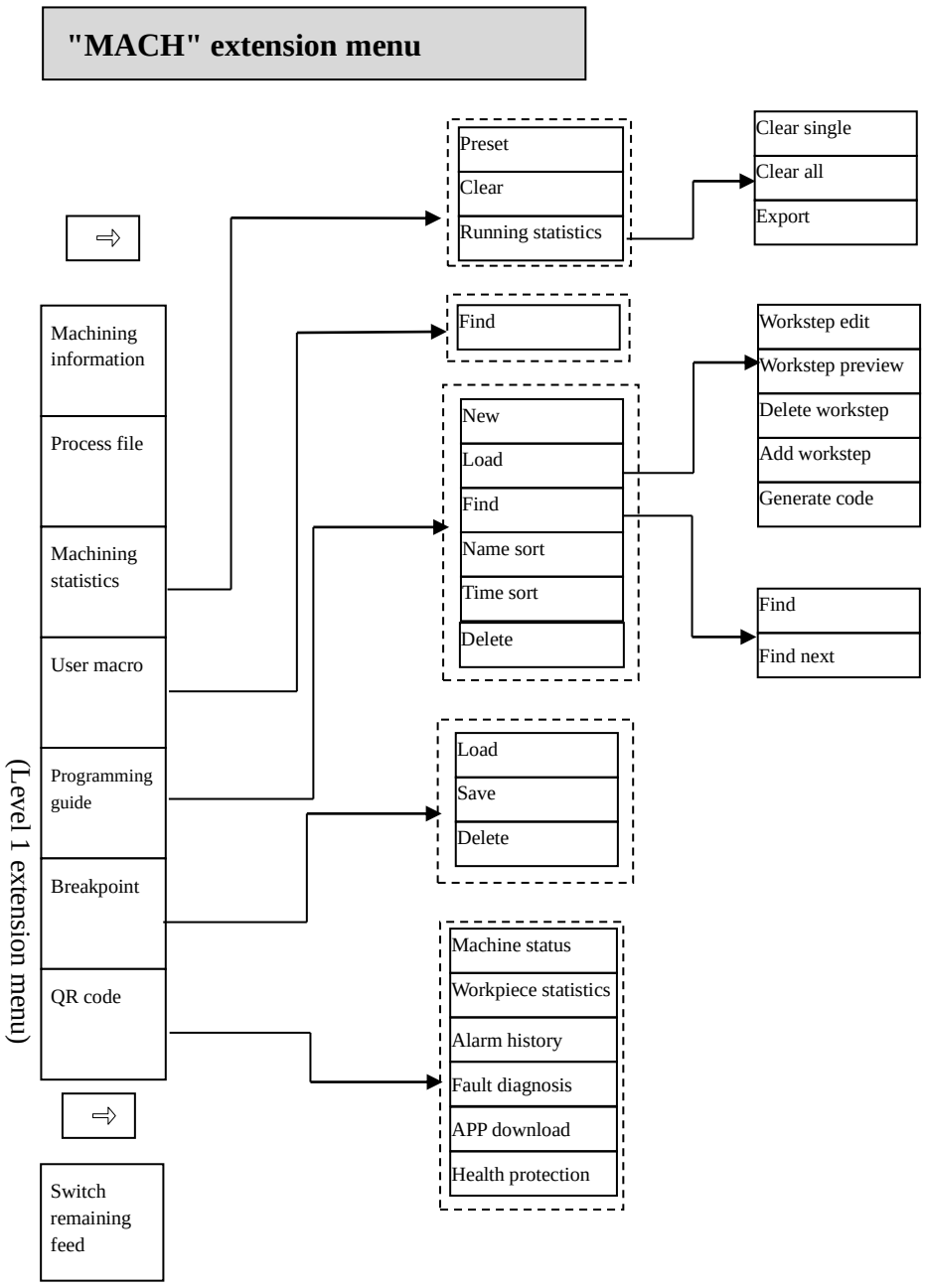

Function key
MACH
(Level 1 menu)
(Level 1 extension menu)
Select prog
Edit prog
Verify
Any line
Switch display
Path setting
Tool compensation
Coordinate system
Machining inform
Process file
Machining statistic
Macro
Programming guide
Breakpoint
QR code
Switch remaining feed
SET
(Level 1 menu)
(Level 1 extension menu)
Tool compensation
Magazine
Tool life
Broken tool check
Coordinate system
Workpiece measur
Auto tool setting
Feature coordinate
Measure tool
Tool shape
5-axis tool measure
Renishaw measure
PROG
(Level 1 menu)
(Level 1 extension menu)
New program
Find
Copy
Paste
Delete
Mark
Rename
New directory
Name sort
Time sort
Writeable
Readonly
DGN
(Level 1 menu)
(Level 1 extension menu)
Alarm message
Alarm history
Ladder
Status display
Macro
Self-check
Fault video
Screw load
工艺评估
Self-tuning
MAINT
(Level 1 menu)
(Level 1 extension menu)
Device configuration
Parameter setting
User setting
Manage data
Manage right
Compensation
Machine informa
System information
Zero calibration
MDI
(Level 1 menu)
Dwell
Clear
Save
Enter

2) Menu structure of "Machining" function set

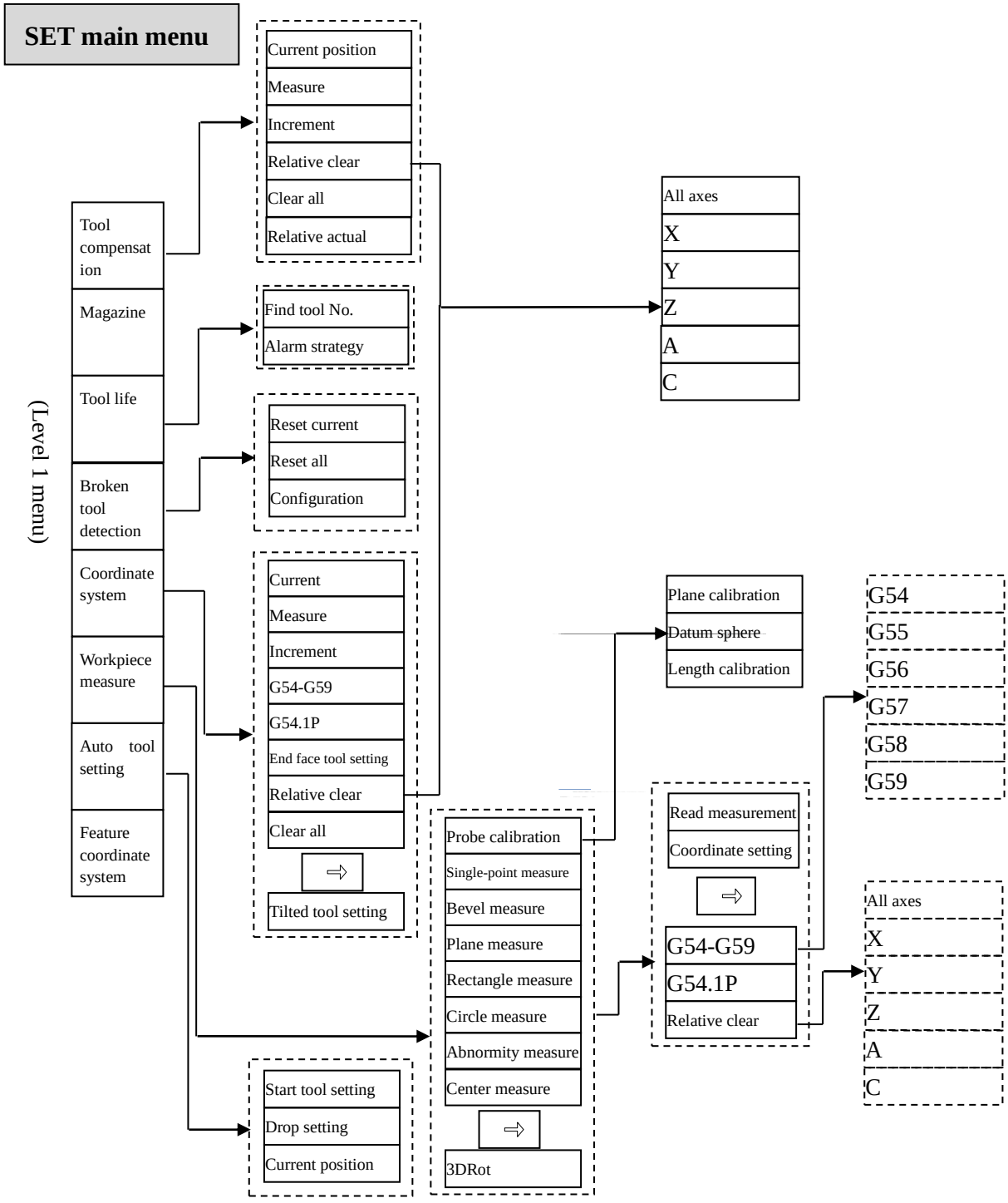
(1) "Machining" main menu



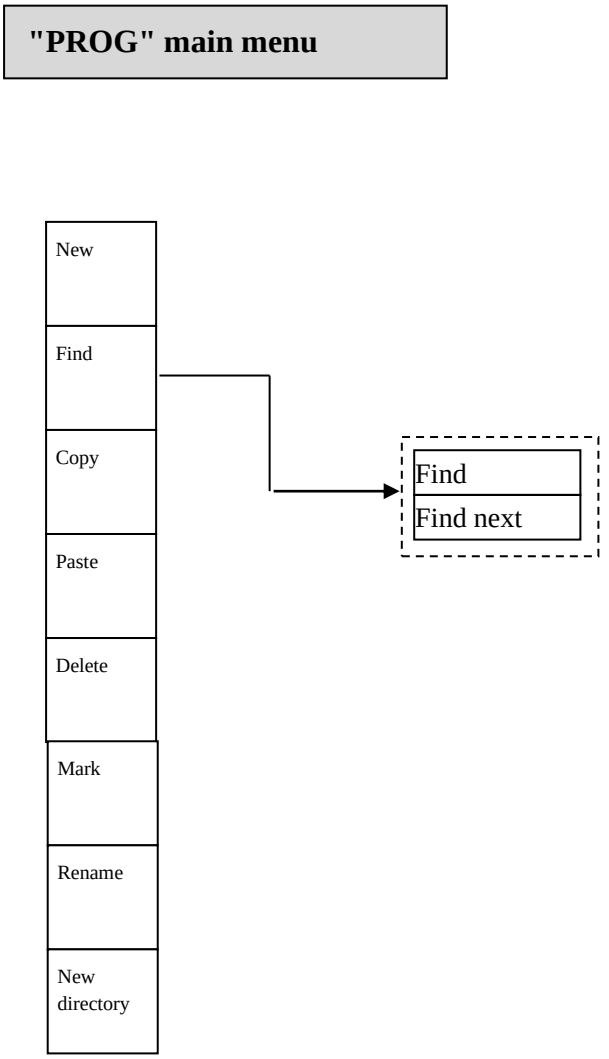
(2) "Machining" extension menu



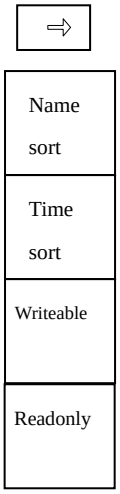
3) Menu structure of "Setup" function set



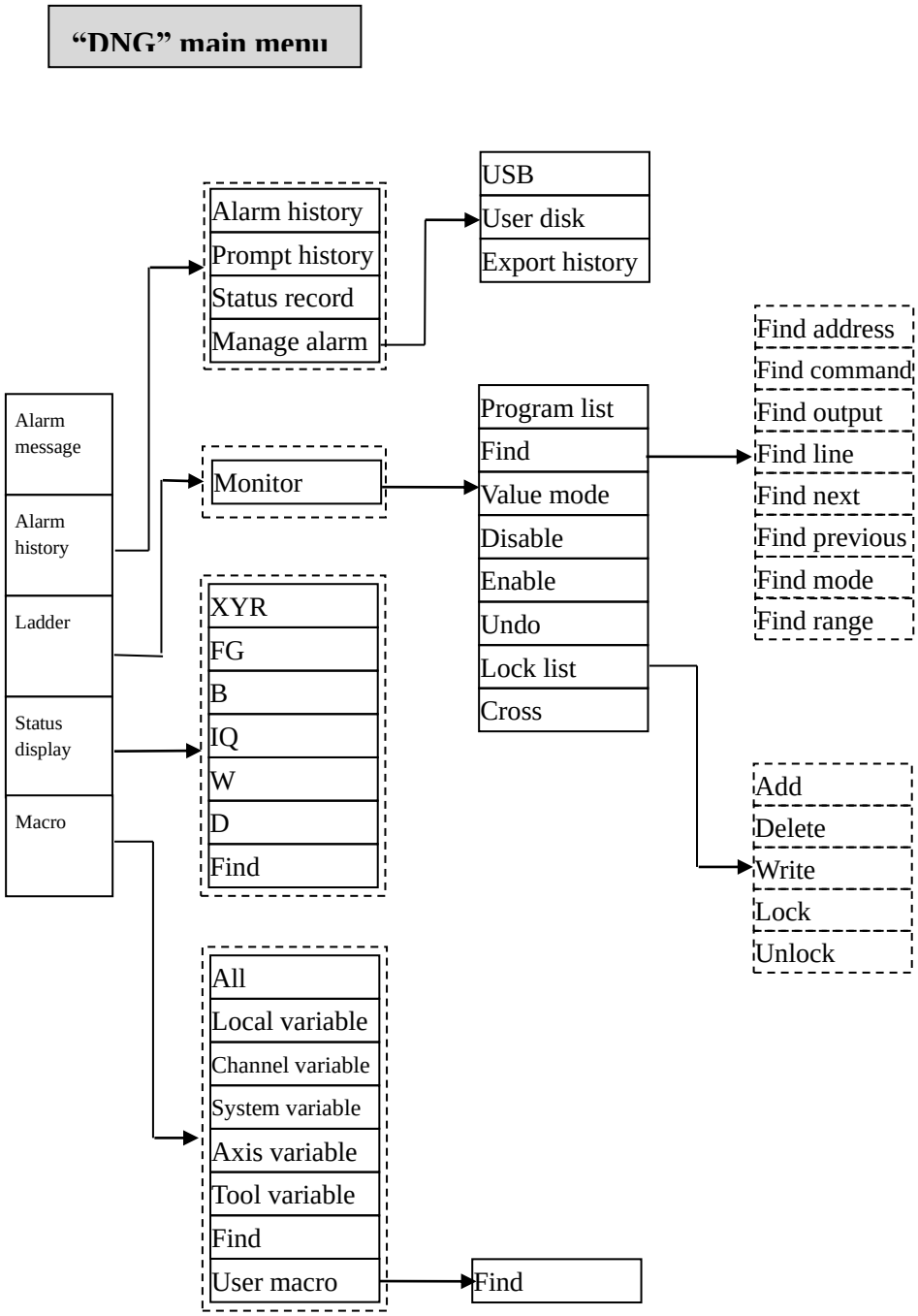
4) Menu structure of "Program" function set



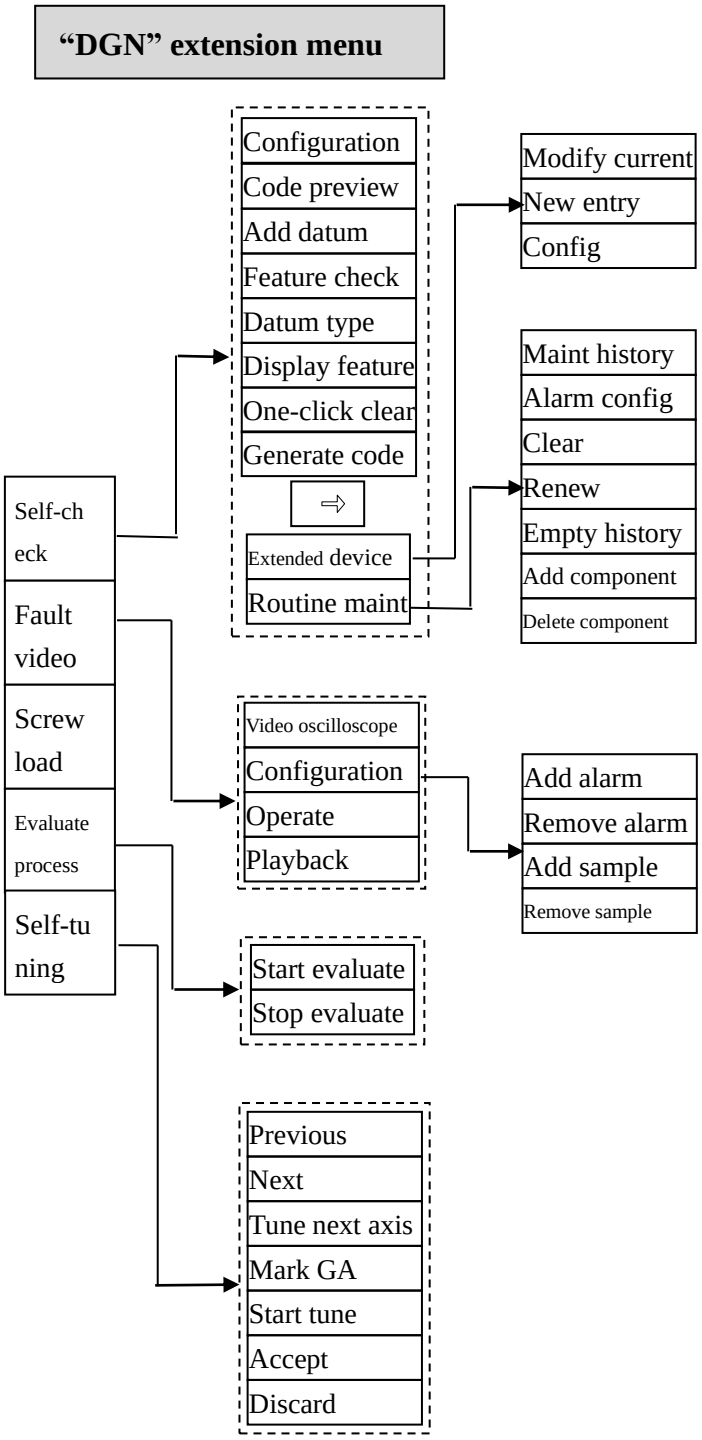
"PROG" extension menu



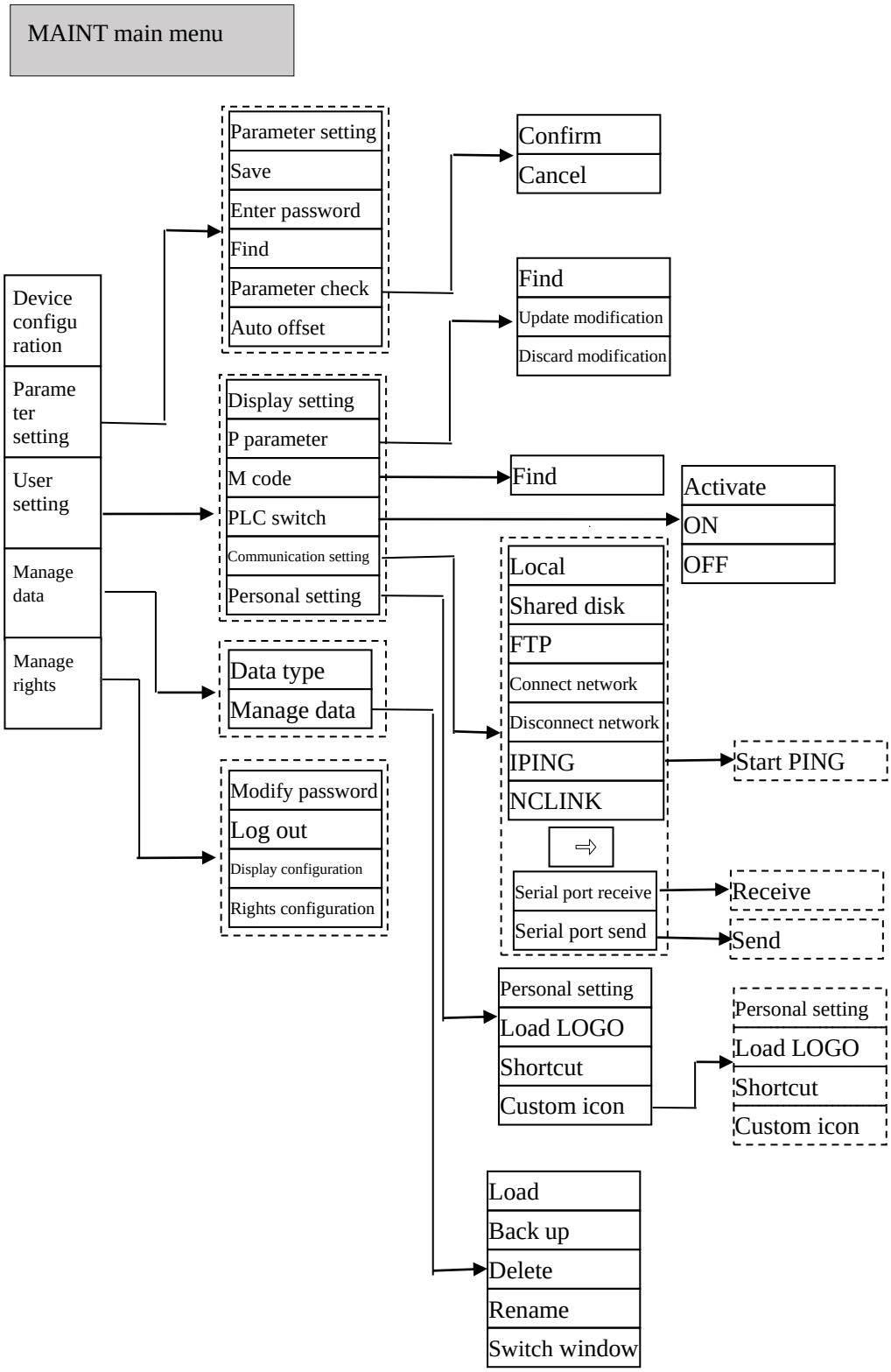
5) Menu structure of "Diagnosis" function set



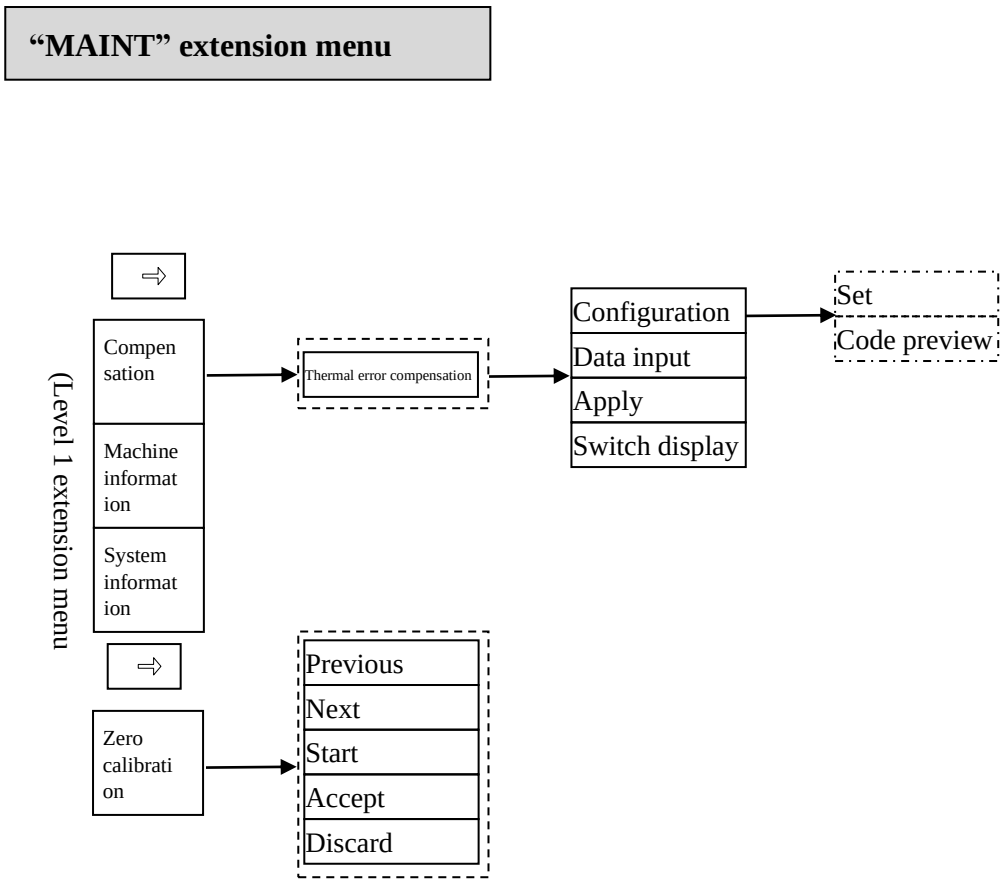
6) Menu structure of "Diagnosis" function set



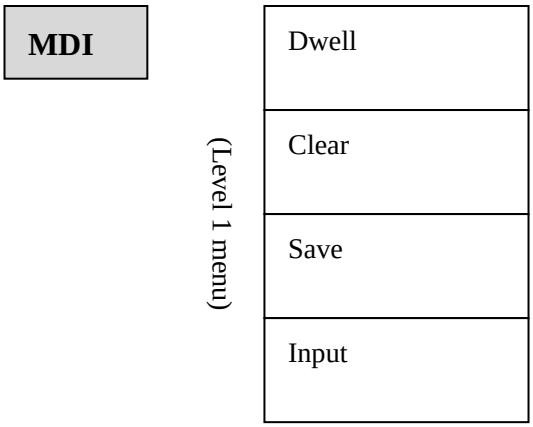
6) Menu structure of "Maintain" function set



6) Extension menu of "Maintain" function set



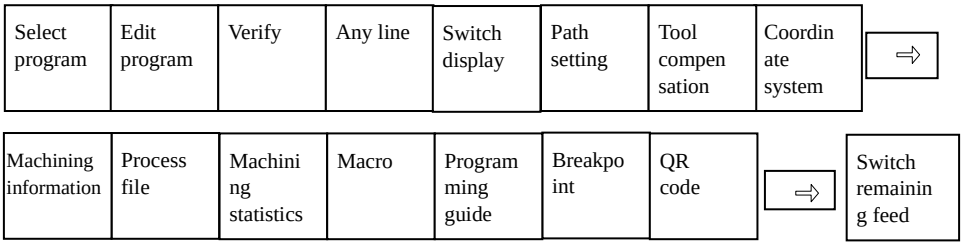
7) Menu structure of "User-defined (MDI)" function set



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 "SET", "PROG" and "DGN", 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, etc. 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.

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

Path setting: 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"

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

Machining statistics: Run statistics on quantity of required parts, machined parts, this running time and total running time.

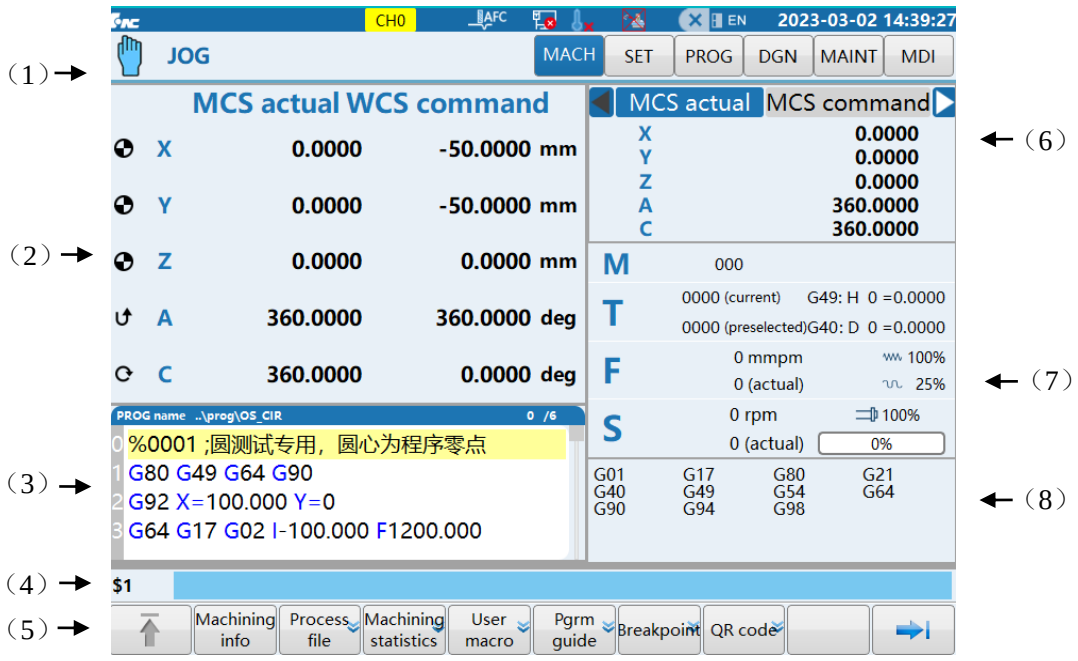
Macro variable: Local variable, channel variable, system variable, axis variable, tool variable and user macro variable of the system can be displayed and queried.

Programming guide: To guide user's programming in the form of work steps. To transform the canned cycle G code into an interactive dialog interface through visual illustrations, parameter descriptions and parameter lists.

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.

3.2.1.1 Machining set interface zoning

After startup, press 〖MACH〗 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.

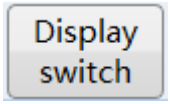
(5) Area---Menu command bar: Operate system functions through function keys in the menu command bar.

(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.

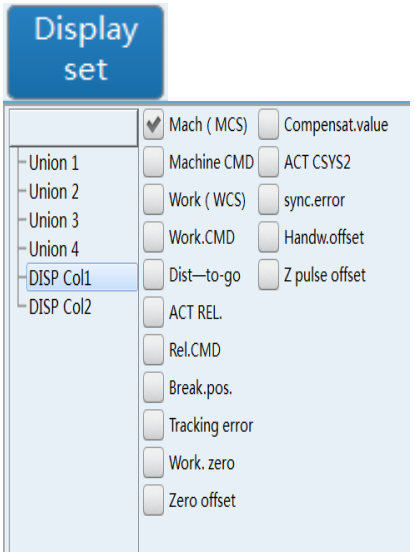
(8) Area--Machining information area: Display G modal, program progress and workpiece statistics during machining.

3.2.1.2 Switching of graphics and G code area display



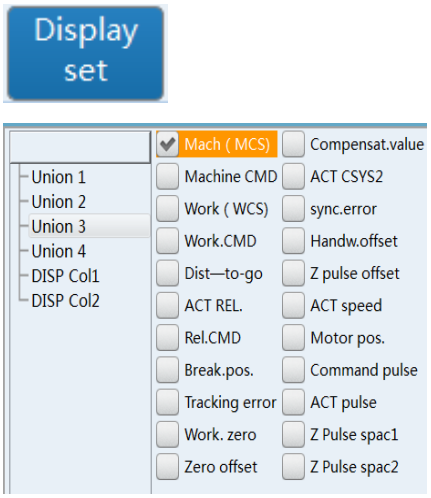
For switching of graphics and G code areas (2) and (3) display , press 『Display switch』 soft key under the main menu interface of this function set, and the display interface switches among 4 interfaces: big character coordinate+ program, joint coordinate, graphics + program and program. (Detailed in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

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



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

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



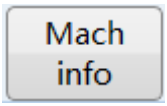
For joint coordinate display setup of coordinates and graphics display area (2), press 『User setting』 soft key under the "MAINT" function set interface to enter the lower-level menu , press『Display setting』 soft key to enter the lower-level menu, select "Joint 1-4", and set 4 coordinate contents in the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

3.2.1.5 Switching of machining and commissioning information area display

For display switching of machining and commissioning information area (6) , press「Alt」+「Left and right arrow 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

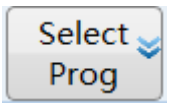
speed, motor position, command pulse, actual pulse, motor speed, waveform frequency, load current and temperature.

3.2.1.6 Switching of machining information area display



For display switching of machining information area (8), press 『Mach info』 soft key under the "MACH" function extension menu interface to switch G modal, machining quantity and other information.

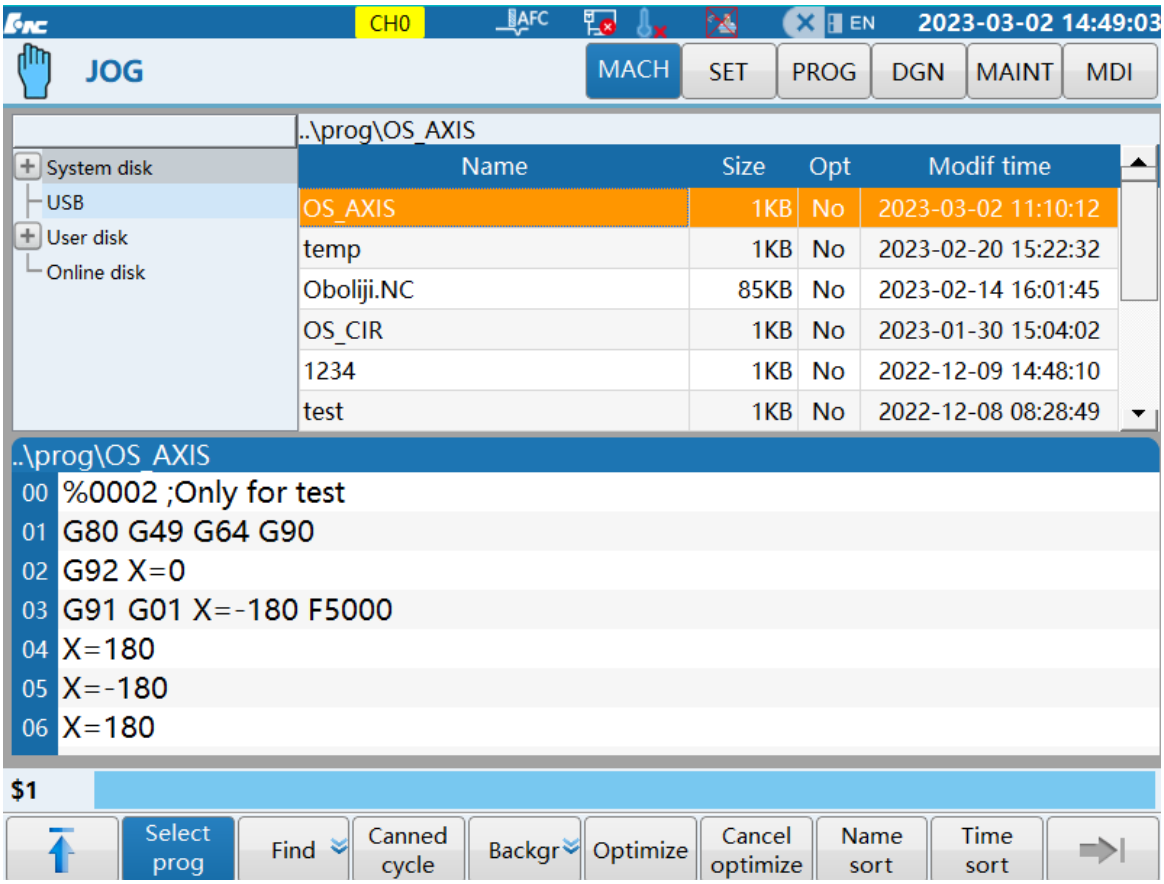
3.2.2 "Select Program" Sub-Interface



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 "Background" in the lower-level menu, and the machine tool should not be at running status while editing current machining program.

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



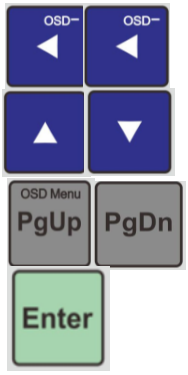
3.2.2.1 Select a program in the USB flash disk and load it as current machining program

- 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.

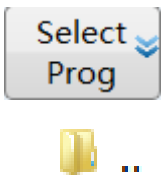
Note: If an error is reported while loading a program, press 「Reset」 to clear it and press 『↑』 to return to level 1 interface);

3.2.2.2 Select a program in the directory as current machining program



- Press 『Select prog』 to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely 『System disk』, 『USB』, 『User disk』 and 『Network disk』, to 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 background

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

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

Note: 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 "Background" sub-interface.

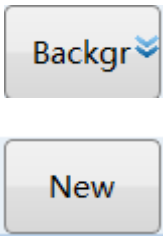
3.2.2.5 Edit other programs than current machining program in the background

- Press 『Select prog』 soft key to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely 『System disk』, 『USB』,『User disk』 and 『Online disk』, to enter corresponding program source disks;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to the selected program file to preview the program;
- Press 『Background』 soft key to enter the editing interface to edit the program;
- After editing or modification, press 『Save』 soft key and a prompt message Saved will be given, then return to the previous interface or other operations.

- Before a file is saved, a prompt message "Save or not" will be given. Press 「Y」 to save the file and 「N」 or 「Reset」 not to save the file.

Note: 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



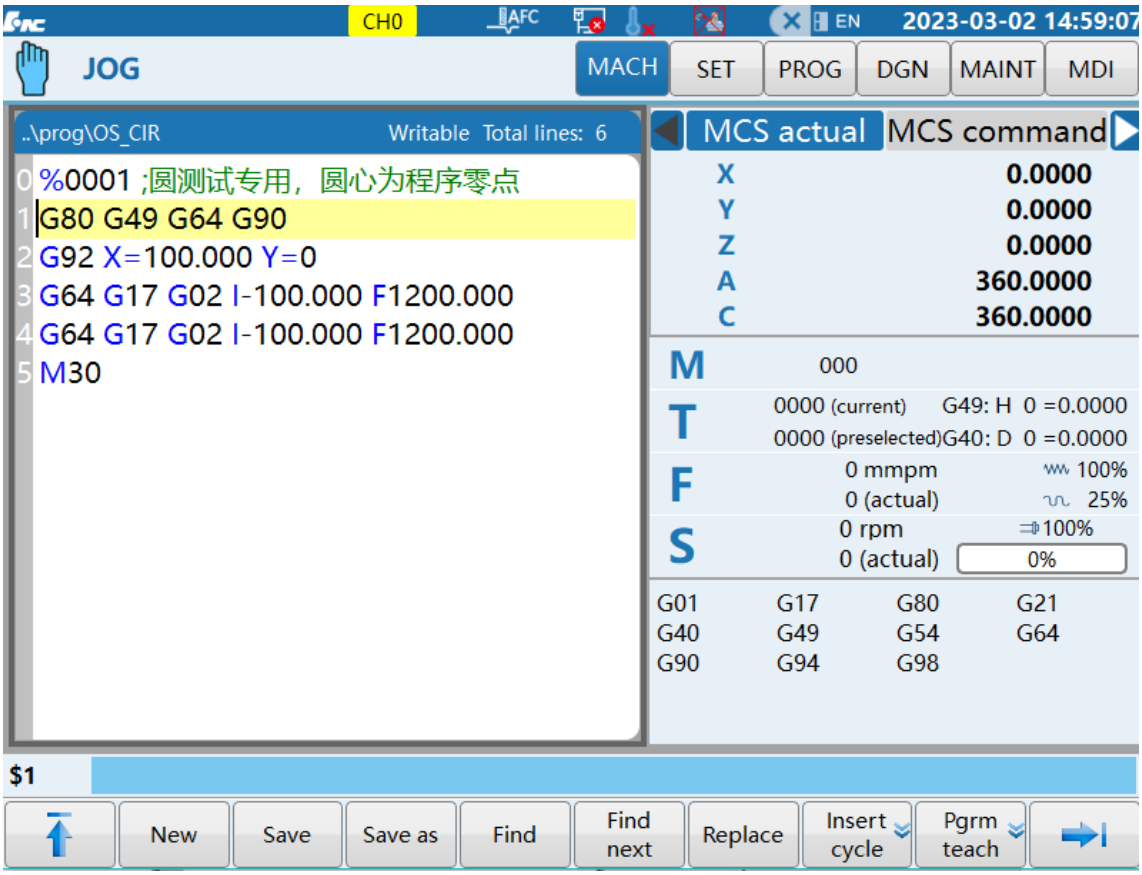
- Press 『Select prog』 soft key to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely『System disk』, 『USB』,『User disk』 and 『Online disk』, to enter corresponding program source disks;
- Press "Background" soft key to enter the "Background" sub-interface;
- Press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

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

3.2.3 "Program Edit" Sub-interface

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

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



3.2.3.1 Edit current machining program

- Press 『Edit prog』 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』 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 program

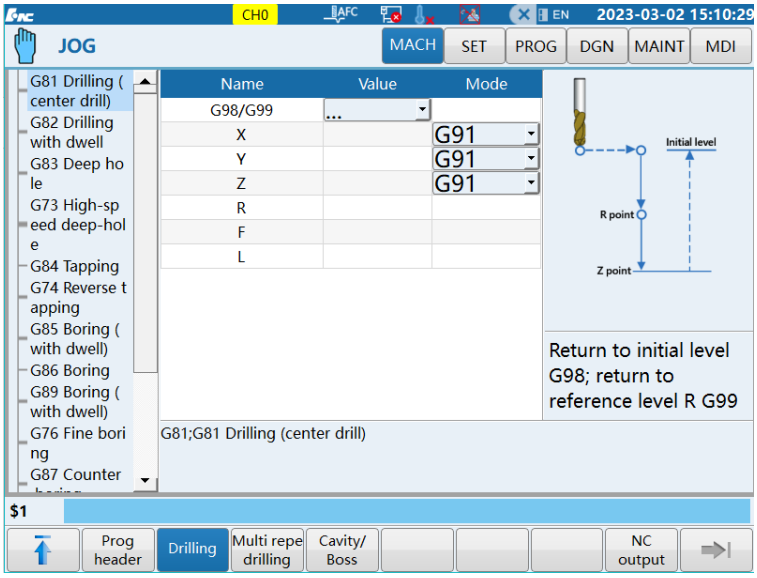
- Select "Edit program" soft key under "MACH" function set to enter the "Edit program" sub-interface;

- 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』 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 Insert cycle and edit cycle

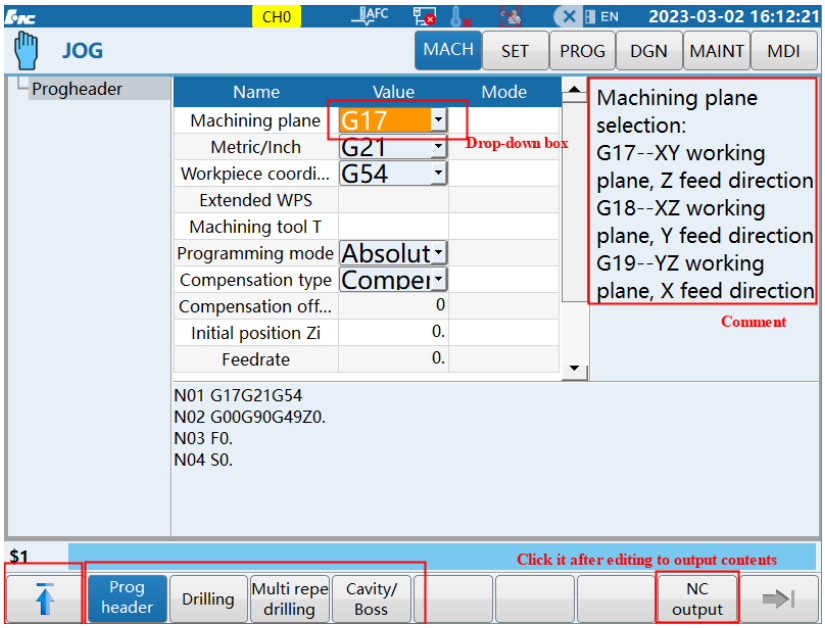
- Select 『Edit prog』soft key under “Machining” function set, and enter the interface of “Insert cycle” or “Edit cycle”.



- **Program header function**

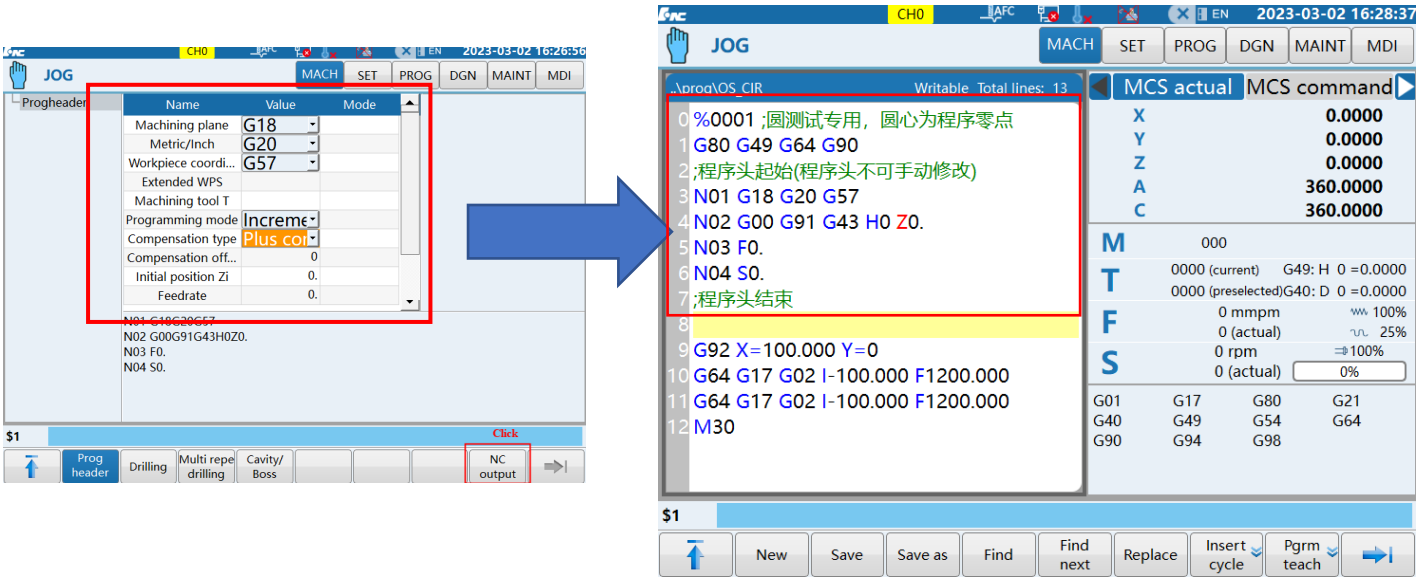
1. By program header, the program preparatory command can be specified easily, such as machining plane selection, coordinate system, federate, etc., which simplifies programming.

   By the up/down button on the panel to wrap lines, and click ENTER to display the drop-down box for selection.



2. Set corresponding commands and preparatory functions on the editing area in accordance with the comment area on the right. Users can preview the documented program on the bottom.

3. After setting corresponding commands, click “NC output” to output corresponding code and go to the edit program interface.

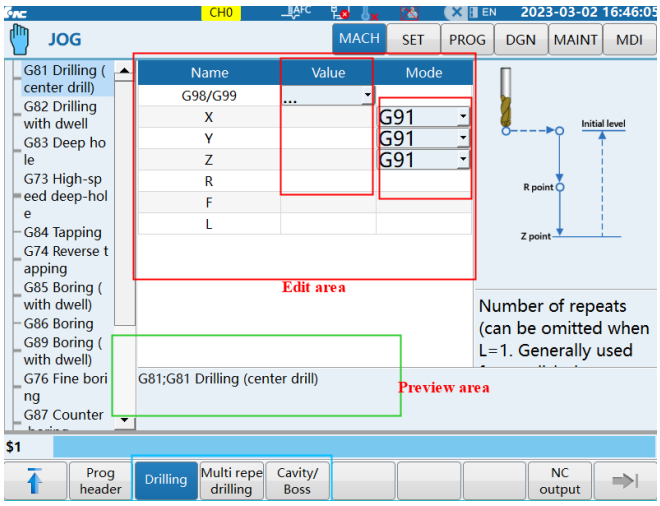


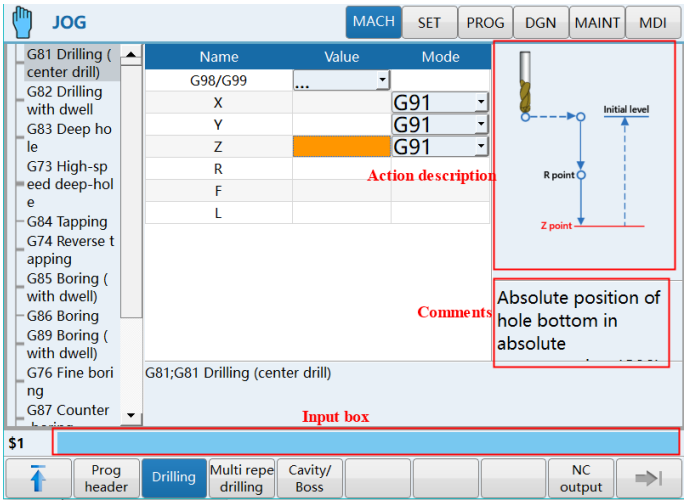
4. Move the cursor to the code line or comment line of program header, click [Auxiliary programming] and [Edit cycle] to read the generated program header code to the input screen, and perform modification on the screen. Then click “NC output” to overwrite the program header codes; If the comment of program header is modified, then the program header input screen cannot be entered.

- **Canned cycle**

There are three types of commands in the canned cycle: drilling, multiple

repetitive drilling, cavity/boss. Users can understand and user rapidly the canned cycle codes by simple action figure and comments.





1. Left/Right buttons to switch left and right interface, Up/Down button to wrap lines. Select canned cycle command to edit, and click ENTER to show drop-down box for setting.

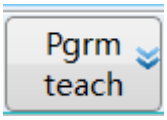
2. Click “NC output” after entering to output corresponding codes and go to the edit program interface.

3. Take G81 as an example, the input contents are shown as below. Press ENTER to select G99, enter the position of hole X30Y20Z10, press ENTER to select G91 or G90, enter reference point R5, input federate F value 1000, and input number of cycles L10. View the edited program lines by preview, click “NC output” to output corresponding code.

4. Move cursor the program line, click “Edit cycle” to modify the input codes. If the commend is modified, the edit cycle screen cannot be read.

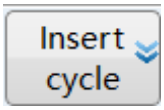
Note: after canned cycle command is finished, click “NC output” to cancel canned cycle command G80, or manually edit G80.

3.2.3.4 Programming teach



- Under “MACH”, click 『Edit prog』 to enter edit program interface.
- Select 『Prgm teach』to enter the main interface of programming teach.

3.2.3.5 Programming guide of canned cycle



- Select 『Edit prog』 in “MACH” .
- Select 『Insert cycle』 to enter wizard programming of canned

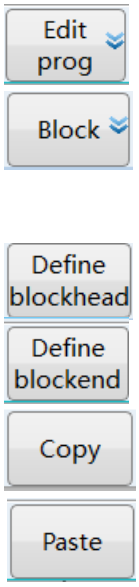
cycle.

3.2.3.6 Block operation

"Block" 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 "Background" function; under "Machining" function set, edit current machining program of "Edit program" function; under "Program" function set, create program by "New" function.

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



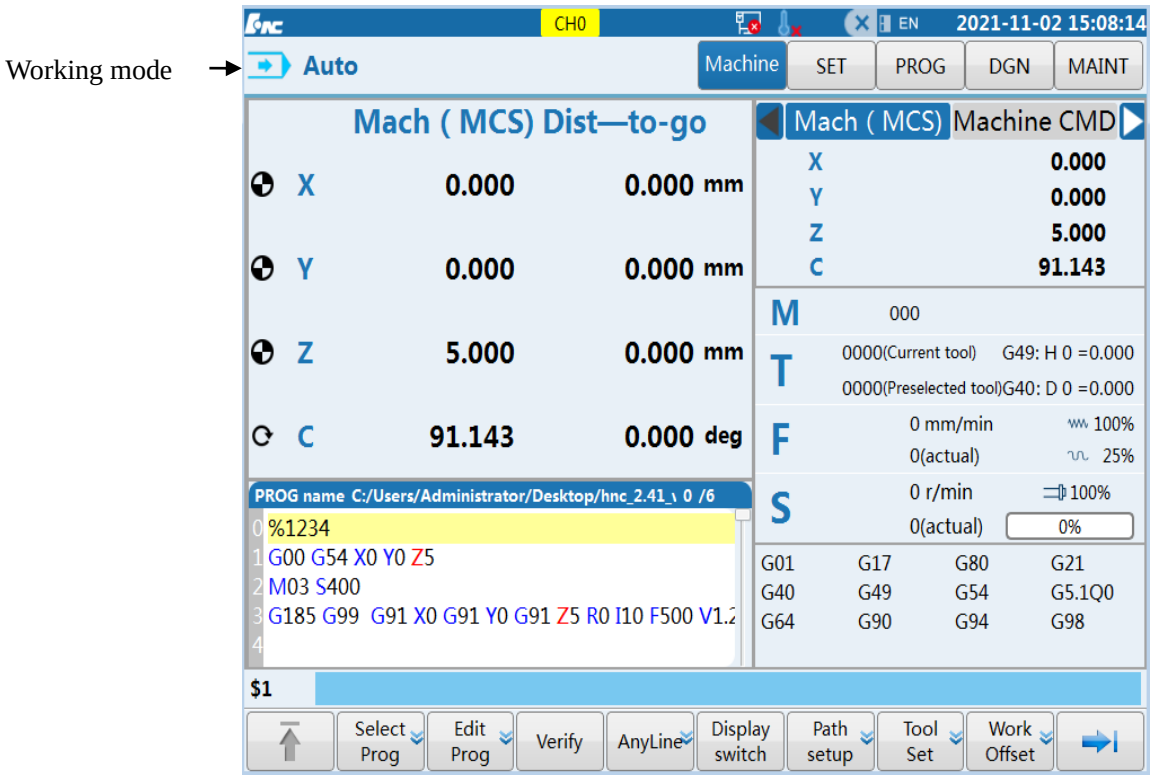
- Press 『Edit prog』 soft key to enter the sub-interface;
- Press 『Block』 soft key to enter the block operation sub-interface;
- Press Cursor or PgUp/PgDn to move the cursor to the first block of the blocks to be edited;
- Press 『Define lockhead』 soft key
- Press「Cursor」or「PgUp/PgDn」to move the cursor to the final block to be edited;
- Press『Define blockend』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 〖MACH〗 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)

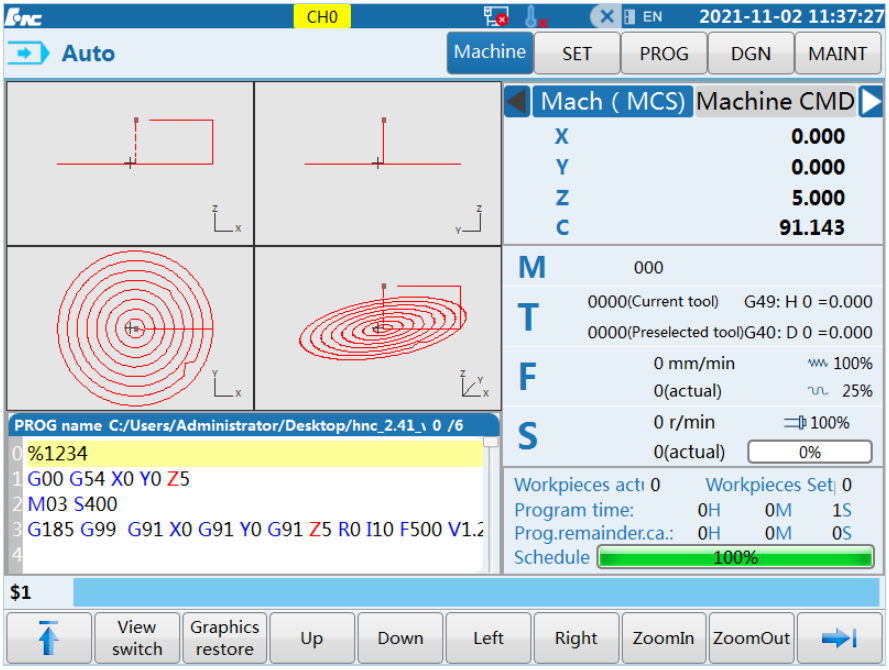
3.2.4.2 Exit "Verify"

- 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.

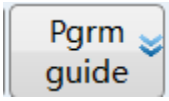
3.2.5 "Path setting" Sub-interface

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.

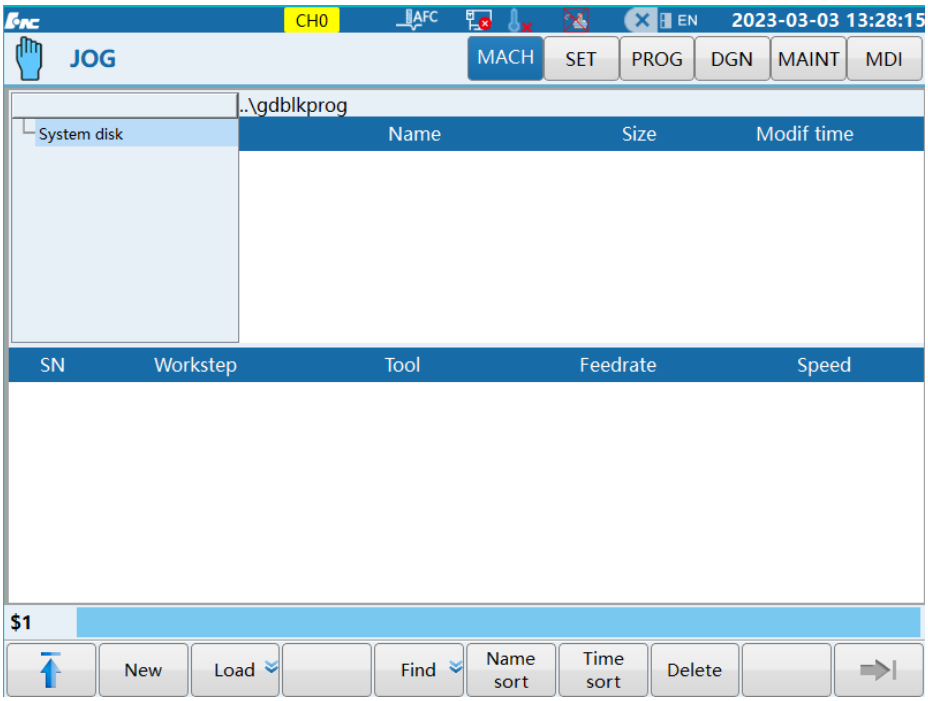
Press 〖MACH〗 function key to enter the level 1 menu of "Machining" function set and press 『→』 to enter the extension menu. Press 『Graphics setting』 soft key to enter the interface, as shown below.



3.2.6 Programming Guide Interface



Press 〖MACH〗→〖Pgram guide〗 to enter the interface.



Workstep file management

On the interface, users can manage the workstep files.

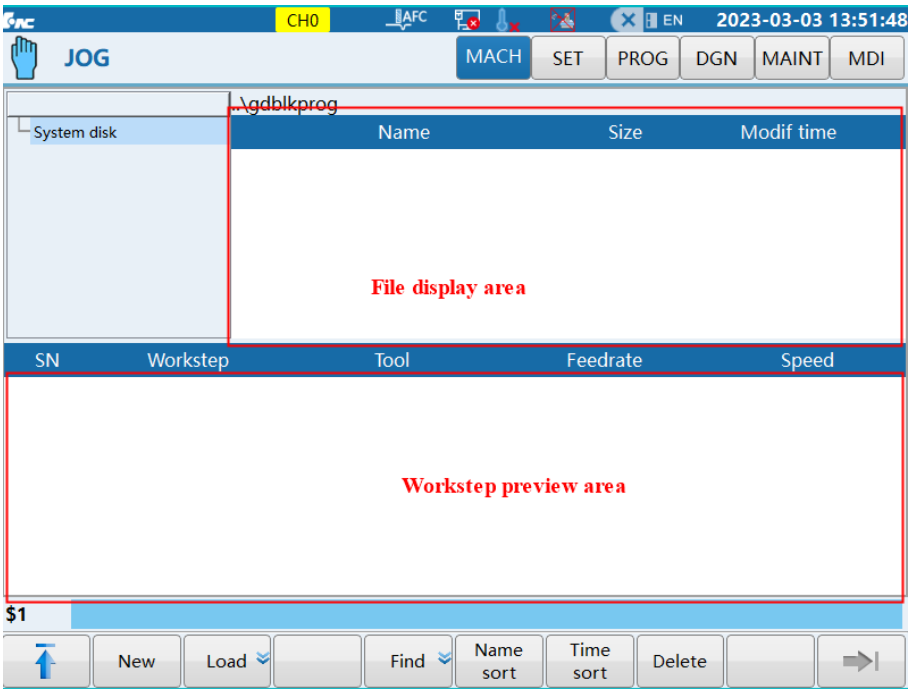
1. New. To create new workstep file. The name of newly-created is OBM0_**.NC under system root directory.

2. Load. To load created file for editing and modification.

3. Find. To find file.

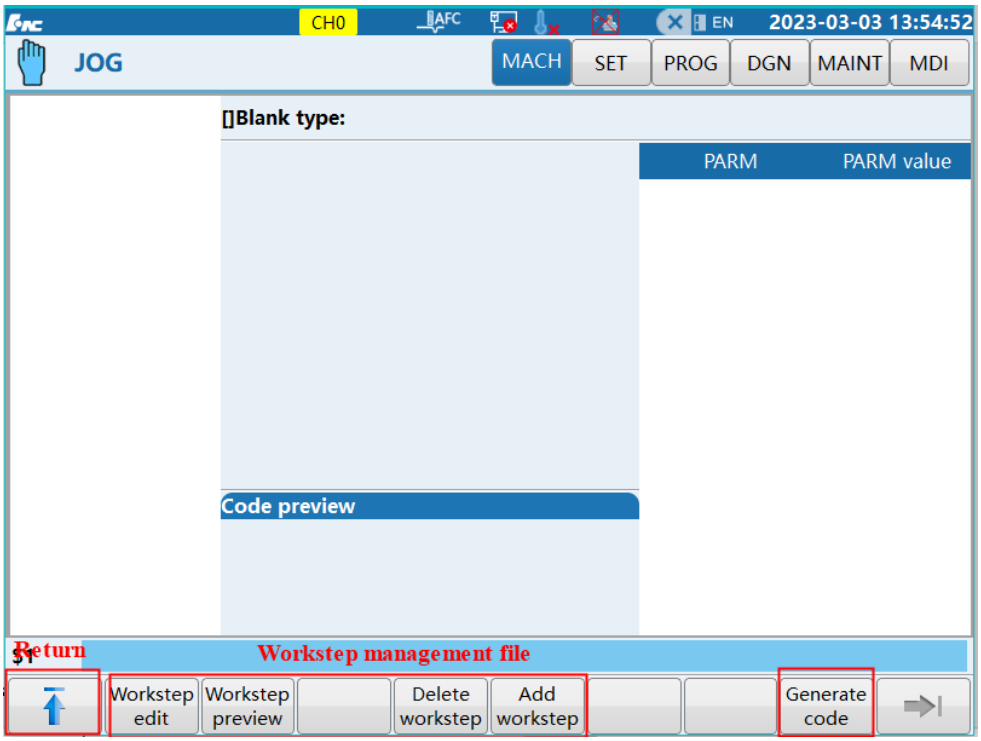
4. Name sort and time sort. To sort files.

5. Delete. To delete workstep file.

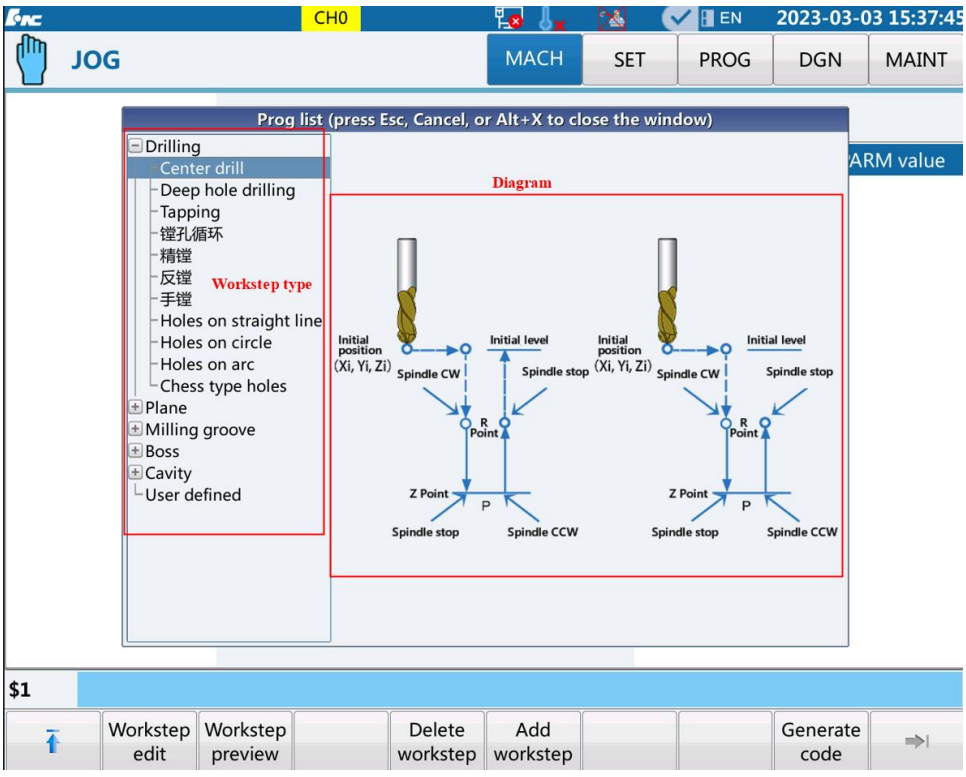


Creation and editing of workstep

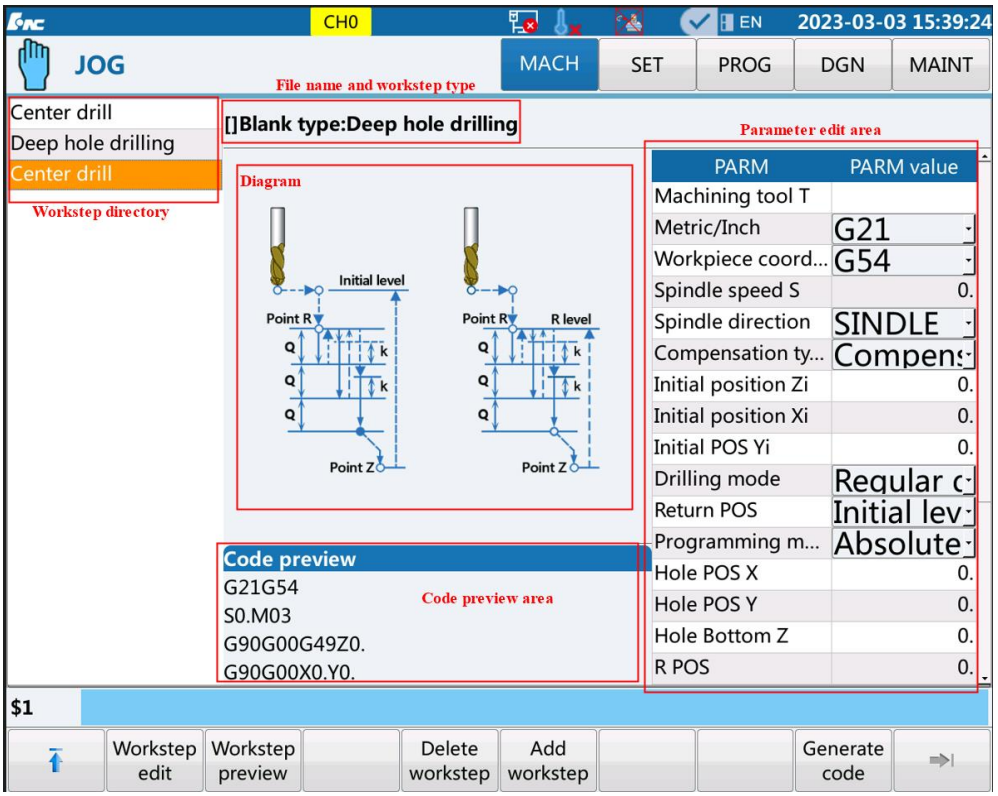
1. Click New or Load to enter the creation and editing interface of workstep.



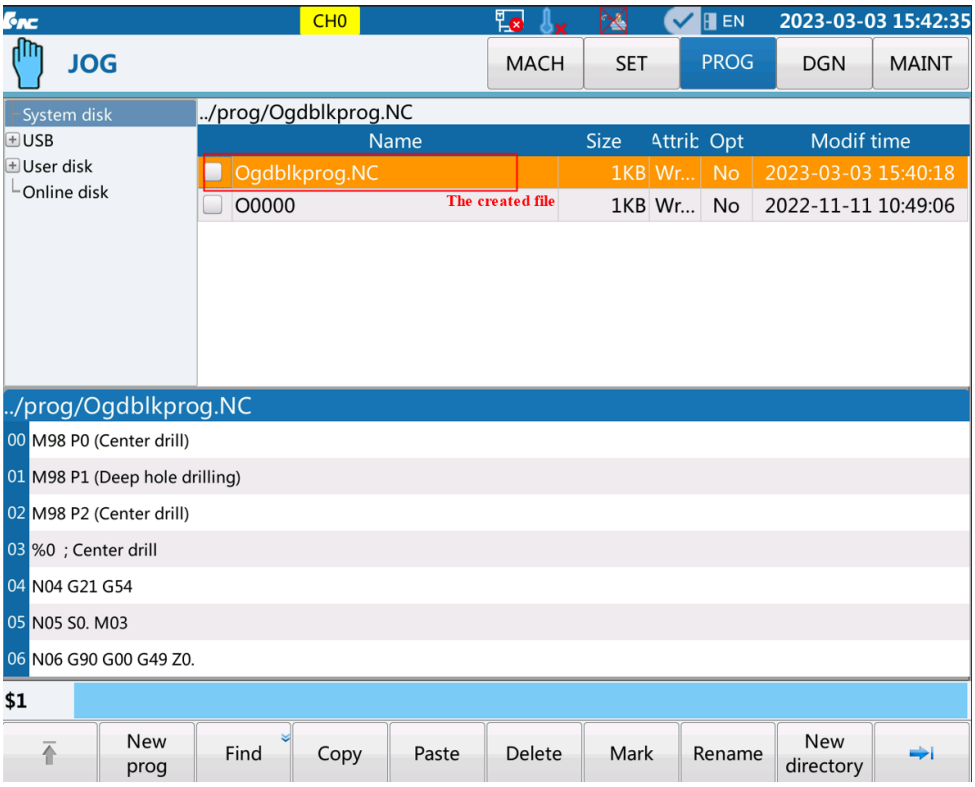
2. Add workstep. Click “Add workstep” to display the workstep type can be inserted.



Enter workstep edit interface after confirming workstep type. Parameter setting area is on the right. Corresponding workstep schematic diagram and code preview is at the middle.

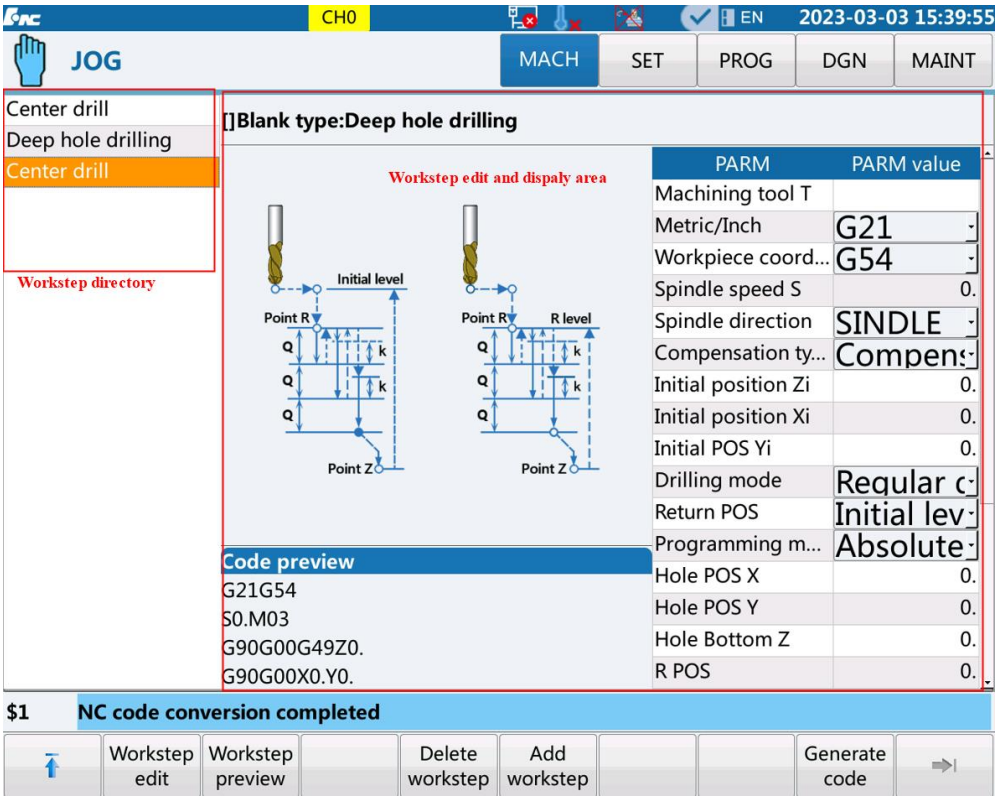


3. Generate code. Click “Generate code” after setting parameters. After the generation is done, the prompt “ NC code conversion done” is issued. Users can enter the machining interface and select the program to load and run. The name of the created file is OBM0_**.NC under system root directory.

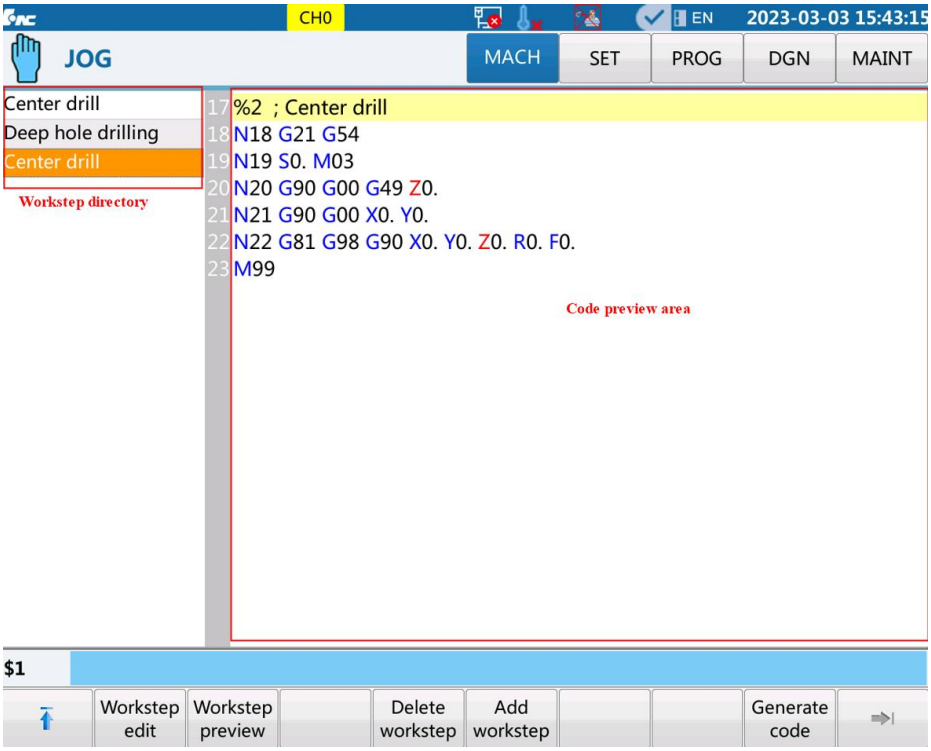


4. Edit workstep. Click “Edit workstep” to enter the edit interface. Select corresponding workstep under workstep directory to display parameter edit

interface of corresponding workstep. Users can modify parameters to generate codes which may overwrite the original file.



5. Preview workstep. Click “Preview workstep” to enter the preview interface. Switch preview area and workstep directory by left and right arrow keys, and select workstep in workstep directory by up and down keys, then corresponding code position will be displayed in the preview area.



3.2.7 Collision Monitoring Interface

The dynamic collision monitoring (DCM) can detect movement of machine components during whole machining to avoid collision of tool and spindlw with machine components; even the collision can be detected before program running to avoid the damage of workpiece caused by interruption of program running. The function is divided into manual collision monitoring and automatic collision monitoring.

Press 〖MACH〗→〖Collision monitor〗 to enter the interface, as shown below.

3.2.7.1 Function enabling and disabling

Enable function: Click F10 twice to find “collision monitor” key, and click it to enter the interface. Click “Auto DCM on” to turn on auto DCM function, and the key turns grey; click “manual DCM on” to turn on manual DCM function, and the key turns grey. After turning on the function, the icon of turning on the function is displayed on the top of the interface.

Disable function: Click “Auto DCM off” to turn off auto DCM function, and the key turns grey; click “manual DCM off” to turn off manual DCM function, and the key turns grey. After turning on the function, the icon of turning off the function is displayed on the top of the interface.

3.2.7.2 Function configuration

Dynamic collision monitoring is a customized function and is not available as a standard function. Before using this function, users need to apply for authorization and complete configuration according to the manual.

DCM interface configuration

After authorization, the DCM key and the corresponding interface cannot be displayed on the interface of CNC controller. It is necessary to set NC parameter 000429 to 1 on the parameter setting interface, and then restart the system to display the key and the interface. The parameters displayed on interface and the interface are shown in the figure below.

Model file configuration

Model file configuration is divided into machine component model file configuration and tool model file configuration.

Machine model files include machine bed model files (Base.stl), traverse axis model files (x.stl, y.stl, z.stl, a.stl, c.stl), and spindle model files (spindle.stl),

as shown below.

Note: Only STL model file is supported.

Tool model file includes the model files of all tools mounted on the machine, as shown below.

Note: The tool model file must be named with the tool number, otherwise loading tool model will fail during collision monitoring.

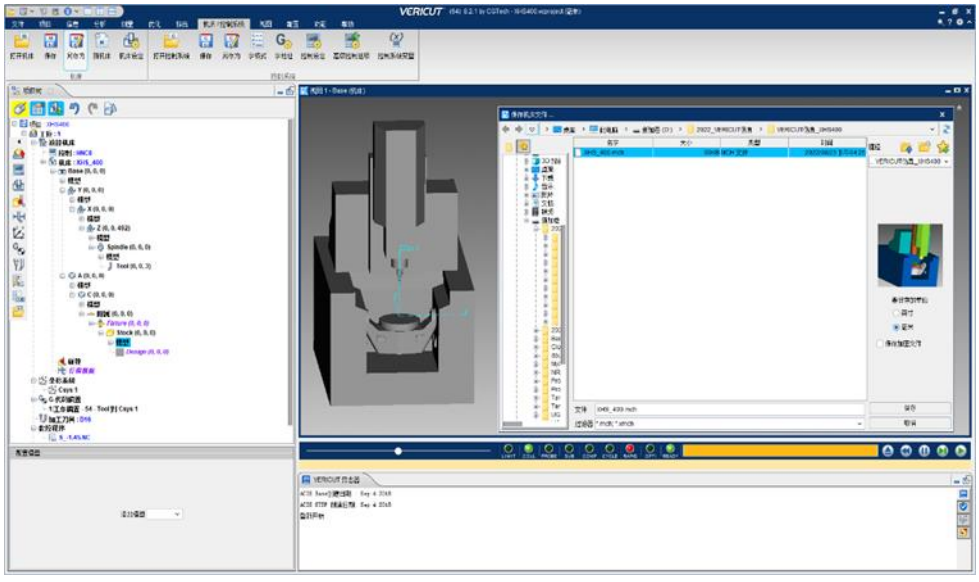
After the machine model file and tool model file are ready, create a data folder, put the machine model folder and tool model folder in the data directory, and then package the data folder into the upgrade package in BTF format, upgrade to the CNC system.

Motion chain file configuration

The motion chain file (json file) contains the folder name of machine components, machine component model file path, tool model file path, model name of motion component, safe distance of manual collision monitoring, safe distance of auto collision monitoring, and motion relationship chain.

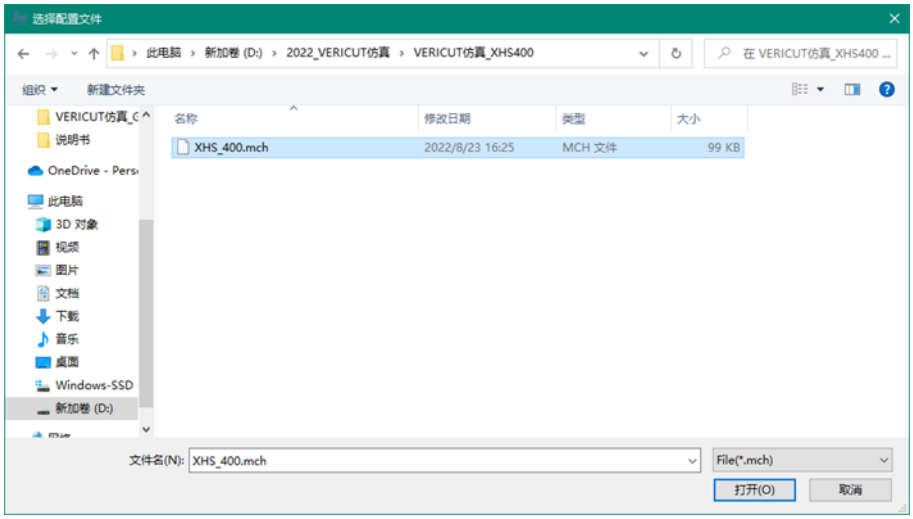
The configuration of the motion chain file is divided into the following steps:

The model file is put into VERICUT simulation software to build the machine motion chain model, and the machine component motion chain file.mch file was exported through the software.

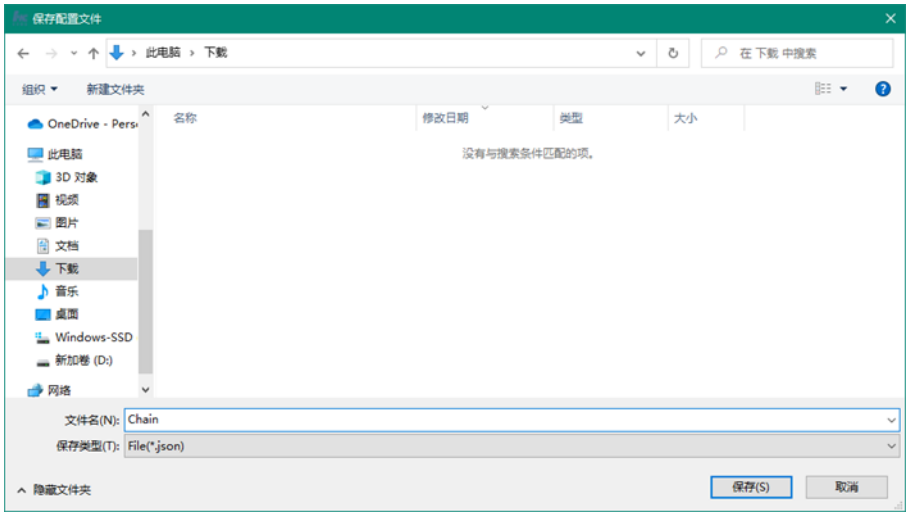


Export motion chain file of VERICUT machine component

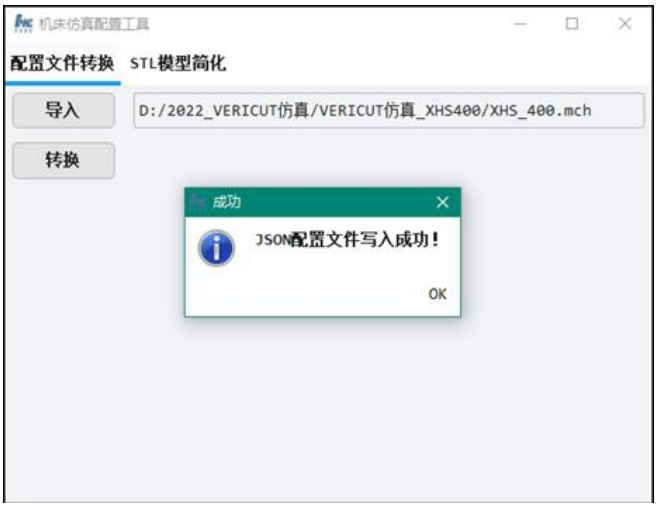
(1) Import the motion chain file .mch of machine component in the format of vericut into the collision monitoring configuration file conversion tool, and the motion chain file in json which matches HNC CNC system is obtained.



Import the motion chain file of VERICUT machine component



Select the export position of Chain.json file



Export Chain.jsn file

(2) Place the newly-generated motion chain file into data directory, and package the data file folder into the upgrade package in BTF, and upgrade to the CNC system.

Model zero offset parameter configuration

The zero of machine model is usually located at the origin of the construction model coordinate system, and the actual zero of machine is related to the encoder zero of logic axis, so there is a position deviation between machine model zero and actual machine zero.

Model zero offset can be set by channel parameters 040676, 040677, 040678, as shown in the figure below.

040676	Collis detec model: X zero offset	0.0000	Restart
040677	Collis detec model: Y zero offset	0.0000	Restart
040678	Collis detec model: Z zero offset	0.0000	Restart

For AC rotary table machine, model X zero offset is usually the same with the RTCP parameter second rotary axis offset vector (X), model Y zero offset is the same with the second rotary axis offset vector (Y), and model Z zero offset is the same with the second rotary axis offset vector Z.

Safe distance and collision monitoring pair configuration

Click “Configuration” key on the menu of collision monitoring interface to enter collision monitoring configuration interface. Users can set manual safe distance, auto safe distance, and collision monitoring pair on this interface. As shown below.

Manual safe distance. The value ranges from 2.0 to 20.0. 10.0 is the recommended value, in the unit mm.

Auto safe distance. The value ranges from 2.0 to 10.0. 10.0 is the recommended value, in the unit mm.

Collision monitoring pair indicates the two different and non-adjacent component that need dynamic collision detection. Users can click F5 key to add, and F6 key to delete, and need to select corresponding components in component 1 and component 2.

After setting safe distance and collision monitoring pair, users need to click Save on the menu to take effect, and click “Cancel” to cancel currently set safe distance and collision monitoring pair.

Note: When manual collision monitoring is enabled, distance between collision monitoring components of current machine cannot be smaller than

safe distance; otherwise manual anti-collision function cannot be enabled, and the prompt box will display no enabling of collision monitoring in collision area.

Take XHS400 as an example,

Example of manual anti-collision function

After manual anti-collision function is enabled, switch the system mode to Jog or MPG, move X and Y axes to the center of rotary table, and move Z axis downward. When distance between the lower surface of spindle between upper surface of C rotary table is smaller than the safe distance, system will issue and alarm, and display the prompt of removing the alarm, as shown in below figure.

Example of auto anti-collision function

After auto anti-collision function is enabled, set G55 coordinate system as: X-210.968, Y-368.271, Z-495.3318.

Then load the below program:

%1234

G43 H1

G53 G01 Z0 F1875

G55 G01 X0 Y0 A0 C0 F5000

G53 Z-338

M30

When The function detects that the distance between two components smaller than safe distance exists in the command path, the system issues an alarm, as shown below. Pressing Reset can remove the alarm.

3.3 "Set" Function Set Interface and Basic Operation

3.3.1 "Set" Function Set Interface and Function

"Set" 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 setting, tool compensation setting*, 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. "Mach" 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 "Set" function set are shown below.

Tool compensation	Magazine	Tool life	Broken tool check	Coordinate	Workpiece measure	Auto tool setting	Feature coordinate system	⇨

Tool measure	Tool shape	5-axis tool measure	Reinishaw measure

- 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 "Set" function set are the same as those of "Tool compensation" under "Mach" function set.

- Magazine: The magazine function interface displays tool number and machining mode. The magazine 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 measure』 soft key.

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

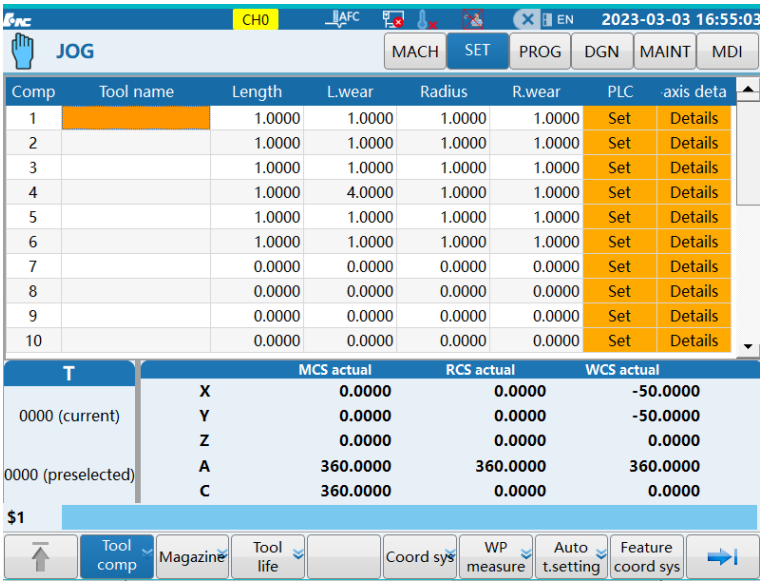
- Workpiece measurement: This function is used for center measurement, plane measurement and circle center measurement of workpiece. The measurement results are stored in G54-G59 and extension coordinate system. For specific operating steps, please refer to "Tool setting and machining setting'.

- 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 setting'.

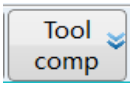
- Renishaw measurement: HNC-Renishaw measurement supports manual measurement and insertion measurement. Manual measurement supports 3-axis and 5-axis mode. Insertion measurement supports 3-axis mode, and auto measurement supports 3-axis and 5-axis modes. The function incudes: dimension measurement, workpiece coordinate system setting, workpiece coordinate system update, and tool compensation.

Some functions of "Set" 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〖Set〗function key to enter the default interface of "Set" function set, as shown below.



3.3.2 Tool Compensation Interface

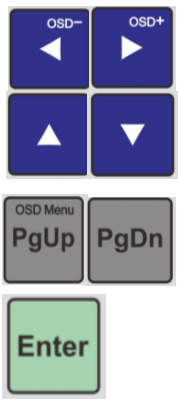


The compensation function mainly achieves the setting of tool length compensation, length wear, radius compensation, and radius wear.

User can manually input tool length compensation value, and the tool compensation can also input automatically by auto measurement mode.

To simplify operations, the system configures the tool compensation function in both “MACH” and “SET”. Their functions and operations are the same.

3.3.2.1 Direct input mode of tool length compensation



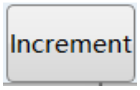
- Press 『Tool comp』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 「Direction」 or 「PgUp/PgDn」 to move the cursor to tool length compensation of corresponding tool number;
- Press 「Enter」 to confirm, activate input status, and the input box gives a prompt message 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 Current position input mode of tool length compensation

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 comp』 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.

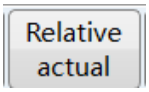
3.3.2.3 Incremental input of tool length compensation



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

- Press 『Tool comp』 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 『Increment』 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.

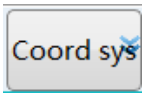
3.3.2.4 Relative actual input of tool length compensation



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

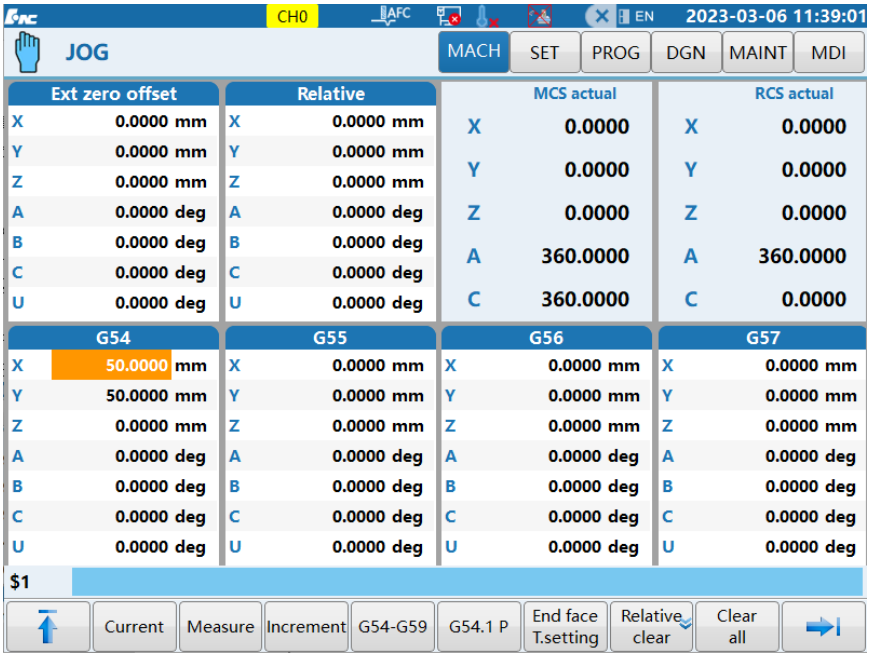


For ease of operation, the system configures "Coordinate system" function both under "Mach" set and "Set" set with the same function and operation. This section is introduced with "Coordinate system" submenu under "Set" set.

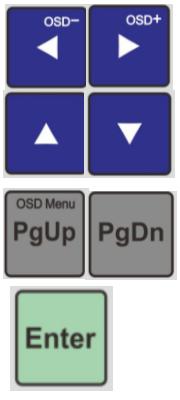
The coordinate values of "External zero offset", "Relative coordinate system", “G54-G59 coordinate system” and "G54.1P1- G54.1P60 coordinate system" can be set under the "Coordinate system" sub-interface (as shown below).

There are 3 areas of coordinates on this interface. 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

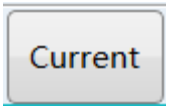


This function can be used to input known workpiece zero coordinates into the selected workpiece coordinate system.

- 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.

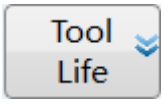
- Press 『Coordinate system』 soft key under the level 1 menu of "Setup" function set to enter the sub-interface
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to select the coordinate system;
- Press 『Current』 soft key to activate the input box, and a prompt message: "Set current position as workpiece zero or not" will be given
- Press 「Y」 to set coordinates of current machine as the the selected workpiece zero;
- Press 「N」 or 「Reset」 to abandon the setting 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.

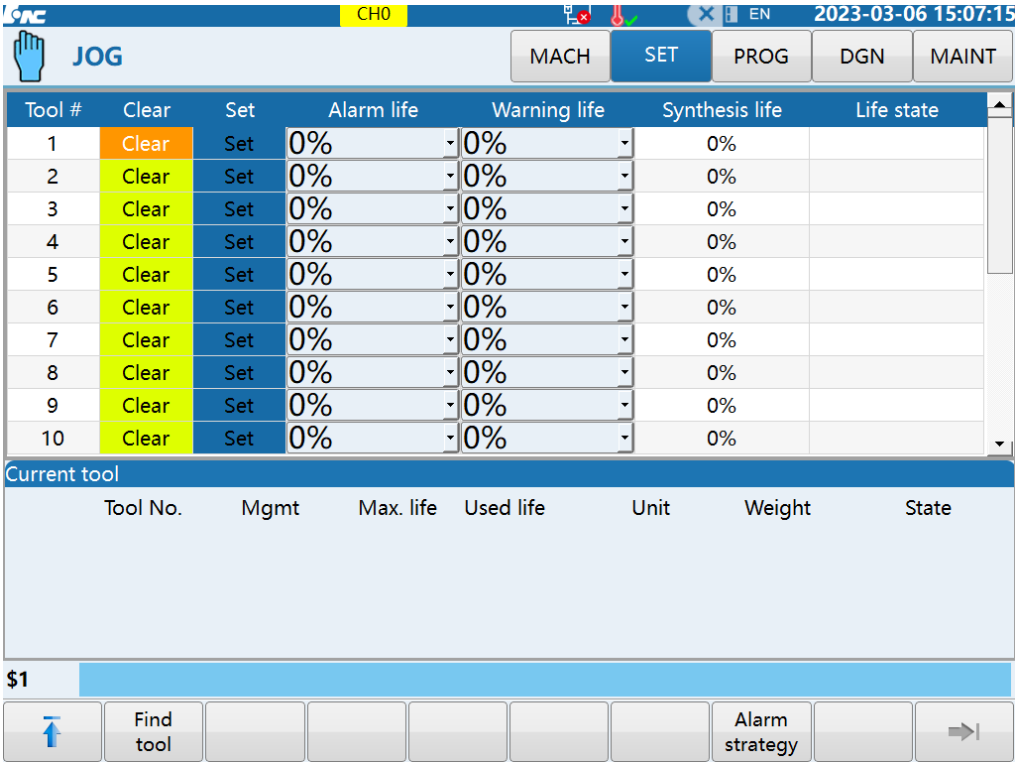
- Press 『Coord sys』 soft key under the level 1 menu of "Set" function set to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to select the coordinate system;
- Press 『Increment』 to activate the input box;
- Input the incremental value of the coordinates in the input box;
- 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

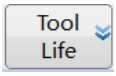


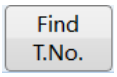
Under the "Tool life" sub-interface (as shown below), 5 tool life determination benchmarks can be set by "Set": Number of installations, cutting time, cutting distance, cutting energy consumption and number of spindle revolution.

When the specified value is reached for one of the benchmarks, the system can determine early warning or alarm status of tool life 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.



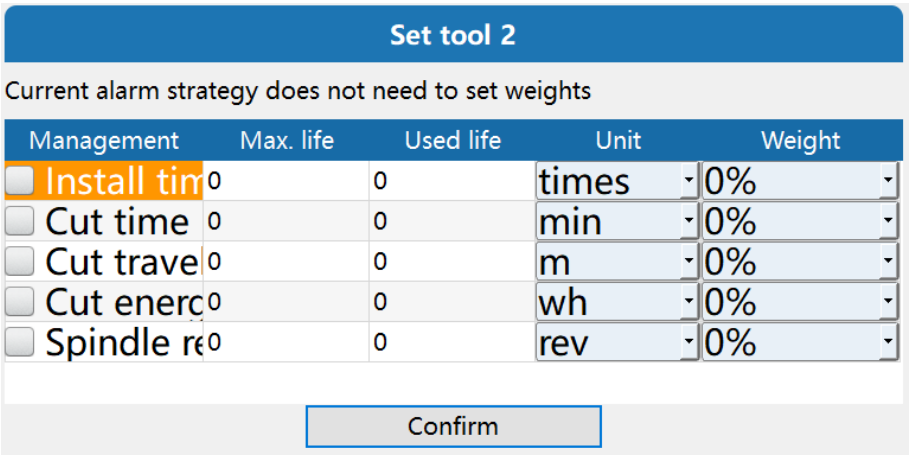
3.3.4.1 Tool life benchmark setting



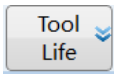


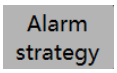
- Press 『Tool life』 soft key under the "set" interface to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 or 『Find tool number』 soft key to move the cursor to the "Set" column of the selected tool;
- Press 「Enter」, the life benchmark setup window will pop up (as shown below);
- Select management 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.

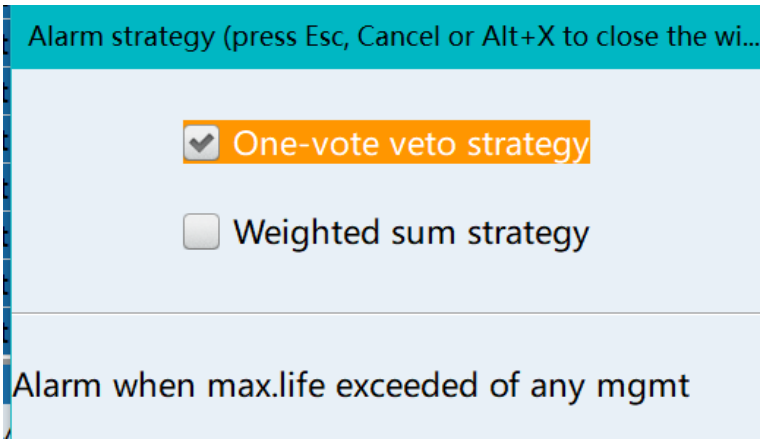


3.3.4.2 Tool life alarm strategy setup

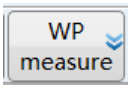




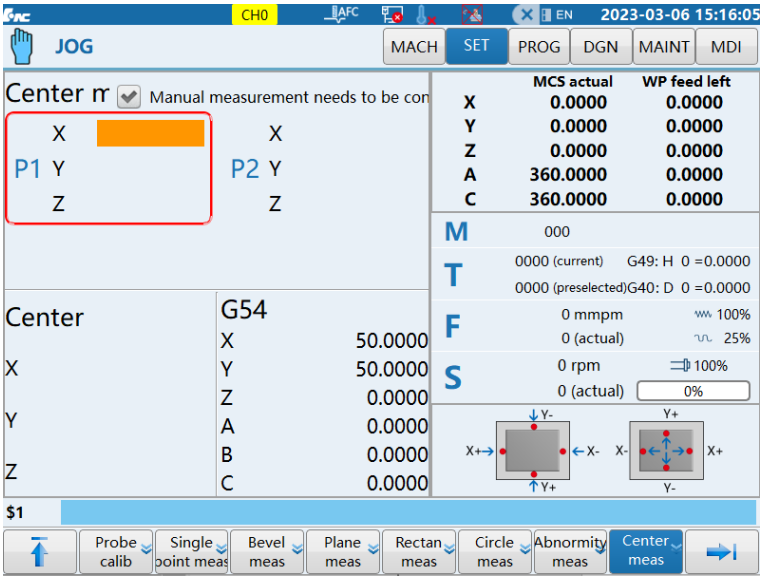
- Press 『Tool life』 soft key under the "Set" interface to enter the sub-interface;
- Press 『Alarm strategy』 soft key, and the strategy selection window will pop up (as shown below)
- Press 「Cursor」 to select the alarm strategy;
- Press 「Enter」 to confirm and exit the selection window.



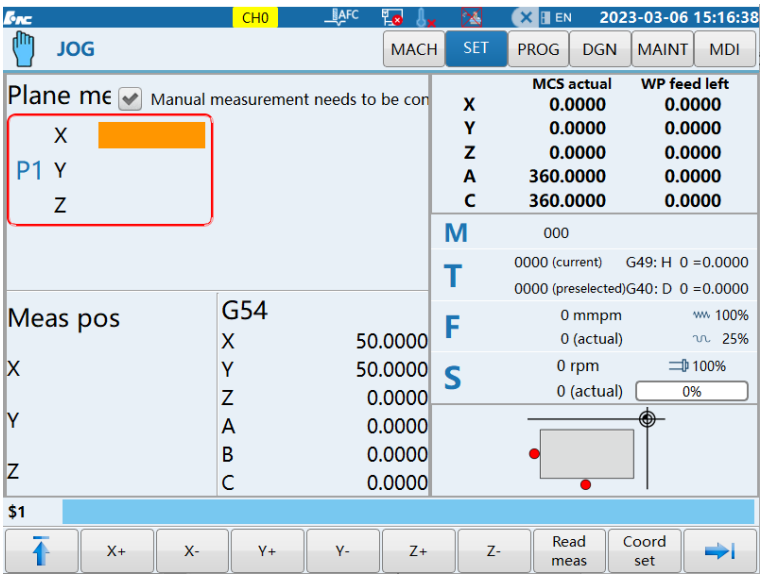
3.3.5 "Workpiece Measurement" Sub-interface



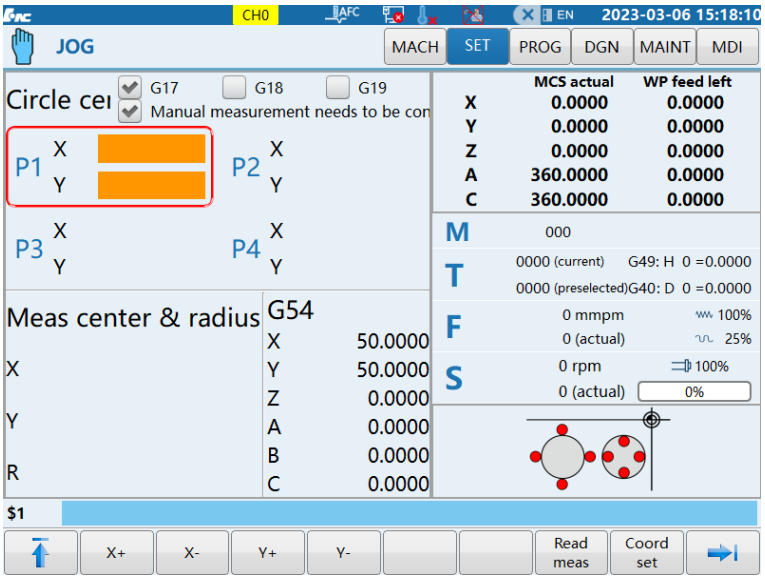
Press 『WP measure』 under the extension menu of "Set" function set to enter the "Workpiece measurement" default sub-interface (as shown below).

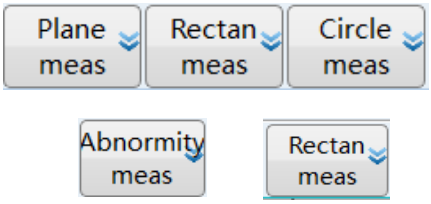


Press 『WP measure』 under the extension menu of "Set" function set, and press 『Plane meas』 to enter plane measurement.

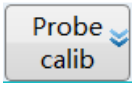


Press 『WP measure』 under the extension menu of "Set" function set, and press 『Circle meas』 to enter circle center measurement.

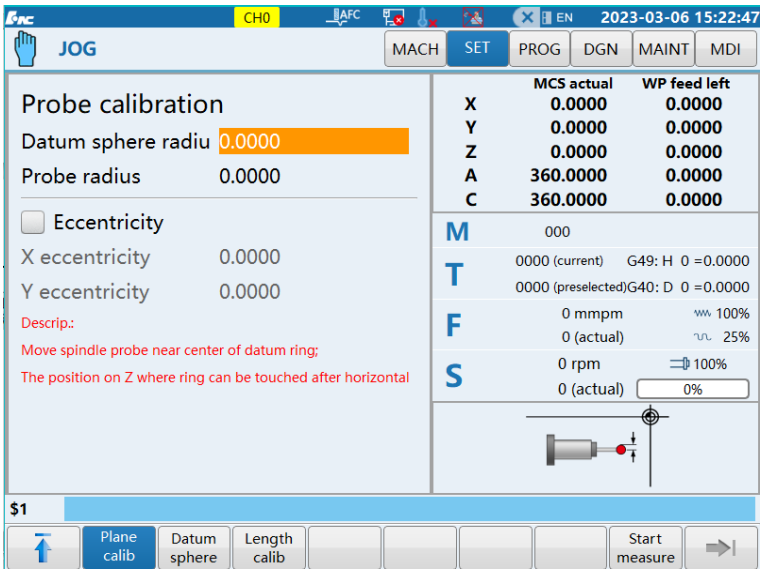


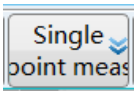


The Standard configuration supports three manual measurement functions: center measurement, plane measurement and circle center measurement, under the "Workpiece measurement" interface. When some appropriate measuring instruments and softwares are configured, the probe calibration, single point measurement,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

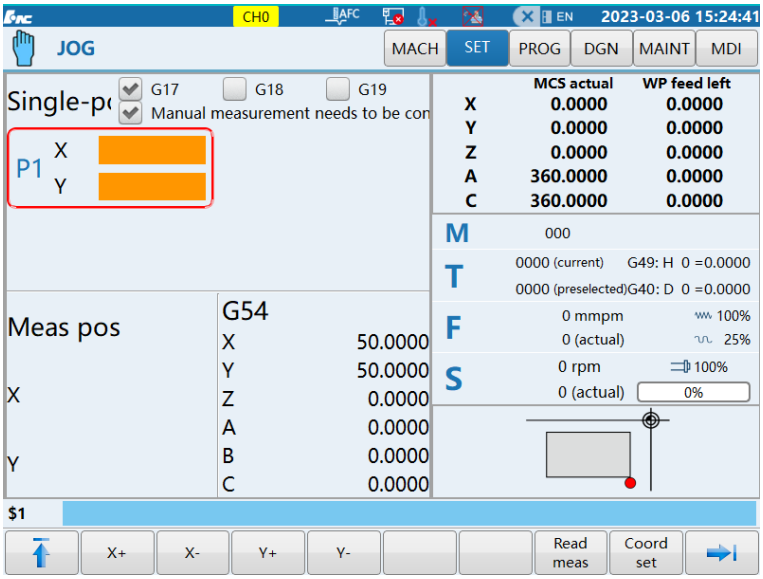


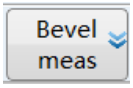
Press 『WP measure』 under the extension menu of "Set" function set, and press 『Probe calib』 to enter circle center measurement.



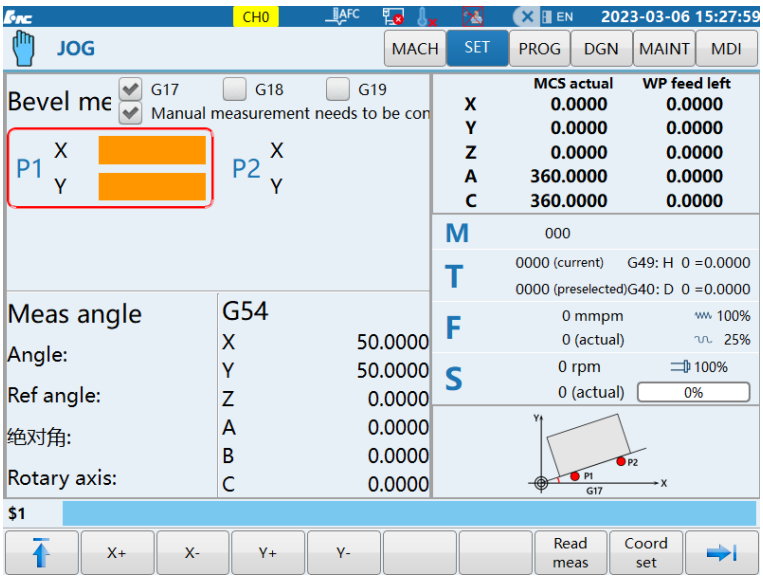


Press 『WP measure』 under the extension menu of "Set" function set, and press 『Single point meas』 to enter circle center measurement.

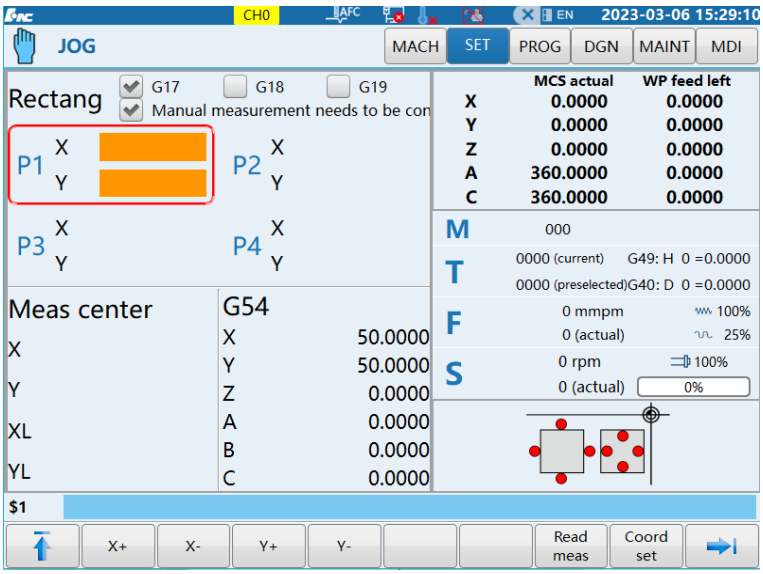




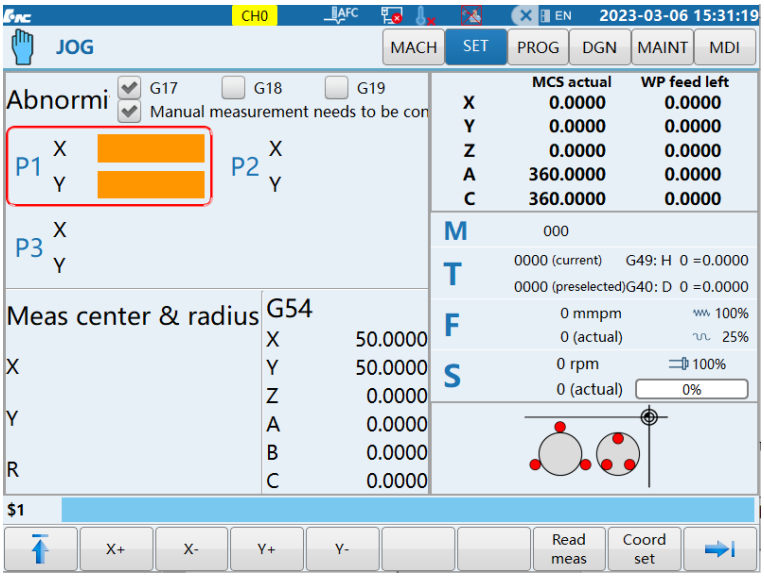
Press 『WP measure』 under the extension menu of "Set" function set, and press 『Bevel meas』 to enter circle center measurement.



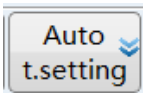
Press 『WP measure』 under the extension menu of "Set" function set, and press 『Rectan meas』 to enter circle center measurement.



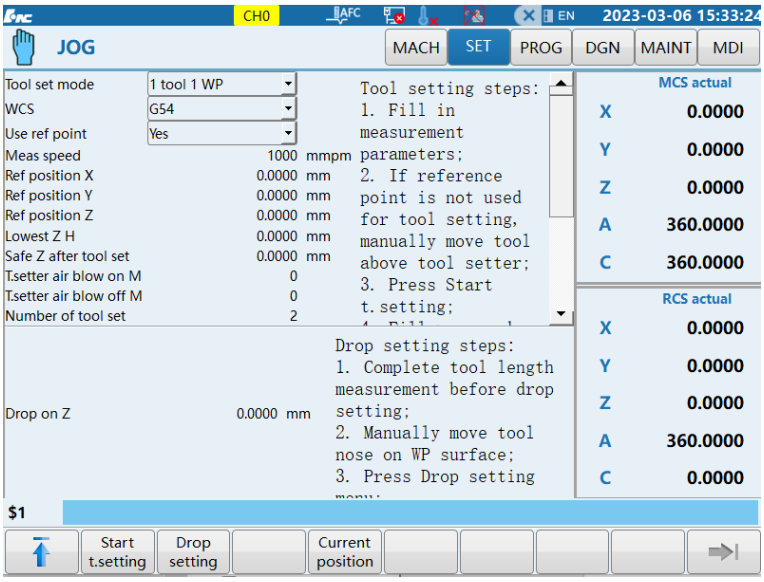
Press 『WP measure』 under the extension menu of "Set" function set, and press 『Abnormity meas』 to enter circle center measurement.



3.3.6 "Automatic Tool Setting" Sub-interface

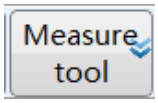


Press 『Auto t.setting』 under the extension menu of "Set" function set to enter the "Auto tool setting" default sub-interface (as shown below).

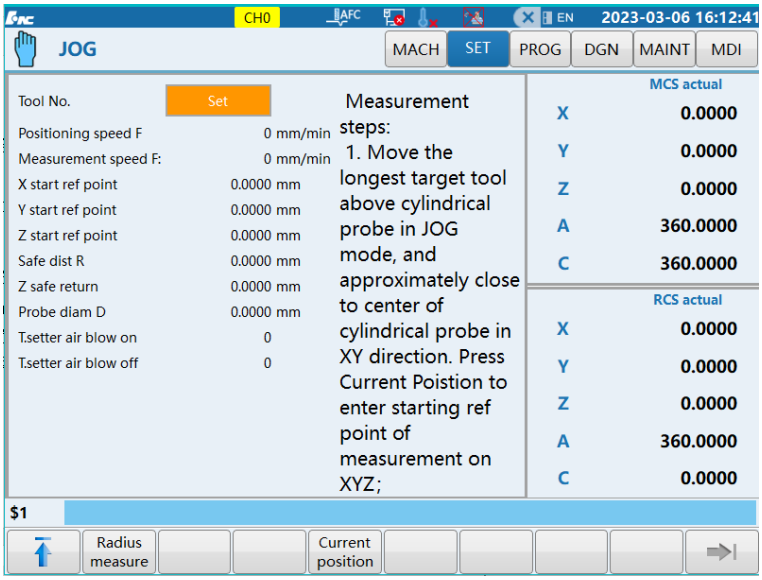


The "Automatic tool setting" sub-interface supports three automatic measurement functions of tool length including single tool single workpiece measurement, single tool multiple workpieces measurement and multiple tools multiple workpieces measurement. To satisfy the three different types of applications, 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 "Measure Tool" Sub-interface



Press 『Measure tool』 and 『Radius measure』under the extension menu of "Set" function set to enter the radius measurement window (as shown below).

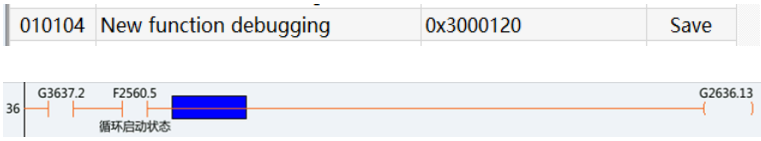


3.3.8 “Adaptive Feed” Sub-interface

Press 『Adaptive feed』 under the extension menu of "Set" function set to enter the adaptive feedrate interface.

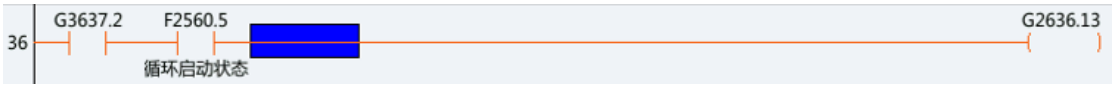
How the adaptive federate function works: In the machining of the five-axis machine, the load current of spindle servo motor or feed axis servo motor (the maximum cutting load axis of X, Y and Z axes) is detected and compared with the pre-set value in the tool data (% of the rated load). If not the same, the feedrate is automatically changed to prevent motor overload.

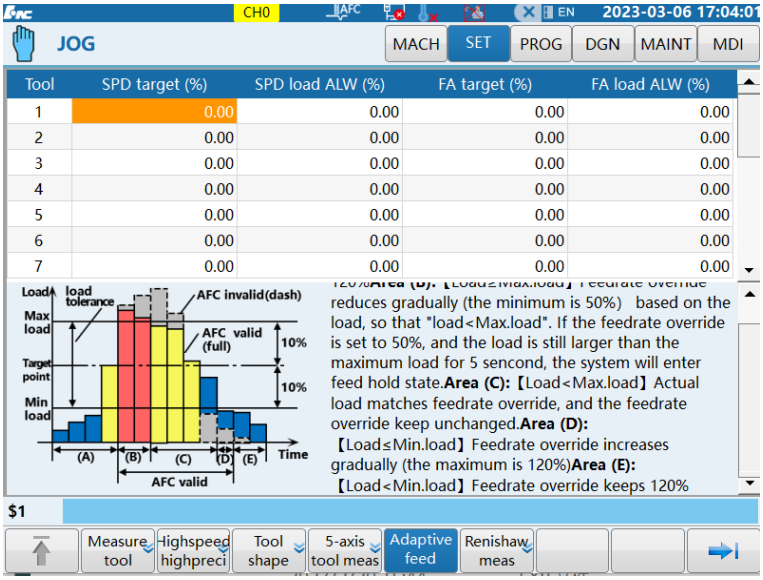
Enable function: When the machine user parameter 0010104 new function debugging parameter is set to 1000000, the adaptive feedrate is enabled. (After the parameter is set, users need to turn on G2636.13 in PLC to take effect the adaptive feedrate control. Add G3637.2 and G2636.13 in PLC.



G code command switch: G115L17P1 is to enable adaptive feedrate.

Physical button: In PLC, after associating the input point of the button with G3637.2, press the OFF button to disable the adaptive feedrate function.





Note:

On this interface, the target load and load allowance (0.10 to 1000.00) % cannot be set to 0, and can be modified to 0 only in MDI.

When only spindle or feed axis load needs to be adaptive, users just need to set two parameters of corresponding axis, and the parameter of the other axis is set to 0. For example, when only spindle load is adaptive, users just need to set spindle target and spindle load allowance. The feed axis target and feed axis load allowance are set to 0.

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.

New prog	Find	Copy	Paste	Delete	Mark	Rename	New directory

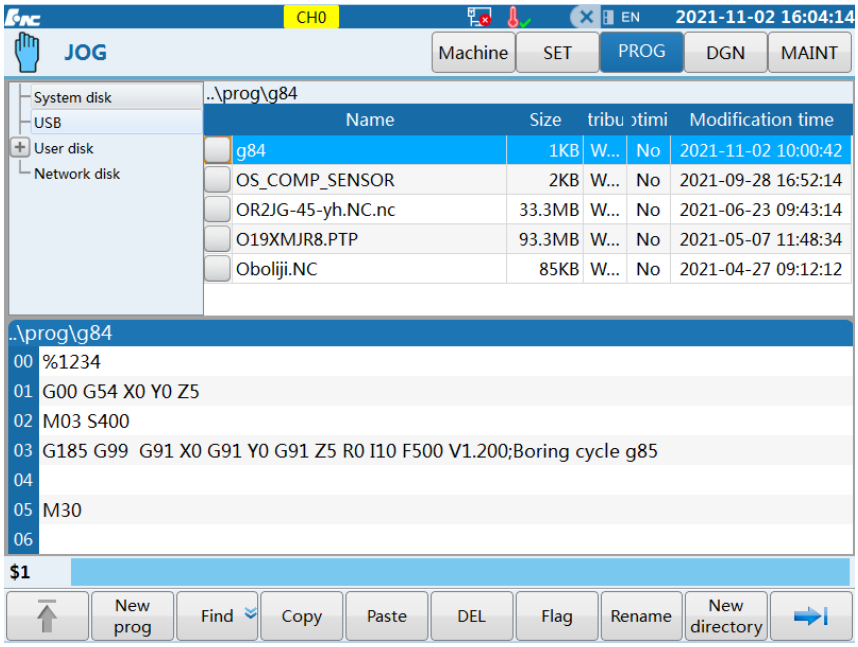
Name sort	Time sort	Writable	Readonly

- 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.
- 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, readonly: Set program files as writable or readable.

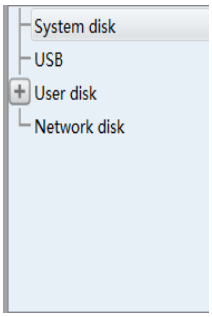
After startup, press 〖Prog〗 function key to enter the default interface of "Program" function set, as shown below.

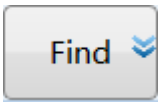
Program can be selected under this interface. Move the cursor to file name of the program to view the first few lines of the program for ease of program identification.



3.4.2 File Management in System Disk, USB Flash Disk and Online Disk

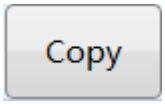
3.4.2.1 Management program search





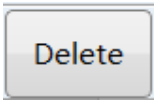
- Select the areas where the program searched may be located under the "Program" default interface, namely 『System disk』, 『USB』 and 『Online disk』;
- If the program to be searched is in the file directory, press 「Enter」 to open it;
- Press『Find』soft key, activate the input box, prompting to input the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



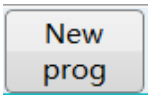
- Press 『Find』 or 「Cursor」 and 「PgUp/PgDn」 under the "Program" default interface to select the program to be copied and pasted;
- Press 『Copy』 soft key and the input box will give a prompt message: Copy succeeds;
- Press 『System disk』, 『USB』 and 『Online disk』to select the target areas
- If the program 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 Deleted will be given.

3.4.3 Create New Programs



- Press 『System disk』, 『USB』 and 『Online disk』under the "Program" default interface to select the areas where a new program is created;

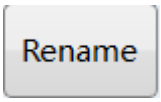
- To create new programs in the file directory, select the file directory and press 「Enter」 to open it;
- Press 『New prog』 soft key and the dialog box will give a prompt message: Ener file name
- Input file name, such as “OHZ1”;
- Press 「Enter」 to confirm input, the working set switches from "Program" 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 prog』 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.

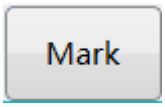
Note 3: While creating a new program under the "Program" function set, the interface and the menu will switch to "Machining" function set automatically, but the new program will not be loaded automatically.

3.4.4 Program Rename



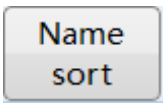
- Press 『System disk』, 『USB』 and 『Online disk』under the "Program" default interface to select the areas where the program to be renamed is located;
- If the program to be renamed is in the file directory, select the file directory and press 「Enter」 to open it;
- Press 「Cursor」 or 「PgUp/PgDn」 to move 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: Enter 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.

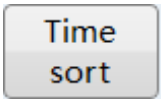
3.4.5 Mark Program



- Press 『System disk』, 『USB』 and 『Online 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 『Mark』, then the program name is prefixed with "√".

3.4.6 Programs Sorted by Name and Time





- Press 『System disk』, 『USB』 and 『Online 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 Setting

- Press 『System disk』, 『USB』 and 『Online 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 『Writable』 or 『Readonly』 to set program attributes.

3.4.8 Create a New Directory

- Press 『System disk』, 『USB』 and 『Online 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 message	Alarm history	Ladder		Display status			⇨

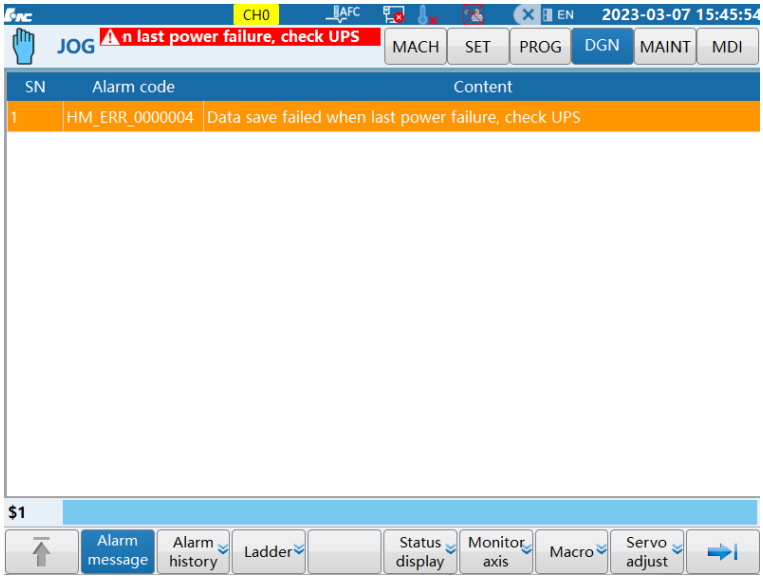
Self check	Fault video	Screw load	Process evaluation	Self-tuning

- 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.

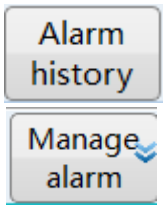
- Self-check: Diagnose the change of health index during operation of machine tool; used for detecting assembling and commissioning consistency.
- Fault video: Record fault related data 10s before the fault occurs. The related data can be preset as position, speed and current of each axis.
- Screw load: 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.

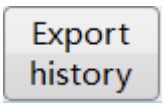
Intelligent functions and interfaces such as "Servo adjustment" and "Self-check" are not introduced in this chapter. For specific operations, refer to the subsequent chapters.

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



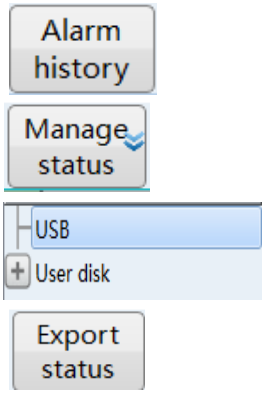
3.5.2 Alarm History Export





- Press 『Alarm history』 soft key under the "Diagnosis" function set default interface;
- Press 『Alarm history』 soft key under the 『Alarm history』 sub-interface;
- Press 『Manage alarm』 soft key;
- Select 『USB』 or 『User disk』 soft keys;
- Press 『Export history』 soft key to export corresponding information to the selected disk.

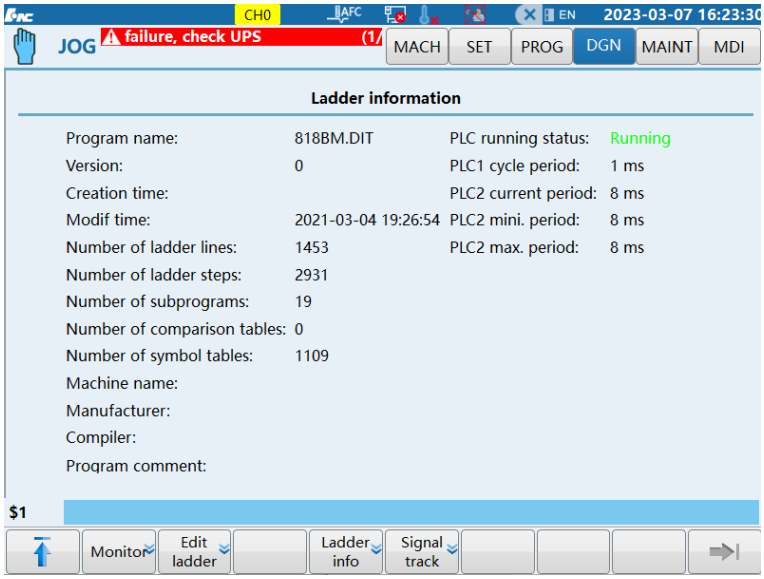
3.5.3 Status Record Export



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

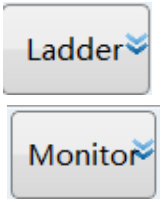
3.5.4 "Ladder " Sub-interface

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 manager or higher permission to enter, please refer to 3.6.5.

3.5.4.1 Ladder monitoring



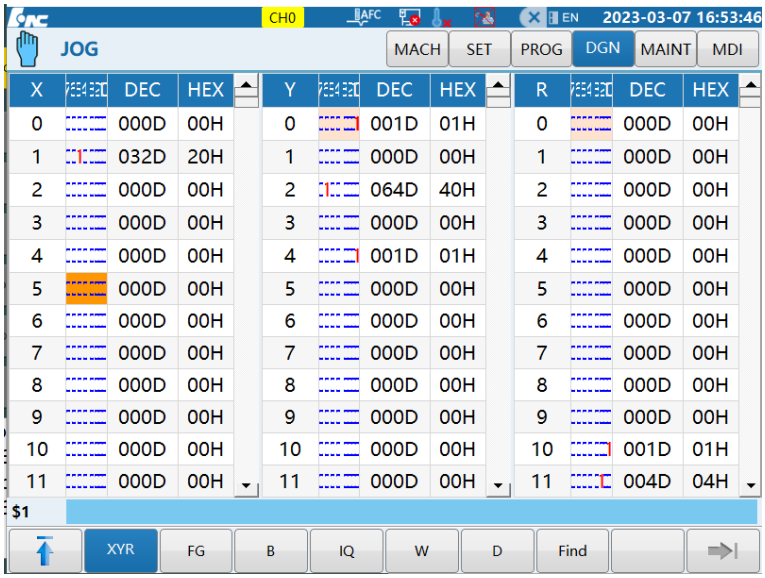
This function is used for monitoring system PLC

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

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 〖DGN〗 function key to enter the default interface of function set;
- Press 『Status display』 or 『Macro』 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

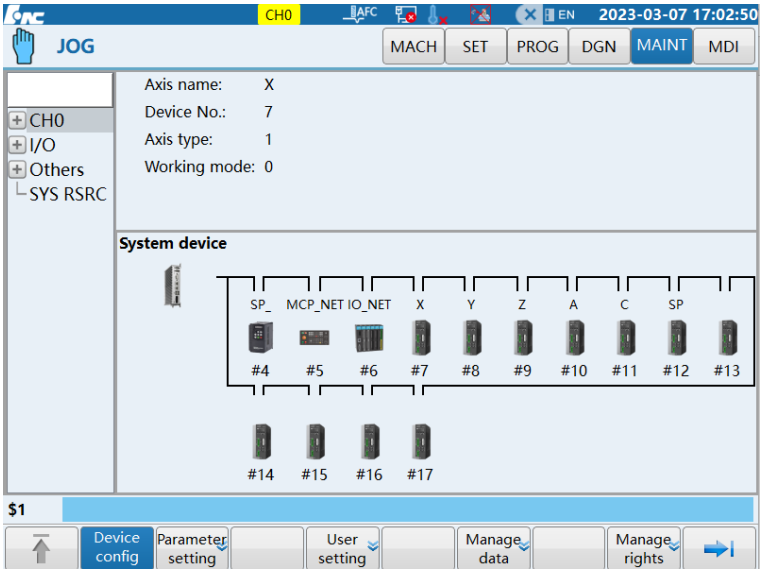
Integrate such functions as parameter configuration, system commissioning and machine tool information under "Maintain" function set. The level 1 main menu and level 1 extension menu of soft key function of "Maintain" function set are shown below.

Device config	Parm setting		User setting		Manage data

- Device configuration: View number of drive, I/O, panel and other hardware devices, and connection sequence of bus;
- Parameter setting: 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.
- User setting: Setting relating to user application. This interface includes display setup, P parameter, M code, PLC switch, communication setup and personalized setup;
- Manage data: Loading and backup of various types of data;

- Compensation: Error compensation setting;
- Machine information: Edit or display machine tool information;
- System information: Display information of the system;
- Zero calibration: To set relevant parameters on zero calibration interface of CNC system to complete phase finding of rotary table and motor zero calibration. This function can assist the debugging personnel to complete the phase sequence test and zero calibration of the third-party motor quickly and effectively.

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



3.6.2 Parameter Setting

1) Parameter selection

- Press 『Device config』, 『Parm category』 and 『Parm setting』 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 setting column, press 「Enter」 to activate the input box

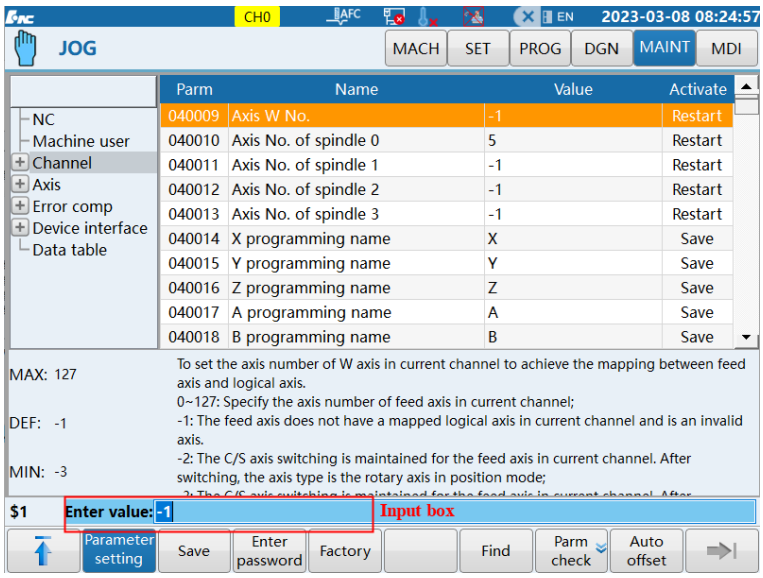
3) Parameter input

- 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.



Automatic system permission logout

- Log in the permission of manager or higher, press MAINT -> Parm setting to enter parameter setting interface for modification.
- Select the NC parameter 000412 to set it. For example, if 60 is filled, then the automatic logout time is 60 seconds.
- Click save and confirm
- Click reset, and parameters take effect. If there is no operation of system in 60 seconds, system will restore to the default permission.

Note:

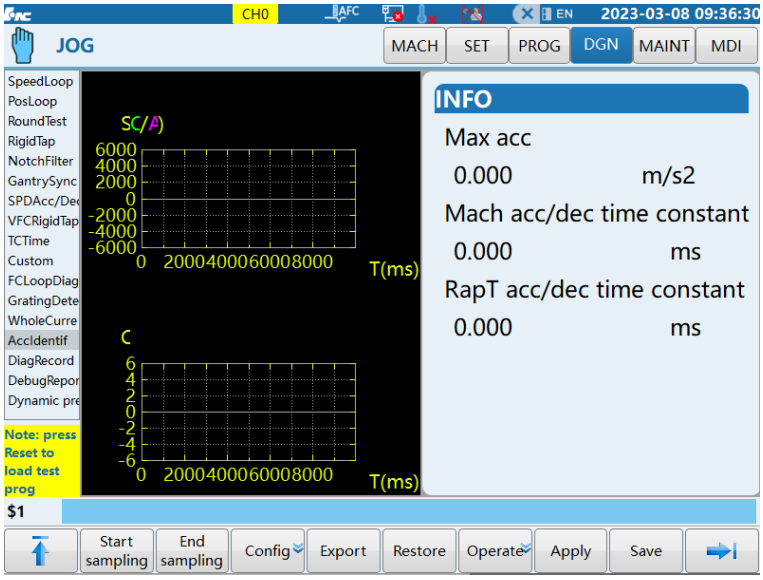
When 000412 is set to 0, the function is turned off

When the current permission is larger than default permission, the function takes effect.

The power-on default permission is controlled by the NC parameter 000359.

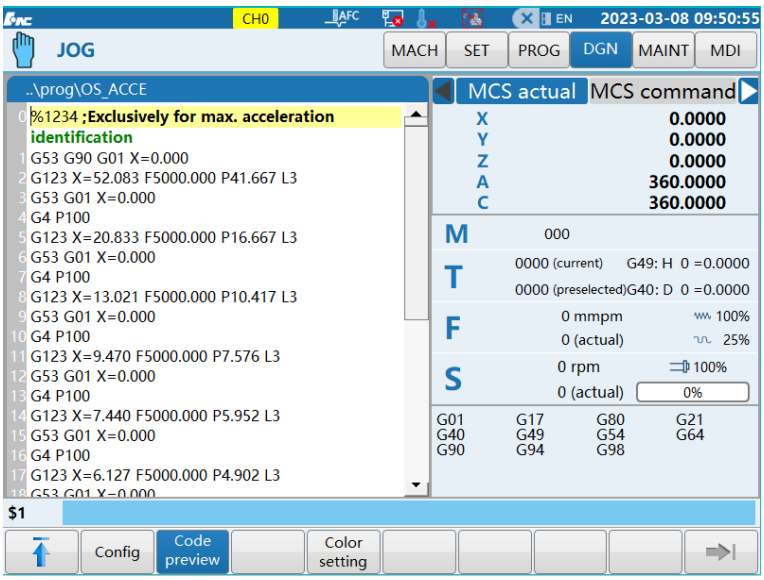
Maximum acceleration identification

- To use this function, users need to click “Enter password” in “Maint” → “Parm setting” to enable the machine manufacturer permission or higher.
- Press “DGN” → “Servo adjust” to enter servo adjustment interface, move to “Acceleration identification”

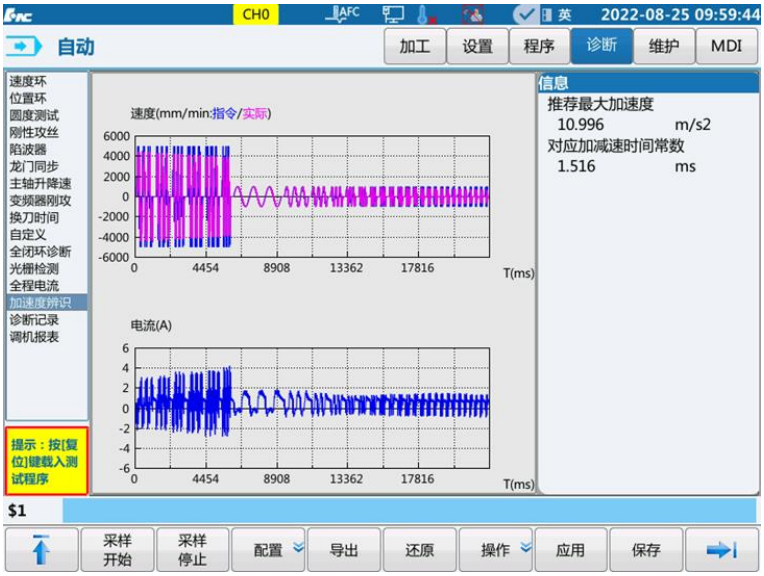


- Enter “Config” interface to configure identification axis number, max. frequency of excitation signal, axis starting position, triangular wave identification speed, and sine wave identification speed.

- Click “Code preview” to view the G code of maximum acceleration identification debugging.

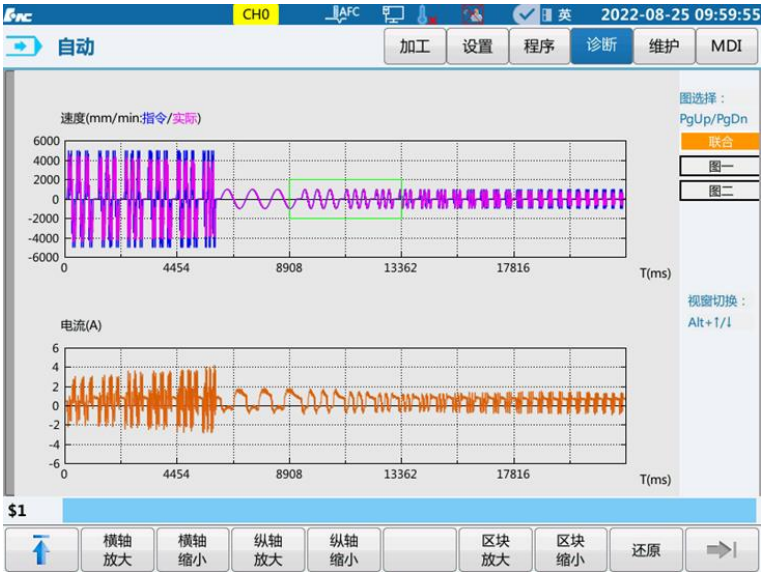


- After configuration, press ⬆ to return to the main interface of acceleration identification sampling, press Reset to load the G code.
- Press “Cycle start” to run G code of acceleration identification. On the acceleration identification sampling interface, users can monitor speed data curve and current data curve of current axis in real time.
- After program is executed, system will generate the recommended maximum acceleration value, corresponding acceleration/deceleration time constant, and speed and current wave.

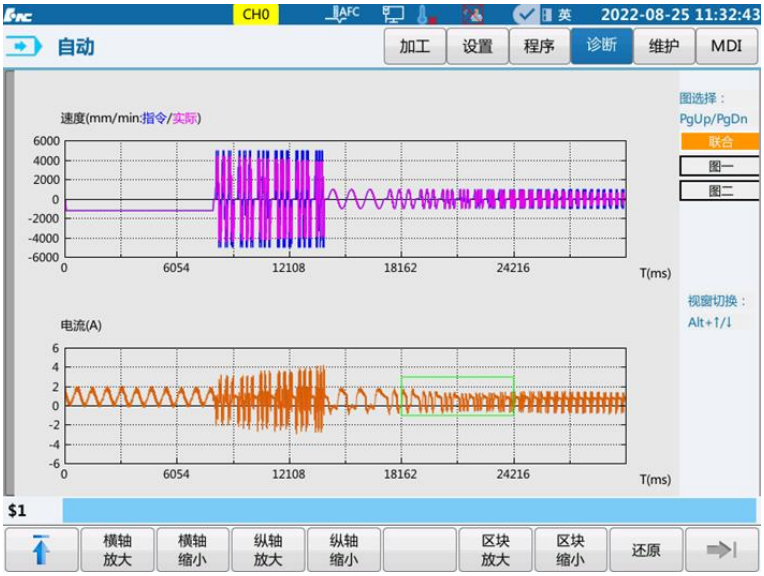


Click “Save” to write the “corresponding acceleration deceleration time constant” to the coordinate axis parameters #36 and #38 (for example, 100036 and 100038 for x axis).

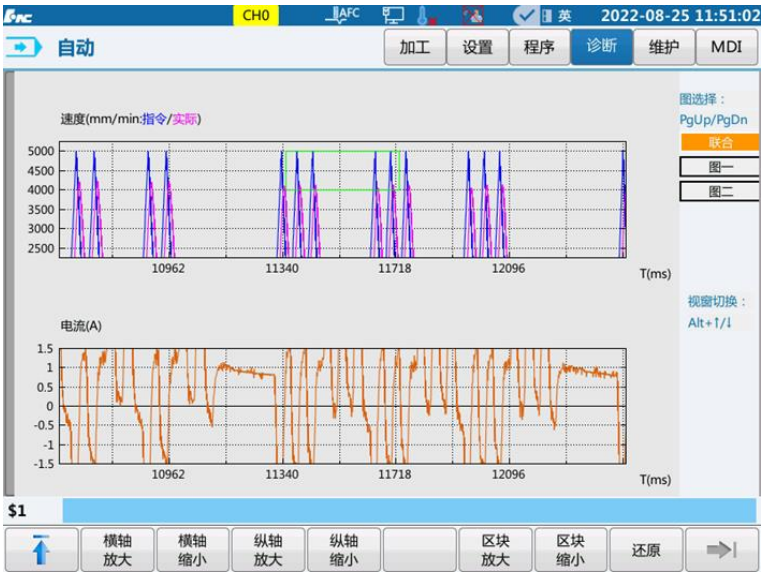
Enter “Operate” interface to view and analyze speed and current sampling data.



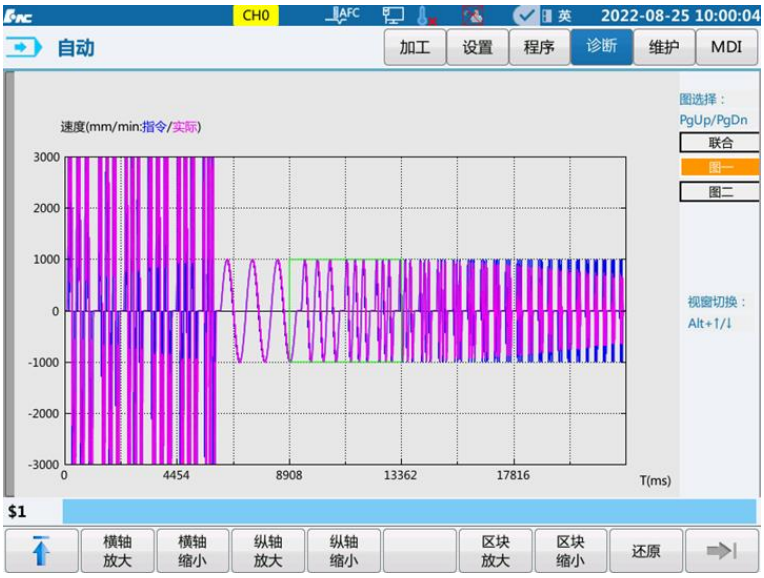
Press Alt+ ↓ / ↑ to switch graphics view.



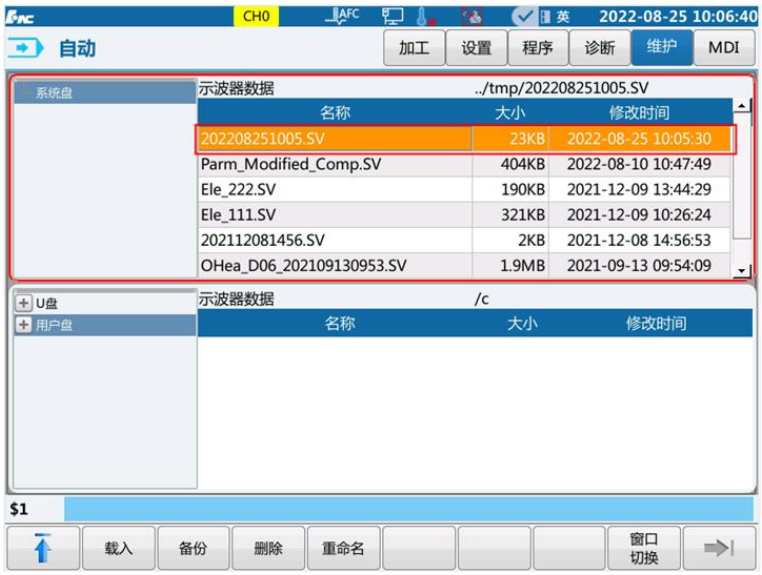
Press “Horizontal axis zoom in”, “Horizontal axis zoom out” , “Vertical axis zoom in”, and “Vertical axis zoom out” menu to zoom in and zoom out the sampling graph.



The sampling graph can be switched by “PgUp” and “PgDn”



Click “Export” to transmit the sampling data to the oscilloscope data directory of manage data interface, and then back up the data to USB.



Note:

If on the acceleration identification interface, the generated “recommended max. acceleration” and “corresponding time constant” are abnormal, or the two parameters are not generated, increase or reduce maximum frequency of excitation signal ranging from 20 to 30;

“Axis start position (G53)” on the “config” interface: the programming is based on the actual machine position G53, and the movement is in positive direction of axis. Therefore, it is recommended that start position < Positive soft limit – 100mm;

When running acceleration identification, it is not allowed to interrupt the program by clicking feed hold or switching working mode; otherwise, system will prompt “not at breakpoint position”;

V2.42 currently only supports the debugging of the coordinate axis parameter “acceleration deceleration time constant”, not support debugging of “acceleration deceleration jerk time constant”.

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:

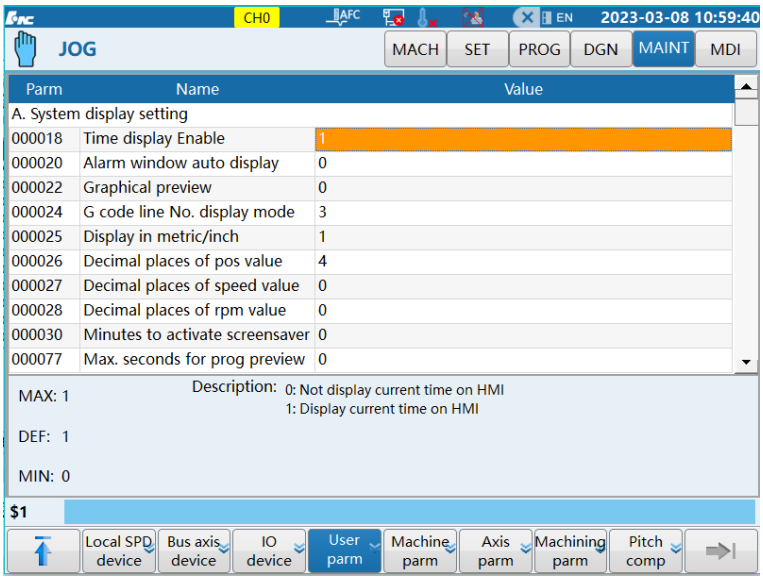
- Parameter setting and modification are limited, input a password of corresponding permission while setting and modifying parameters.

3.6.4 "Parameter Category" Sub-interface

The "Parameter category" 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 buttons 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 "Parm setting" sub-interface are limited functions, and password should be entered to enable them.



3.6.4.1 Direct input of parameter category value

- Press 『User parm』, 『Machine parm』 and 『Axis parm』 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 "Successfully set, 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 "Saved successfully" will be given and parameters will be validated immediately.

3.6.4.2 Direct input of pitch error compensation value

- Press 「Pitch comp」 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 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, manager, machine provider, CNC provider and administrator. Management functions of various permissions are roughly as follows:

"Administrator": Development, test and customer service with system software maintenance permission.

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

"Manager": 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.

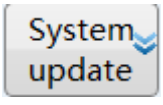
Operation function \ Permission type		Administrator	CNC provider	Machine provider	Manager	Operator
Parameter Classification	User parameter	Yes	Yes	Yes	No	No
	Machine parameter	Yes	Yes	Yes	No	No
	Axis parameter	Yes	Yes	Yes	No	No
	Pitch error compensation	Yes	Yes	Yes	No	No
	I/O device	Yes	Yes	No	No	No
	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 setting (except display setting)		Yes	Yes	Yes	Yes	No
Data management		Yes	Yes	Yes	Yes	No
Spatial compensation		Yes	Yes	Yes	No	No
Time setting		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	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 setting	Yes	Yes	Yes	Yes	No
	User macro	Yes	Yes	Yes	Yes	No
Setup function set	Tool compensation	Yes	Yes	Yes	Yes	Yes
	Coordinate system	Yes	Yes	Yes	Yes	Yes
	Workpiece measurement	Yes	Yes	Yes	Yes	Yes
	Magazine, tool life setting	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 "Manager" 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 〖Manage rights〗 soft key to enter the "Permission management" sub-interface
- Press 〖Log out〗 soft key to exit the current permission;
- Press 〖Left and right cursors〗 to select the required permission;
- Press 〖Lo
- gin〗 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.

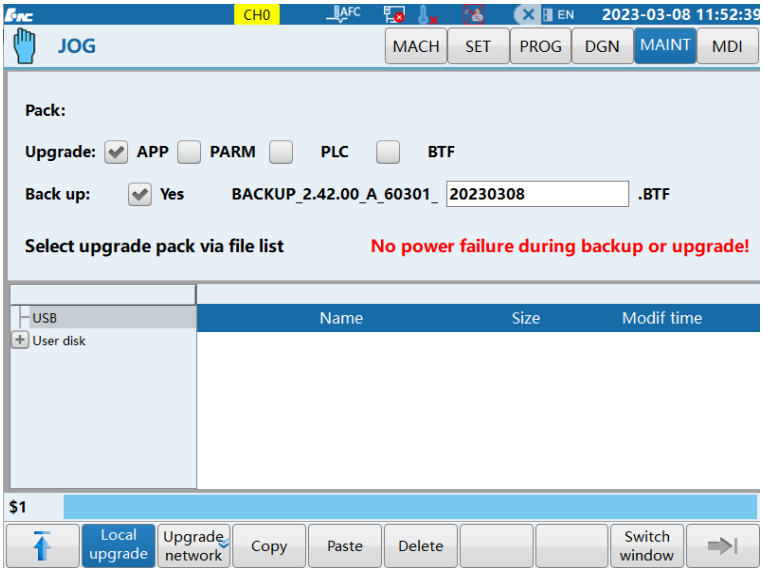
3.6.6 System Upgrade



"System upgrade" function is available for the system manufacturer only.

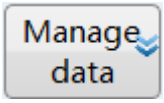
Thus, users should set the permission under the "Maintain" function set after startup (The permission is not saved after shutdown).

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



- Press 『Switch window』 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 『Switch window』 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);

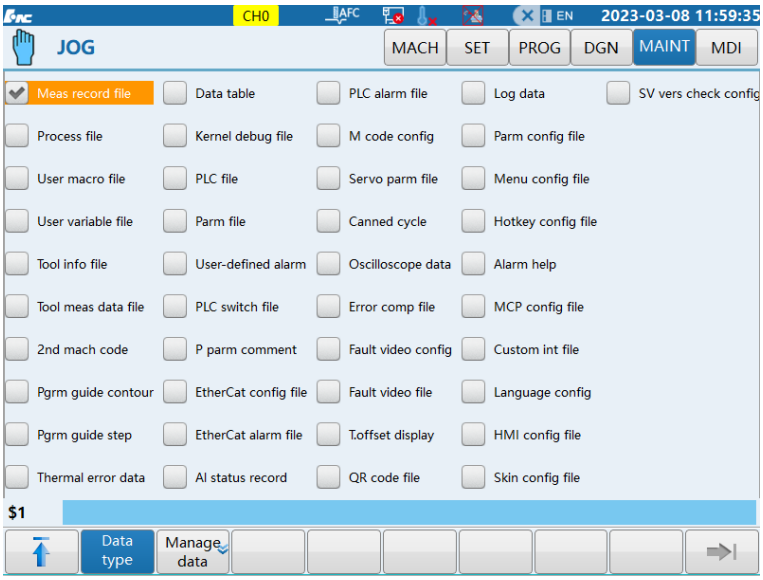
3.6.7 Data Management



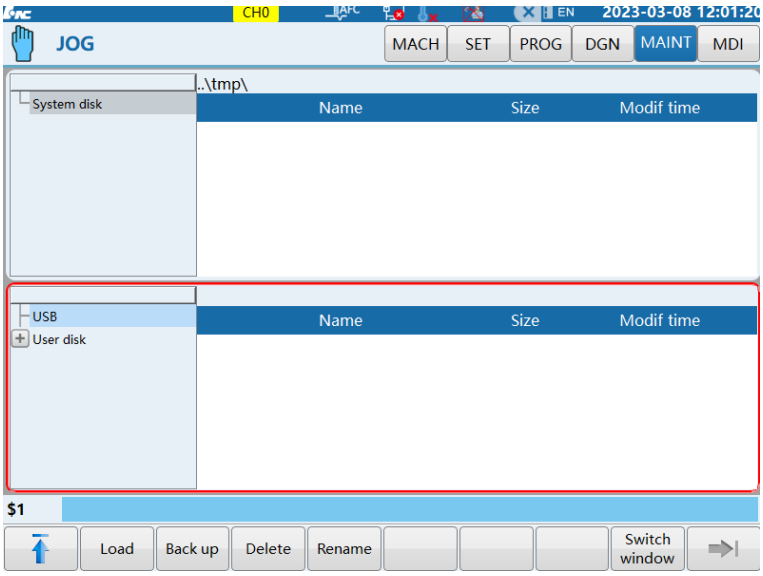
In the data management interface, single file of parameters, PLC, canned cycle, log, compensation, oscilloscope can be loaded/backed up. This section takes the example of loading/backing up system parameter files. The operation steps of loading and backing up other files (except the error compensation file) are the same.

Press 『Manage data』 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』 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 『Switch window』 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.

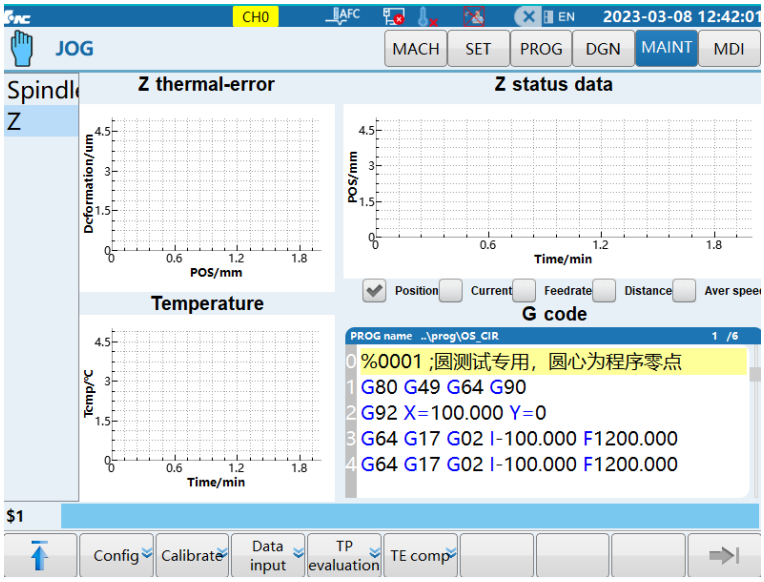
➢ To back up data in system disk in USB flash disk or user disk, press 『Switch window』 soft key to move the red box to the system disk above the above figure;

- Press 「Enter」 to open the file directory and press 「Cursor」 to select the data file to be backed up;
- Press 『Back up』 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.

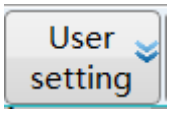
Note: Power failure is not allowed during backup or loading.

3.6.8 Thermal Error Compensation

In the main menu of MAINT function set, press “Comp” soft key , and “TE comp” to enter the submenu.

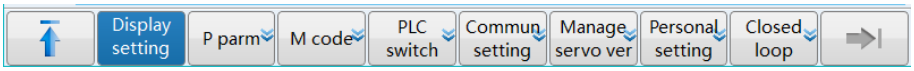


3.6.9 User Setting



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

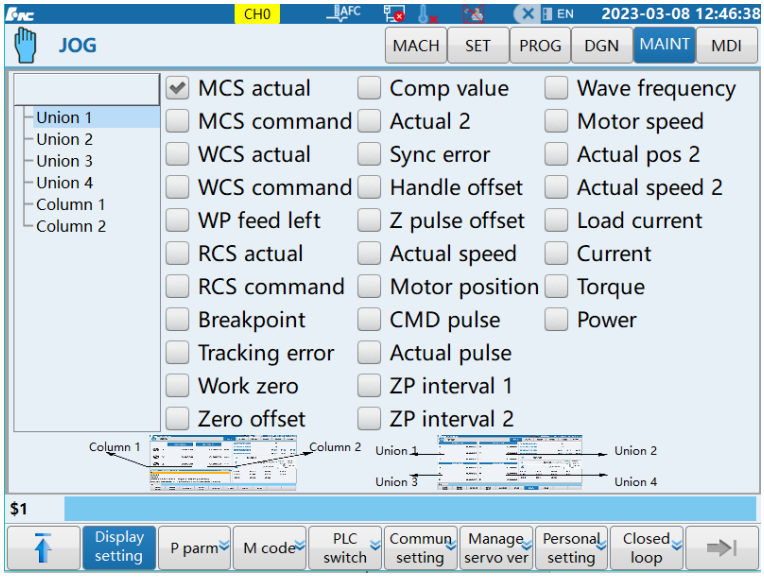
Press 『User setting』 soft key under the main menu of the "Maintain" function set to enter the "User setup" submenu (as shown below).



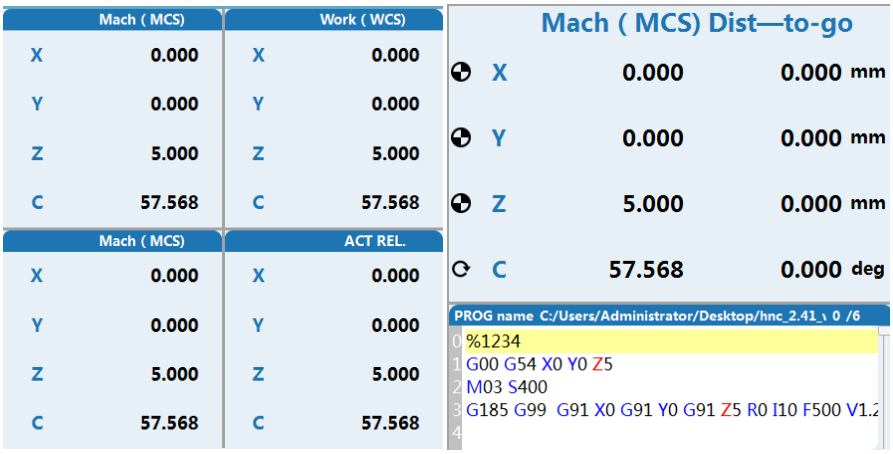
3.6.9.1 Display setting

As mentioned in the introduction of the processing interface, the soft key "Switch display" under the "Machining" 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 setting』 and enter “Display setting”



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

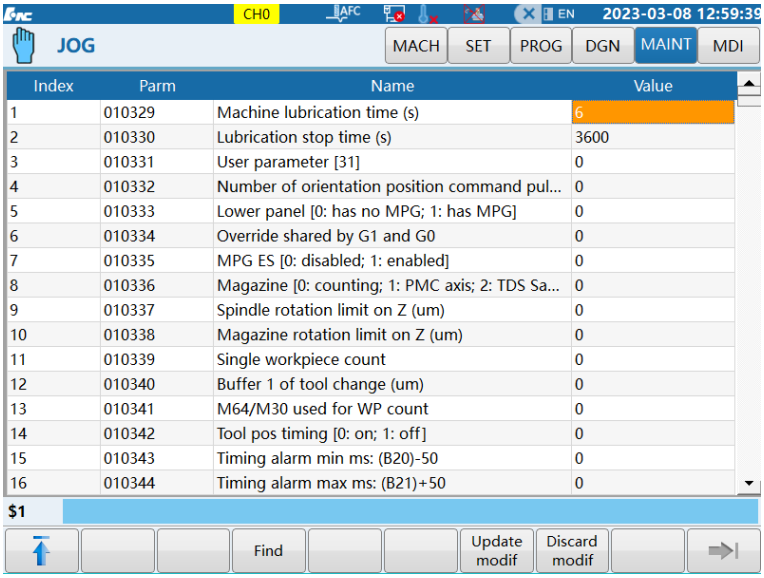
When the cursor moves to the left column 2 in the above figure (display columns 1-2), the content of the right column 2 is displayed on the right of the above figure and the display content in the "Big character coordinate + program" interface can be selected from the right column 2 (as shown in the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

3.6.9.2 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 parm』 soft key under the "User setting" sub-interface to enter the "P parameter" sub-interface (as shown below).

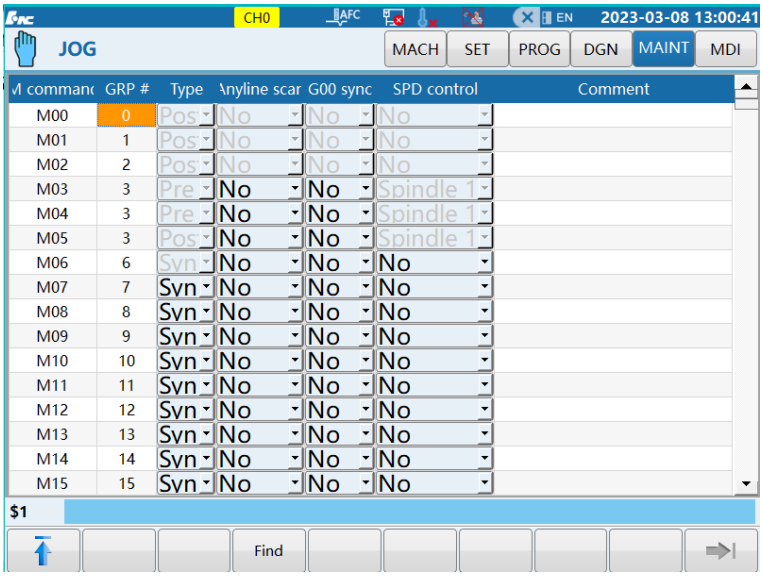


It should be noted that when a function is enabled through setting P parameter, not only should PLC switch be set, but also other relevant parameters and functions should be set. e.g.: When rotation function of mill spindle is enabled, not only should parameters of spindle rotation ON/OFF be set, but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

3.6.9.3 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).



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 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 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.

Note: 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;

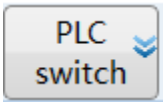
No: M code and G00 code cannot be executed synchronously.

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.

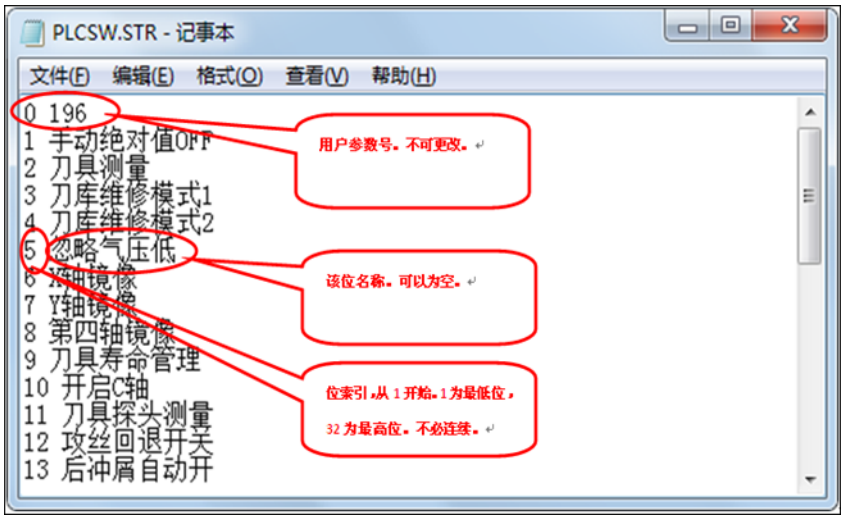
d) Spindle control

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.

3.6.9.4 Set "PLC switch"

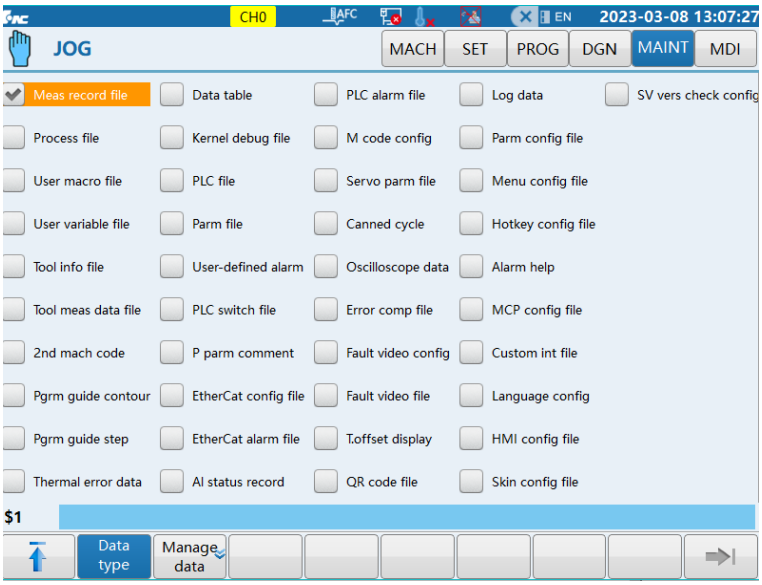


PLC switch setting function is to decompose the designated user P parameter to 32 bits and every bit is a configurable PLC switch. PLC switch is configured by PLCSW.STR configuration file under parm directory. The file format is shown below:

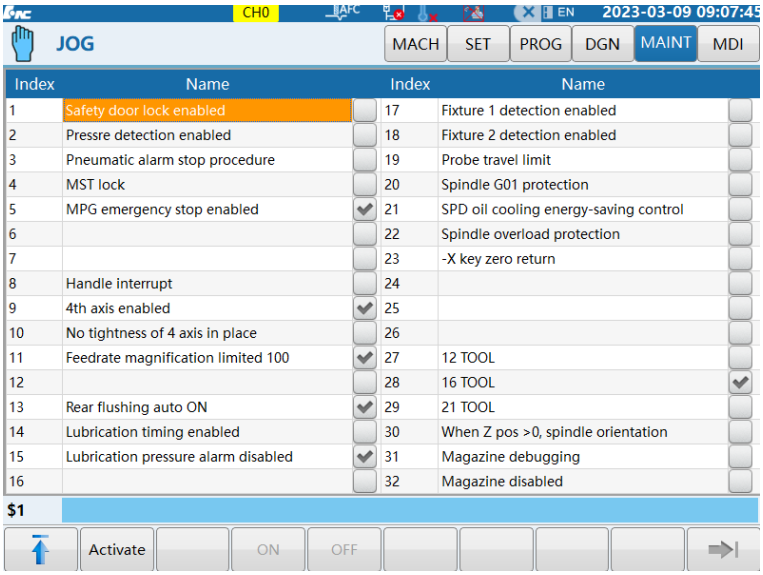


Note: Machine user parameter P196 of HNC-9 system is used as the PLC switch file by default. The setting is closely related to PLC function and cannot be modified without the guidance of CNC system commissioning personnel.

After PLCSW.STR file is made, select “PLC switch file” in “Manage data” to import PLCSW.STR file into the system. As shown below.



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



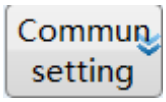
Effective setting: Protect “ON” and “OFF” menus which can be operated only through pressing "Activate " menu;

ON: Set to ON;

OFF: Set to OFF;

Setup result is saved in the designated user P parameter.

3.6.9.5 Communication setting



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

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

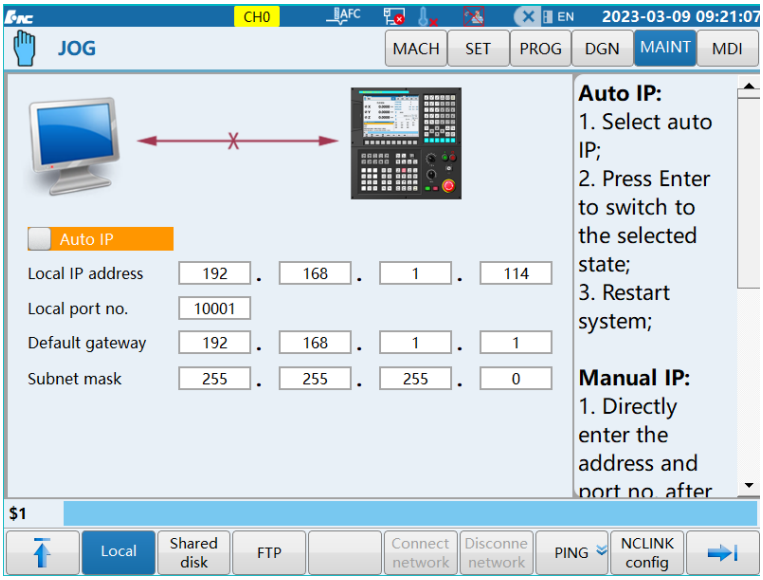
a) Communication between the upper computer of the CNC system and the computer

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,

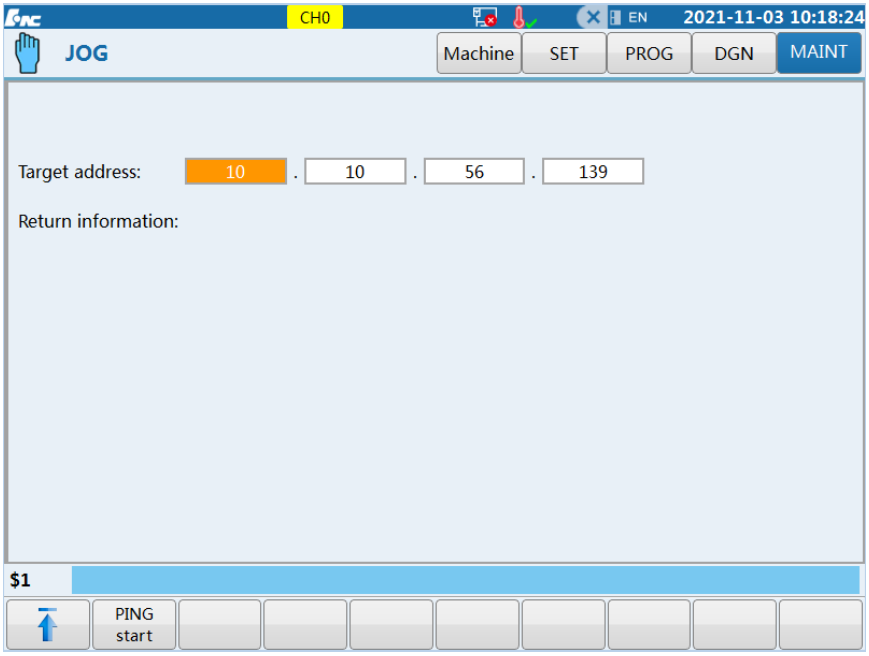
ip of the CNC system is 192.168.1.113 by default. ipv4 of the computer is set to 192.168.1.XXX at the time of connection, and default gateway and subnet mask of the computer are consistent with those of the CNC system. Specific steps:

- 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 figure);

- 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

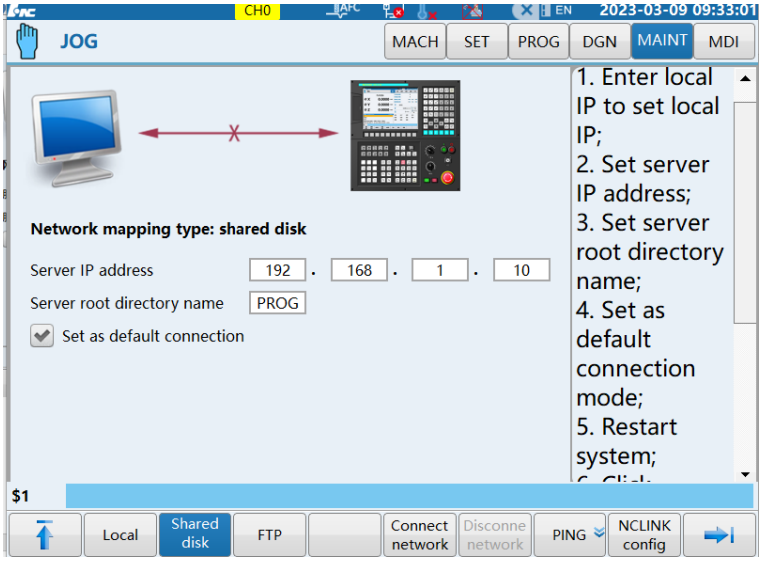
in PING interface and click on 『Start PING』 soft key;

- 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 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 setting』 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

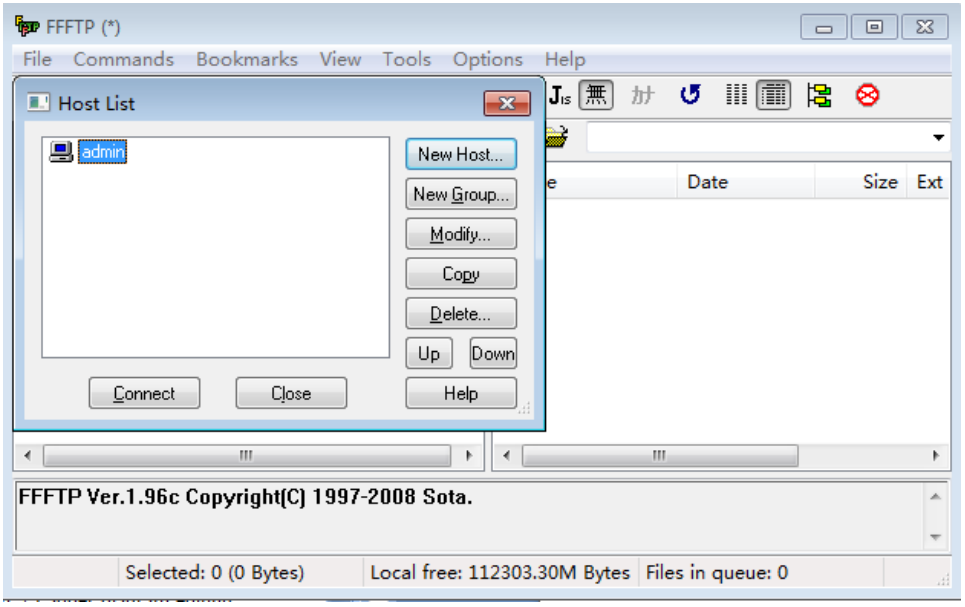
FTP is a kind of connection software used to copy data from the computer to the system or from the system to the computer.

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

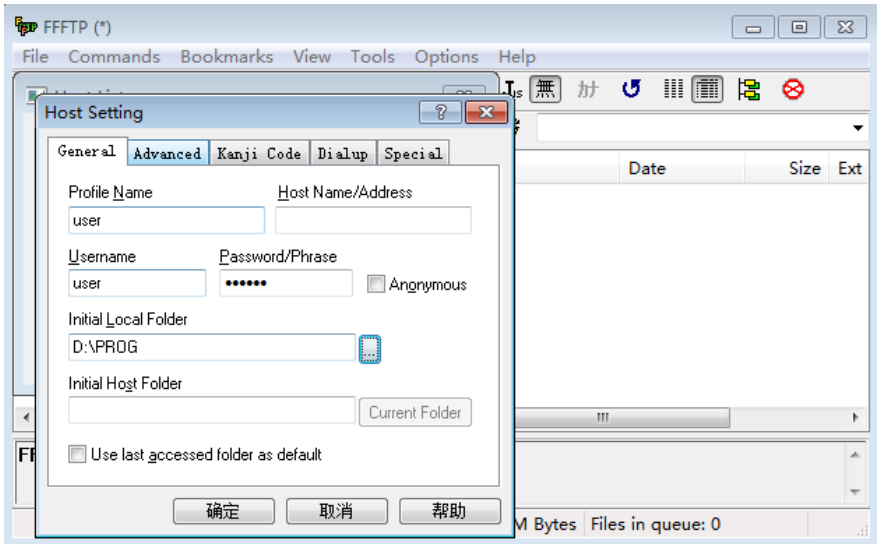
Then, install the software FFFTP on computer (download from the official website of HCNC, see the right icon)

After ping the system and the computer, click on FFFTP icon and the

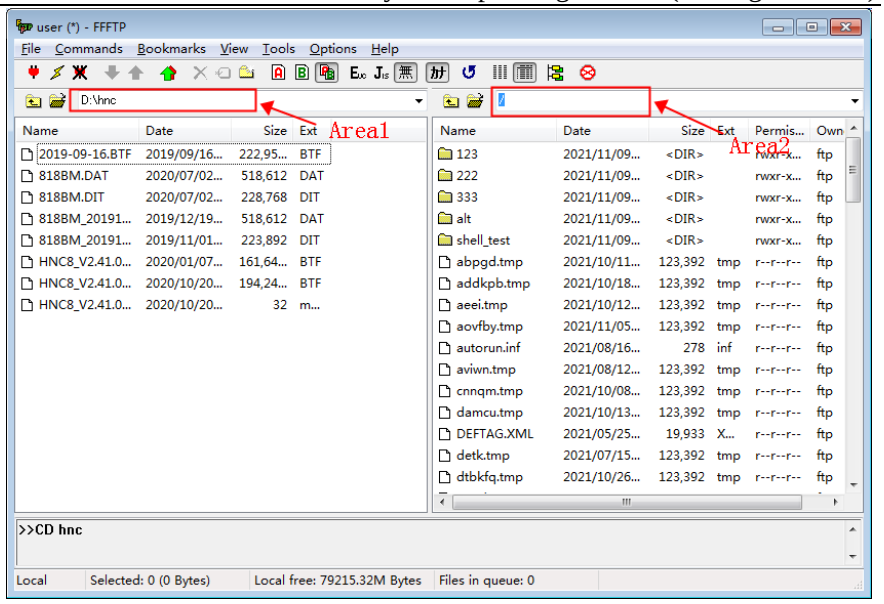
interface is displayed as below:



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.

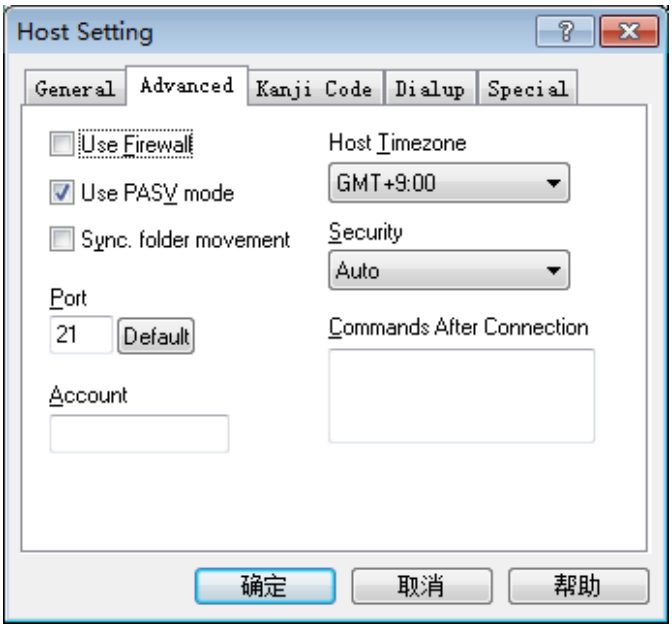
After a path is designated, drag the file using mouse and transmit files between the system and the computer.

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,

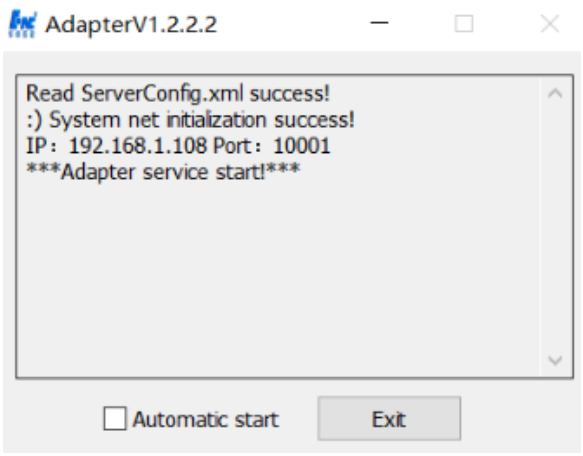
Setup steps for HCNC controller adapter software are as follows (it should be noted that 1.26 and 2.XX series adapters are different).

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

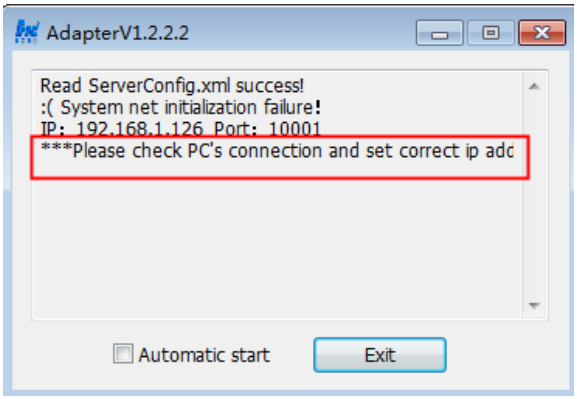
(2) Open Serverlp.xml and replace LocalIP to IP of this computer, which is 192.168.1.108 in this example. The port number must be consistent with that on the system and should be edited and saved.

```
1  <?xml version="1.0" ?>
2  <ServerConfig>
3      <NetConfig LocalIp="192.168.1.108" Port="10001" />
```

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

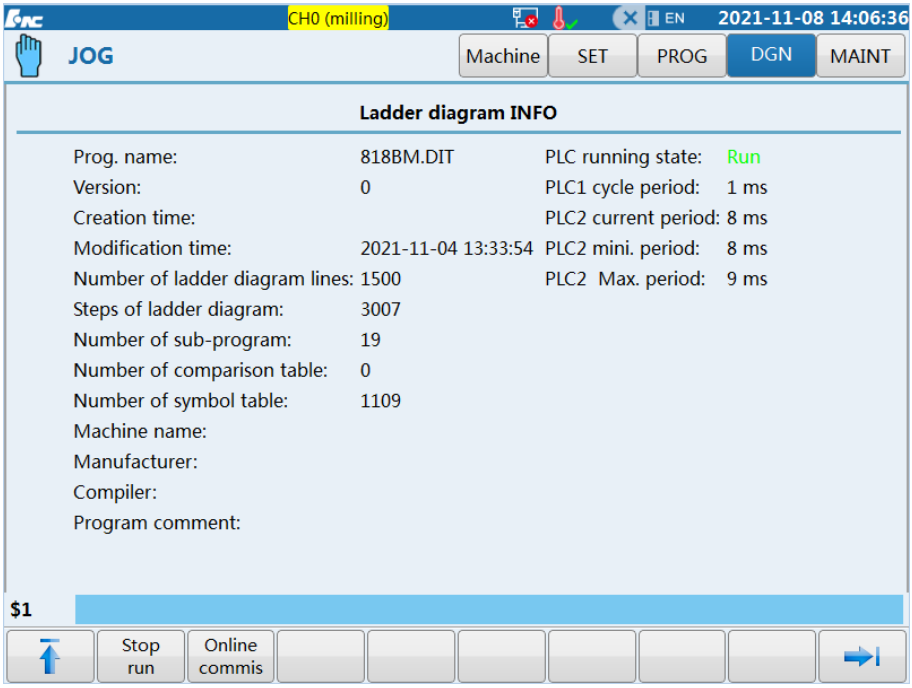


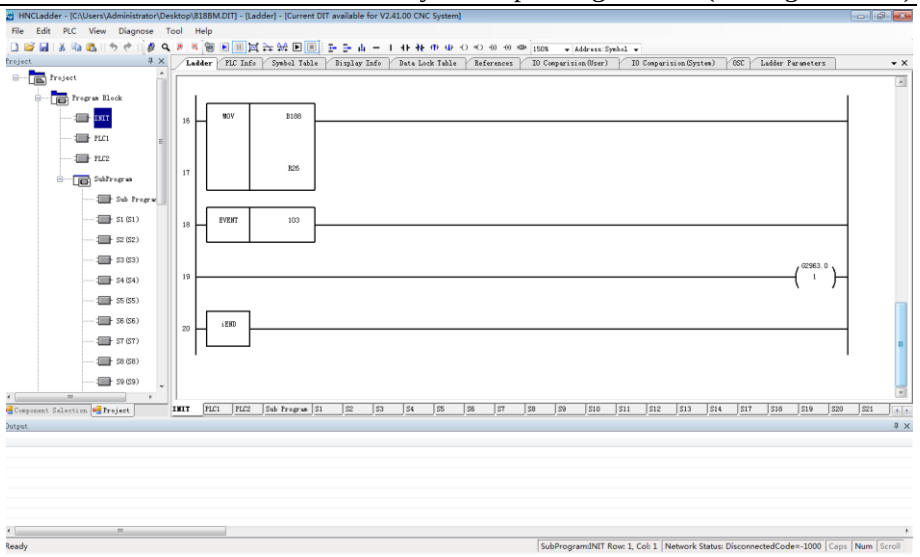
If Serverlp.xml is not modified, an alarm will be given as shown in the following figure

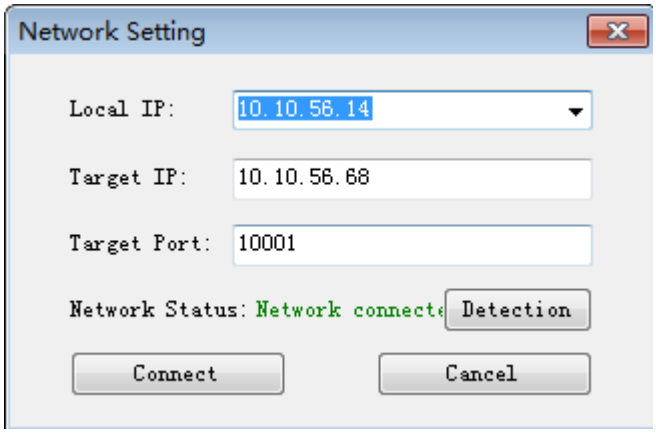


(4) Find the online commissioning in the second page of ladder ----ladder information menu under the DGN interface and click on it, the system will give a prompt message that PLC is being commissioned online.

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

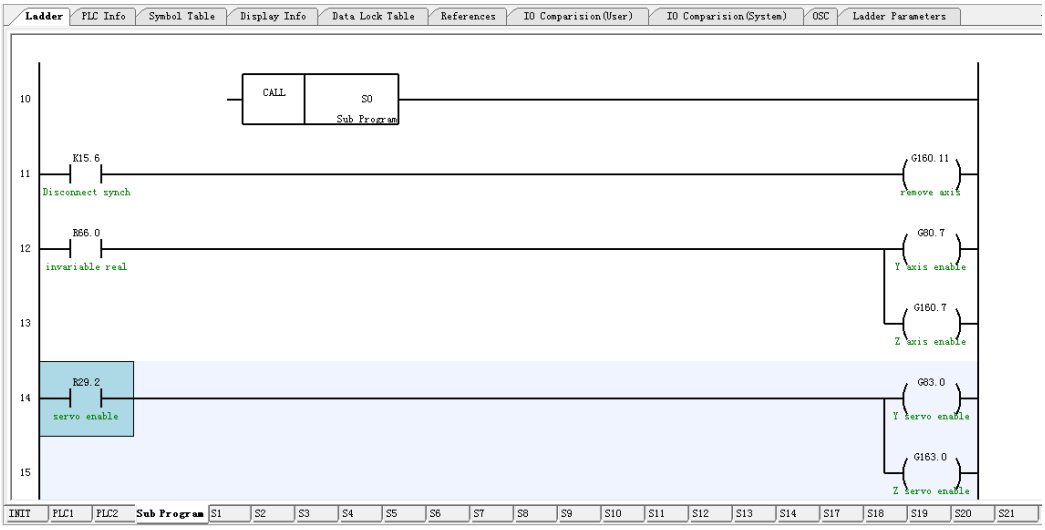






Correctly fill in IP of the computer and IP of the system and click to connect

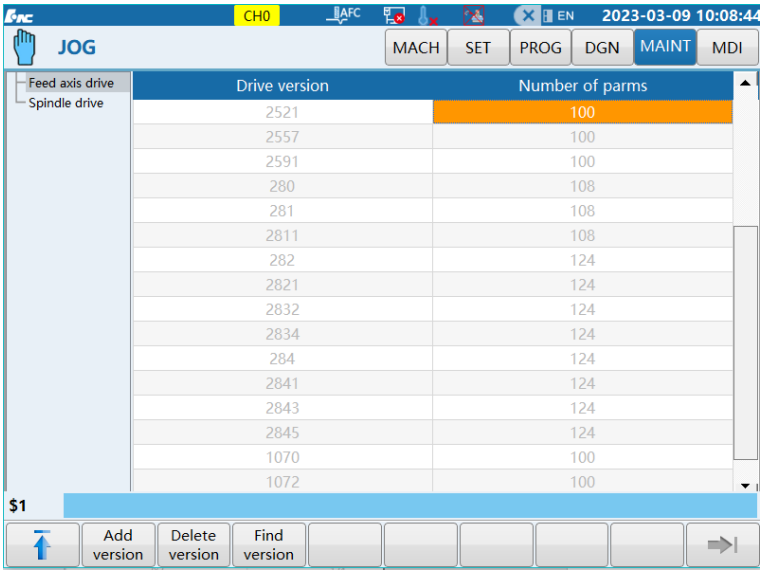
(6) Click on "Start program monitoring" under the diagnosis menu, and the PLC commissioning software will load system PLC automatically, as shown below.



e) Servo version management

MAINT → User setting → Manage servo ver (lower than permission of administrator)

On the interface, the system can display feed axis drive, spindle drive version number, and number of servo parameters, and users can make a modification on the system and add version information of servo drive.



Add version

Add servo drive version number of current feed axis and spindle and number of servo parameters to the servo version management interface.

Delete version

Delete servo drive version number of current feed axis and spindle and number of servo parameters to the servo version management interface.

Find version

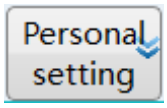
Look for servo drive version number of current feed axis and spindle and

number of servo parameters on the interface.

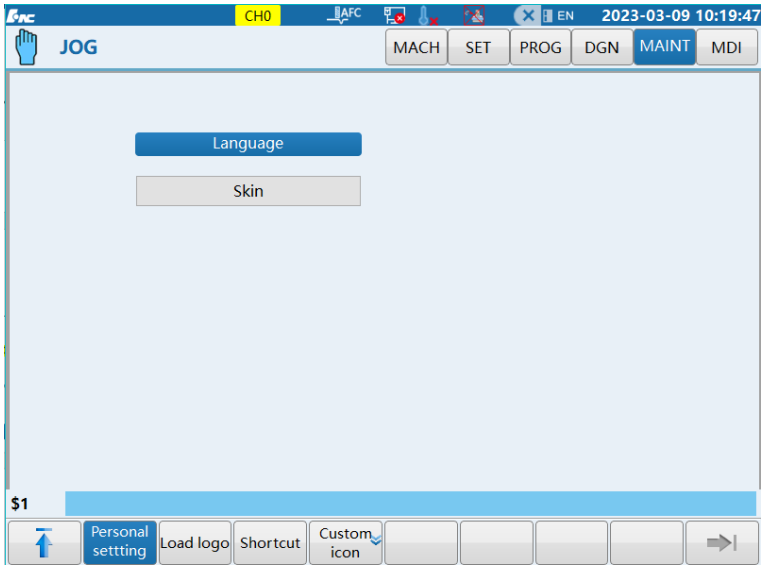
Note: Cannot add existing content of the version number;

For the area in grey, it indicates the configuration item of which unchanged attribute is 1, and cannot be modified on system.

3.6.9.6 Personal setting

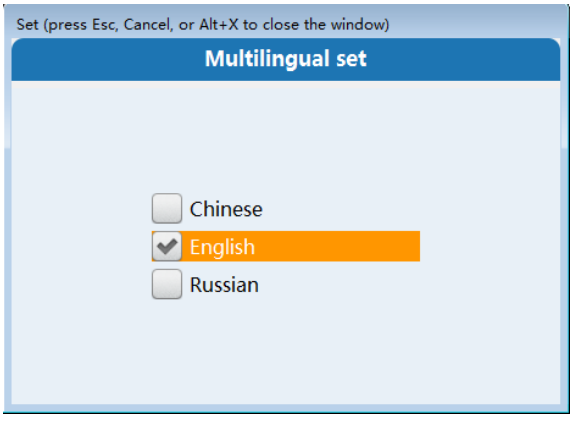


The system version can be realized through personalized setting: Language setup, resolution setup and skin setup. Press『User setting』soft key under "Maint" function set to enter the personal setting sub-interface, as shown below. The modification should be validated after restart.



1. Language setup

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



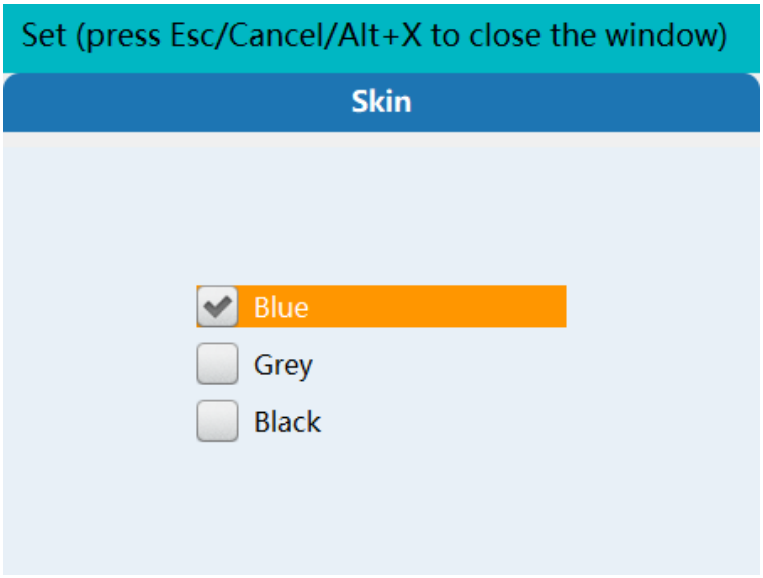
2. Resolution setup

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).

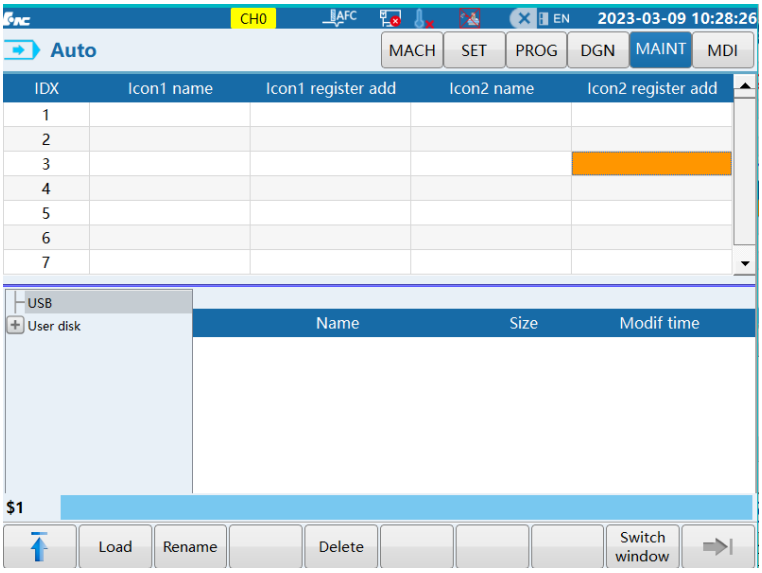
3. Skin setup

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



Two display position are provided form custom icons on information bar. Different icons can be displayed based on the status of register. Each icon can correspond to different registers.

Permission of manager or higher can enter “Custom icon” setting interface by Maint → User setting → Personal setting → Custom icon.



Switch window: The window is divided into upper part: icon and register information and lower part: file display. User can press the menu to switch the operation area.

Load: To load image to be displayed.

- Select the upper part of the window, move cursor the position to be loaded;
- Switch to the lower part, move the cursor to the image file to be loaded;
- Click Load;
- Switch the window to fill in the corresponding register address.

Delete: To delete the image loaded

- Select upper part of the window and select the image to be deleted;
- Click delete, and confirm;

Rename: To rename the original file

- Switch the window to the lower part, and select the file to be modified;
- Click Rename.

Note:

1. Up to two groups of icon are supported, and each group of icon supports 8 register status at most.

2. When multiple registers are all turned on, the last icon is displayed according to the order in table.

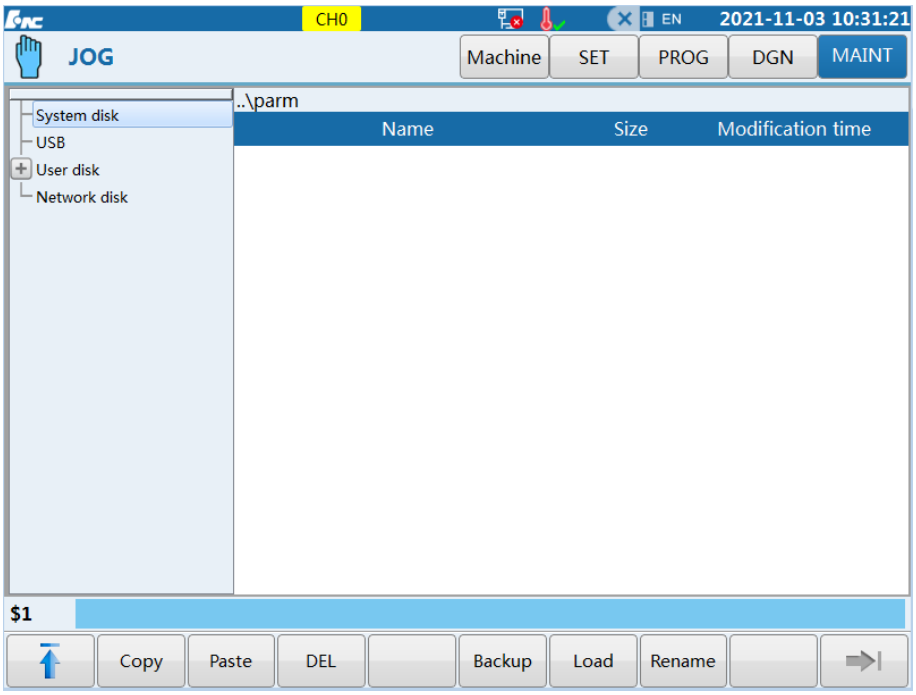
3. When all registers are not turned on, not icon is displayed.

4. The image is in the format of user**+.png, 50X50

3.6.10 Process Package Setting

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



- Back up

  Users can select an XML file and click on Back up to back up the parameter values corresponding 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 corresponding parameters 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 online 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;

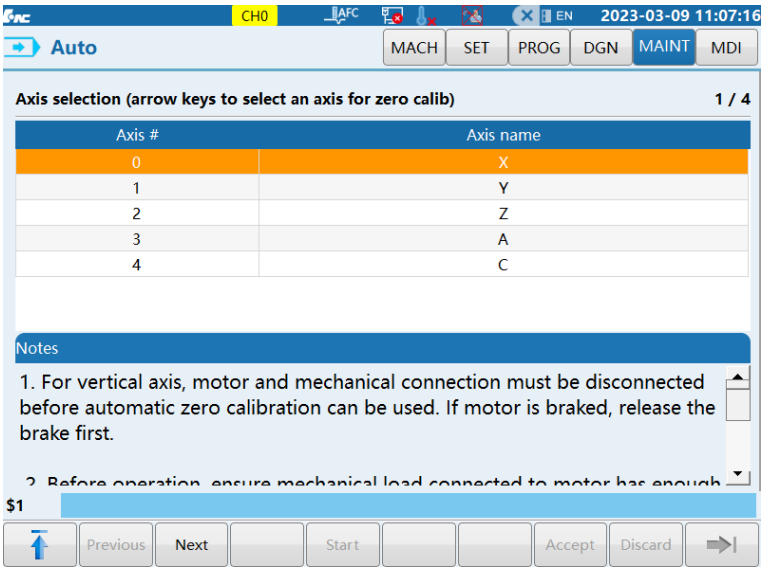
no is the parameter number;

val is the parameter value;

3.6.11 Phase Finding and Zero Calibration

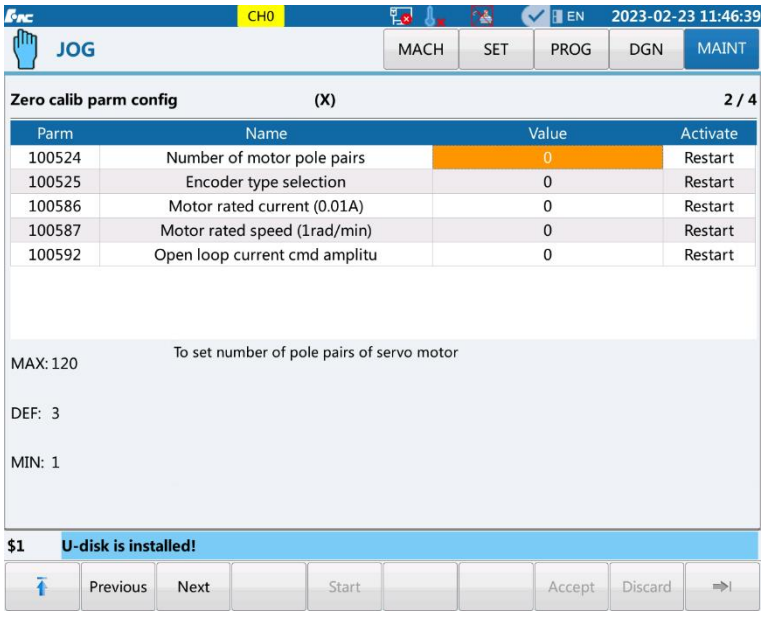
The function is suitable for the feed axis of third-party motor. System software V242 or higher, servo software version V2845 or higher (V2845 firmware. FPGA: 4666; DSP: 2845).

In MAINT, click “Zero calib” to enter zero calibration interface.



After system enters zero calibration interface, all configured feed axis are identified and the notes are displayed. Acoording to the prompt text, check whether mechanical connection of motor is loosened, check whether motor has enough safety travel, and check motor encoder type.

Select the axis need phase finding and zero calibration, click Next, enter key parameter setting interface, and correctly set number of motor pole pairs, encoder type, motor rated current, motor rated speed, current amplitude of oepn loop.



Click Next, system will automatically clear the phase angle of the servo parameter PA26 “Encoder zero offset”. Set the PA23 parameter “Control mode selection” to mode 5 “Auto zero calibration mode”, and prompt to restart the system (system and servo need to be powered off and restart). If not restart, the zero calibration key keeps disabled to avoid misoperation.

Note: if the spindle axis which has no servo is selected, system will prompt “Current servo cannot switch to zero calibration mode” and the “Start” key is shielded.

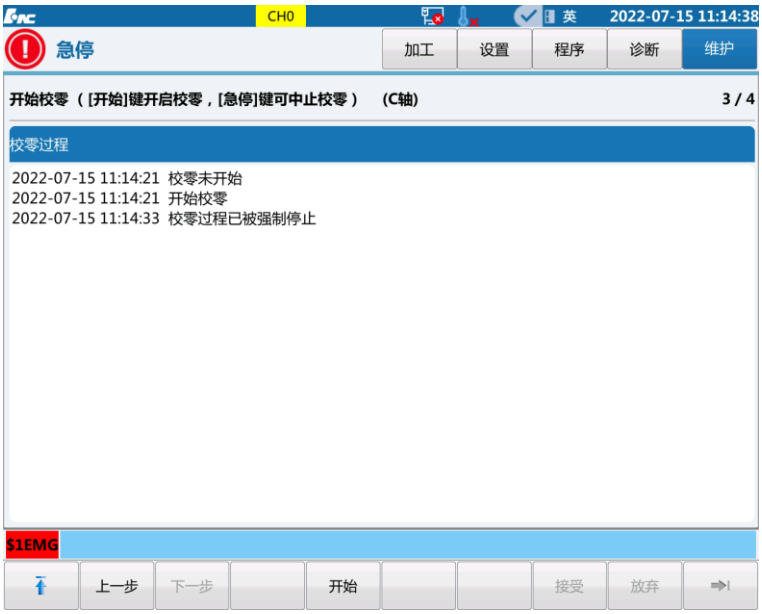
After restarting, system automatically goes to zero calibration inteface and the window pops up to remind “Confirm current brake is release before zero calibration”, press Enter or Cancel to close the window and start the zero calibration.

Confrim that mechanical connection of motor is loosened, brake is released, and motor has enough safety travel before selecting “Start”. If incremental encoder is used, users need to manually rotate motor to find Z pulse of the motor.

Release emergency stop and enable it, the click “Start”. Motor starts to move a magnetic pole distance in both positive and negative directions for

phase finding and zero calibration.

a. During zero calibration, the process can be terminated forcely by pressing emergency stop. After that, users can also relesase emergency stop, enable it, and press “Start” to start zero calibration again.



If the phase sequence is incorrect, the servo alarm A50 “Motor phase sequence incorrect” will be issued. And users need to power off to change the motor phase sequence.

If the phase sequence is correct, system will prompt zero calibration done after the zero calibration action is completed, and will automatically switch PA23 “Control mode selection” to mode 0. The zero calibration key keeps disabled to avoid misoperation.

c. During zero calibration, if motor moves a tool small distance, servo alarms A51, and system alarms “move moves too small”. At the point, users can select “Previous” and increase PB48, and then restart the system to perform zero calibration.

d. During zero calibration, if locked phase angle deviation of motor is larger than ±1° (mechanical angle), servo alarms A52, and system alarms “Zero calibration result deviation too large”. At this point, users need to check whether motor-related parameters are set correctly. After confirming the parameters are set correctly, select “Previous” and increase PB48, and then restart the system to perform zero calibration.

Click “Next” to enter phase angle data confirmation interface.

a. Click “Accept”, system will write the phase angle data into PA26 “Encoder zero offset”, and prompt to restart the system to take effect.

b. Click “Discard”, system will zero out the phase angle data of PA26 “Encoder zero offset” , and prompt to restart the system.

Perform from step 1 if users need phase finding and zero calibration.

After restarting, the motor where phase angle and zero calibration is completed can run correctly.

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 [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 Protection 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.		

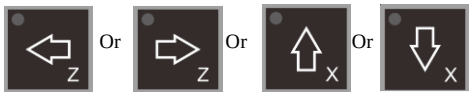
SN	Operation steps	Key	Description
1	Press 【Jog】 or 【Handle】		● Set effective working mode
2	Press [Overtravel release] and [Axis feed]		● Press [Overtravel release] and [Axis feed] simultaneously ● Select [Axis feed] in the reverse direction of overtravel axis

Note:

- Under jog (handle) 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 Overtravel Release

Operation name	Overtravel release	Working mode	Jog, handle
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」		● Remove 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:

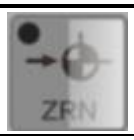
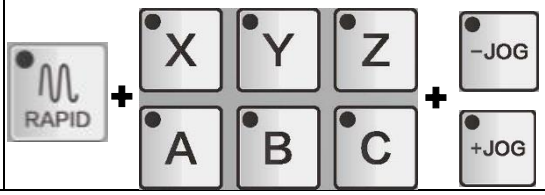
- Press **"Emergency stop"** to reduce electric shock of device before power-on and power-off.

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 【ZRN】		● Set effective working mode
2	Select the axis and press [axis feed]		● [Axis feed] key* in the specified stipulated direction

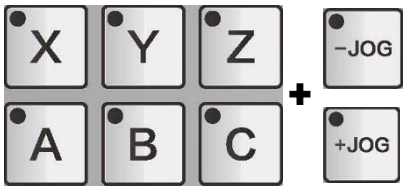
Note:

- 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	Jog
Basic requirements	(1) The need for continuous movement of machine tool		

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

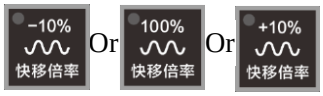
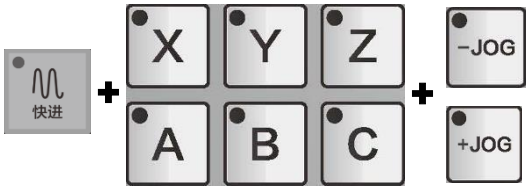
Note:

- 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	Select an axis, 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 Handle

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

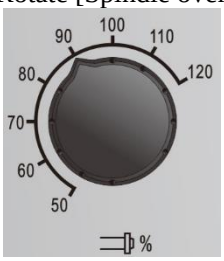
Operation name	Move the coordinate axis by handwheel	Working mode	Handle
Basic requirements	(1) The need for continuous accurate movement of machine tool		

SN	Operation steps	Key	Description
1	Press 【Handle】		● 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

SN	Operation name	Start operation	Stop operation	Description	Effective working mode
1	Spindle rotation CW	Press [SPD CW] key	Press [Spindle stop] or [Reset] key  	1. [SPD CW], [SPD 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.	Handle, incremental, jog
2	Spindle rotation CCW	Press [SPD CCW] key 	Press [Spindle stop] or [Reset]		
3	Spindle stop	Press [Spindle stop]	Press [Reset]		
6	Spindle speed magnification	Rotate [Spindle override] knob 		1. Override range: 50%-120%.	Handle, incremental, jog, auto, MDI

5.6 Other Manual Operations

SN	Operation name	Start operation	Stop operation	Description	Effective working mode
1	Light ON/OFF	Press [Machine lamp] 	Stop after the next tool is changed	Press this key to turn on or off the machine light, the default is OFF	Jog, handle, increment, auto, MDI, single block
2	Protective door	Press [Protec door] 	Press this key again	Press this key to open or close protective door. The default is closing	
3	Cooling	Press [Cooling] 	Press this key again	To enable or disable the cooling pump. The default is disabling. Cannot control it by M8 or M9	
4	Spindle orientation	Press [SPD orien] 			
5	Spindle jog	Press [SPD jog] 			

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).

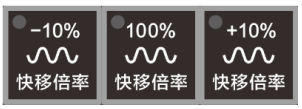
Note:

- 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.

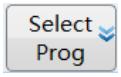
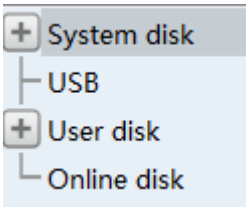
6 Program Edit and Management

6.1 Program Search

HNC-848 system has the program search function under "MACH" function set and "PROG" function set, but usage of programs found under the two function sets is different. Programs found under "MACH" function set are used for machining program loading, program backstage editing and other operations; while programs searched under "PROG" function set are used for program management including program copy, paste and delete and program transmission among different disks,etc.

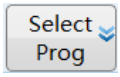
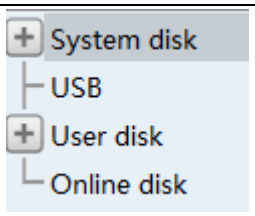
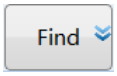
6.1.1 Machining or Editing Program Search

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 〖MACH〗		● Default interface, main menu	
2	Press 『Select prog』		● "Select program" sub-interface and menu	
3	Press 『System disk』 or 『USB』, etc		● Select system disk, USB flash disk, network disk and user disk	
4	Press 「Cursor」 or 「PgUp/PgDn」		● Complete the search and move the cursor to the program name to be searched ● The found programs can be used for two purposes	
5.1	Press 「Enter」		● The found programs are used for loading machining program and running	

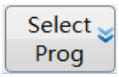
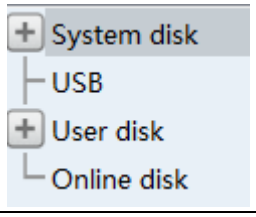
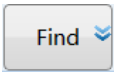
5.2	Press 『Background』		● The found programs are used for entering program editing status

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 〖MACH〗		● Default interface, main menu	
2	Press 『Select prog』		● "Select program" sub-interface, level 2 menu	
3	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk and user disk	
4	Press 『Find』		● Prompt: Enter a file name	
5	(Enter a file name)	---	● e.g.: Onc123	
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 『Background』		● Searched programs are used for entering program editing status	

6.1.1.3 Search programs under the directory 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

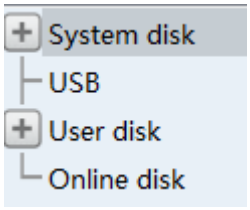
SN	Operation steps	Key	Description
1	Press 〖MACH〗		● Default interface, main menu
2	Press 『Select prog』		● "Select program" sub-interface, level 2 menu
3	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk and user disk
4	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the selected file directory name
5	Press 「Enter」		● Confirm and open the directory
6	Press 『Find』		● Prompt: Enter a file name
7	(Enter a file name)	---	● e.g.: Onc123
8	Press 「Enter」		● 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」		● The found programs are used for loading machining program and running.
9.2	Press 『Background』		● The found programs are used for entering program editing status

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

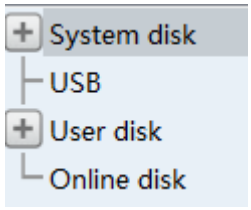
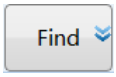
6.1.2.1 Direct search

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 〖Prog〗		● Default interface, main menu

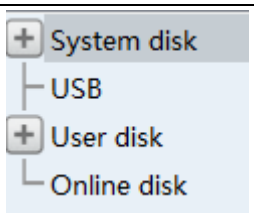
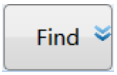
2	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk and user disk
3	Press 「Cursor」 or 「PgUp/PgDn」		● Complete search ● Move the cursor to the program name to be searched

6.1.2.2 Search programs under different disks by "Find" 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 〖Prog〗		● 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 Find what:	
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 "Find" 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 〖Prog〗		● Default interface, main menu
2	Press 『System disk』 or 『USB flash disk』, etc.		● Select system disk, USB flash disk, network disk and user disk
3	「 Cursor 」 or 「PgUp/PgDn」		● Move the cursor to the selected file directory name
4	Press 「Enter」		● Confirm and open the directory
5	Press 『Find』		● Prompt: Enter a file name
6	(Enter a file name)	---	● e.g.: Onc123 
7	Press 「Enter」		● Complete search ● Move the cursor to the program name to be searched

6.2 Program Edit

This system enters program editing status in 4 ways. 1. " New " under "Mach" function set; 2. "Edit prog" under "Mach" function set; 3. "Background" under "Mach" function set; 4. "New " under "Prog" function set.

6.2.1 Create New Programs

The "New " function can be found both under "Mach" function set and "Prog" 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 "MACH" function set

1) Create new programs under "Edit program" sub-interface

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

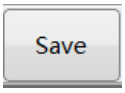
Basic requirements	The new program name cannot be the same as existing program names	Display interface	3.2.3 "Edit program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖MACH〗		● Default interface, main menu
2	Press 『Edit prog』		● 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 "Background" sub-interface

Operation name	Create new programs under "Mach" 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 〖MACH〗		● Default interface, main menu
2	Select 『Select prog』		● "Select program" sub-interface, level 2 menu
	Press 『Background』		● 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 "Prog" function set

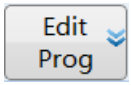
Operation name	New programs created under "Prog" function set		Working mode	Auto, single block
Basic requirements			Display interface	3.4 "Program" function set interface
SN	Operation steps	Key	Description	
1	Press 〖PROG〗		● Default interface, main menu	
2	Press 『New』		● A prompt message "Please enter 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 "MACH" 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

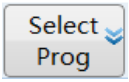
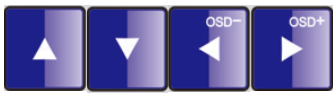
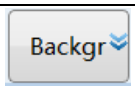
SN	Operation steps	Key	Description
1	Press 〖MACH〗		● Default interface, main menu
2	Press 『Edit prog』		● Enter current loading program editing status
3	(Edit program)	---	● Edit and modify existing loading programs
4	Press 『Save』		● Program has been saved

Note:

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

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

Operation name	Editing and modification of non-loading program in the background	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 〖MACH〗		● Default interface, main menu
2	Press 『Select prog』		● "Select program" sub-interface
3	Press 「Cursor」		● Select current programs to be edited and modified
4	Press 『Background』		● Enter program editing status
5	(Edit program)	---	● Edit and modify existing loading programs

6	Press 『Save』		● Program has been saved

Note:

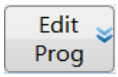
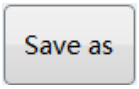
- When current loading program is selected in the "Select program" sub-interface, the current loading program also can be edited through "Background" function.

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 "Background" sub-interface in order to realize the saving.

6.2.3.1 Save “Current loading program” as

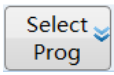
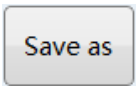
Operation name	Save "Current loading program" as	Working mode	Auto, single block, jog
Basic requirements	The saved file name cannot be the same as the existing program names	Display interface	3.2.3 "Edit program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖MACH〗		● Default interface, main menu
2	Press 『Edit prog』		● Enter current loading program editing status
3	Press 『Save as』		● Storage target selection dialog box ● "System disk, USB 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

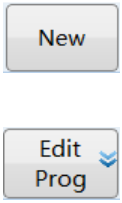
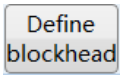
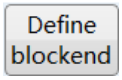
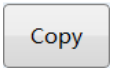
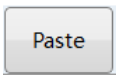
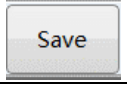
Operation name	Save "Non-loading program" as	Working mode	Auto, single block, jog
Basic requirements	The saved file name cannot be the same as the existing program names	Display interface	3.2.2 "Select program" sub-interface

SN	Operation steps	Key	Description
1	Press 〖MACH〗		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface
3	Press 「Cursor」		● Select the current program to be saved
4	Press 『Background』		● Enter program editing status
5	Press 『Save as』		● Storage target selection dialog box ● "System disk, USB, 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」		● 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

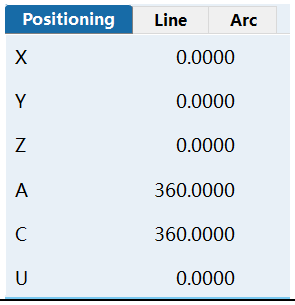
Operation name	Copy and paste of program block	Working mode	Auto, single block, jog

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		● 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

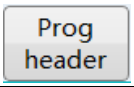
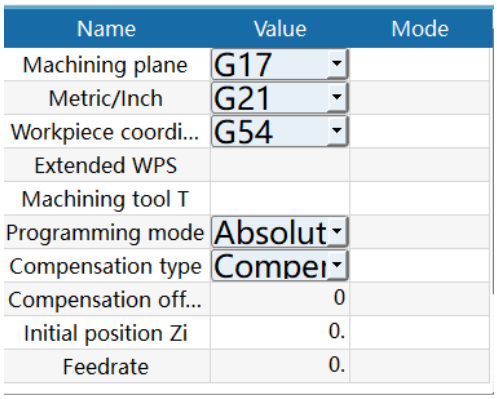
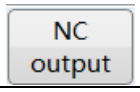
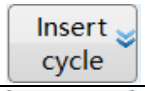
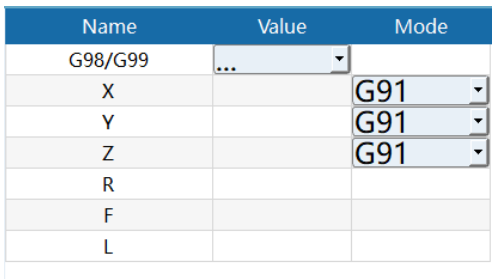
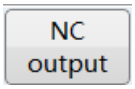
6.2.5 Programming Teach

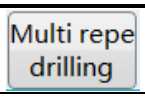
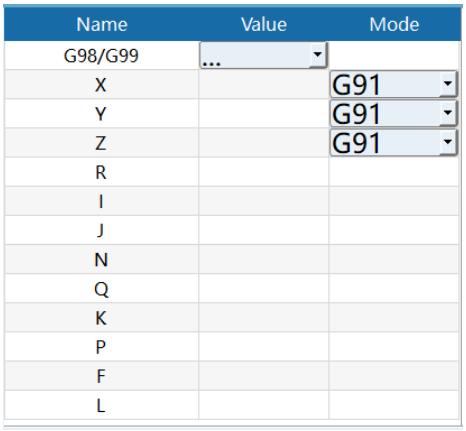
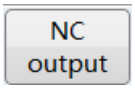
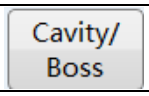
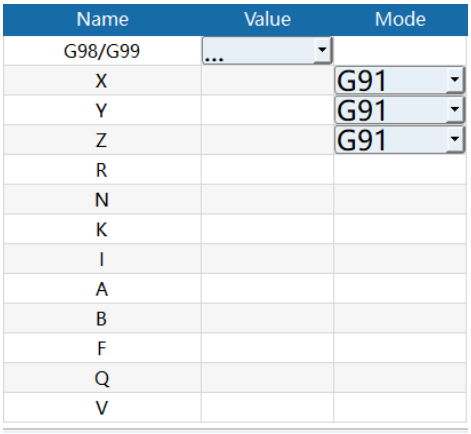
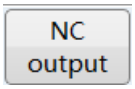
Operation name	Programming teach	Working mode	Auto, single block, jog
Basic requirements	Enter program teaching status	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description
1	Edit program	Pgrm teach	● Enter “Pgrm” under “Edit prog”
2	Insert	Insert	● Enter “Insert” sub-interface, current machine coordinate system is recorded
3	Rapid positioning	Rapid position	● Move the cursor to X/Y/Z/A/C input box ● Enter X/Y/Z/A/C coordinate value 
4	Insert	Insert	Current machine coordinates X/Y/Z are recorded
5	Linear interpolation	Linear interp	● Move the cursor to X/Y/Z/A/C input box ● Enter X/Y/Z/A/C coordinate values ● Enter F feed value
6	Insert	Insert	Current machine coordinates X/Y/Z/A/C are recorded
7	3-point circle mode	Spatial circle	● Move the cursor to X/Y/Z input box ● Enter X/Y/Z coordinate values ● Enter F feed value
8	Insert	Insert	Current machine coordinates X/Y/Z are recorded
9	Radius circle mode	Circle radius	● Move the cursor to X/Y/Z input box, and select direction and plane ● Enter X/Y/Z coordinate values of start point, radius and end point ● Enter F feed value
10	Insert	Insert	Current machine coordinates X/Y/Z are recorded
11	Feedrate	Feedrate	Modify feedrate in current mode
12	Delete line	Delete line	Delete a line of codes
13	Switch window	Switch window	Switch local window of system

6.2.6 Canned Cycle Guide

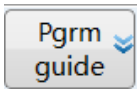
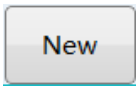
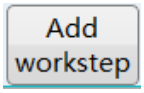
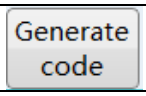
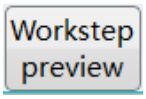
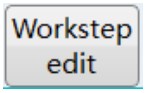
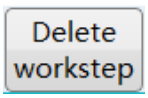
Operation name	Programming guide	Working mode	Auto, single block, jog
Basic requirements	Enter “Insert cycle”	Display interface	3.2 "Machining" function set interface

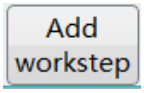
SN	Operation steps	Key	Description
1	Edit program		● Enter “Insert cycle” under “Edit prog”
2	Program header		● Enter “Program header” sub-interface
3			● Fill in item by item
4	NC output		Click “NC output” to output corresponding codes and go to the “edit program” interface
5	Insert cycle		● Enter drilling cycle interface ● Enter “Insert cycle” under “Edit program”
6	Drilling		Enter “Drilling” sub-interface
7			● Fill in item by item
8	NC output		Click “NC output” to output corresponding codes and go to the “edit program” interface
9	Insert cycle		Enter insert cycle interface

10	Multiple repetitive drilling		Enter multiple repetitive cycle interface
11			Fill in item by item
12	NC output		Click “NC output” to output corresponding codes and go to the “edit program” interface
13	Insert cycle		Enter insert cycle interface
14	Cavity/Boss milling		Enter cavity/boss sub-interface
15			Fill in item by item
16	NC output		Click “NC output” to output corresponding codes and go to the “edit program” interface

6.3 Programming Guide

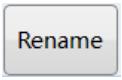
Operation name	Programming guide	Working mode	Auto, single block, jog
Basic requirements	“Machining” interface	Display interface	3.2 "Machining" function set interface

SN	Operation steps	Key	Description
1	Cursor		● Page turning of menu
2	Programming guide		● Enter “Programming guide” sub-interface
3	New		● Create a new workstep file
4	Load		● Load newly-create workstep file, and enter the workstep edit inteface
5	Add workstep		● After selecting drilling, plane, milling, pocket, cavity, press confirm
6	Center drill		● Enter “Center drill” sub-interface, and fill item by item
7	Generate G code		
8	Workstep preview		● View edited G codes
9	Edit workstep		● Edit and modify the added workstep parameter
10	Delete workstep		● Can delete the workstep not required

11	Add workstep		Continue adding

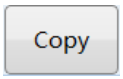
6.4 Program Management

6.4.1 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 〖PROG〗		● 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』		● Prompt: Enter a new file name	
5	(Rename a file)	---	● Enter a new file name	
6	Press 「Enter」		● Confirm the new file name ● Prompt: Rename the old file as a new file	

6.4.2 Copy and Paste File Directory and Program

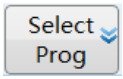
Operation name	Copy and paste file directory and program		Working mode	Auto, single block, jog
Basic requirements	Existing programs can be searched		Display interface	3.4.2.2 "Program copy and paste" sub-interface
SN	Operation steps	Key	Description	
1	Press 〖PROG〗		● Default interface, main menu	

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』		● 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』		● Prompt: Pasted

6.4.3 Program Deletion

6.4.3.1 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 〖MACH〗		● 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: "Confirm to delete the selected file? (Y/N)"
5	Press 「Y」 or 「N」		● Press 「Y」 to complete deletion ● Press 「N」 to abandon deletion

6.4.3.2 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 〖PROG〗		● 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

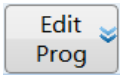
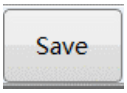
7.1 Auto Operation

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

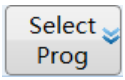
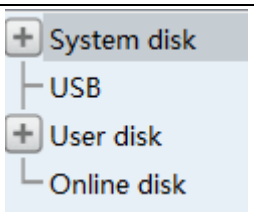
SN	Operation steps	Key	Description
1	Press 【Auto】		● Maintain the original interface
2	Press 〖MACH〗		● Default interface, main menu
3	Press 『Edit program』		● The cursor enters the loaded program editing area
4	Press 『New』		
5	(Enter a file name)	---	● Enter a new file name, such as “nc123” ● Address 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』		● The newly edited program is loaded as the machining program immediately ● A prompt message File 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 /online 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 〖MACH〗		● 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.

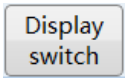
7.1.3 Program Verify

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

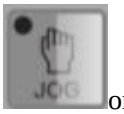
			
4	(Loading program)	---	● Load machining program as per 7.1.1
5	Press [Cycle start]		● Press Cycle start once to execute a block of program, and cycle in turn

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

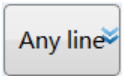
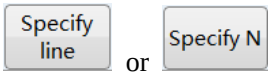
SN	Operation steps	Key	Description
1	Press 【 Jog 】 or 【Handle】	 or 	● Block skip operation function is set only under jog, handwheel or incremental mode
2	Press [Block skip]		● Program blocks with the skip symbol will be skipped
3	Press 【Auto】		● Maintain the original interface
4	Press 〖MACH〗		● Default 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 〖MACH〗		● 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』		● The indicator light lights off ● Operation is suspended
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 〖MACH〗		● 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 〖MACH〗		● 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 turn off MST manually	
6	(Disable M and S functions)	---	● Disable MST manually	
7	Press [Emergency stop]		● Terminate the operation ● Reset	

7.3 MDI Operation

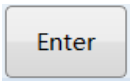
"MDI" function has two forms,

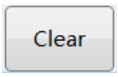
- MDI key is the working mode key【MDI】 of MCP panel
- MDI key is the function set key 〖MDI〗 of NC panel

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 『Enter』		● 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 『Enter』		● Repeat steps 3 and 4
5.2.2	Press [Cycle start]		---
5.3 To suspend program running			
5.3.1	Press Dwell		Machine stops running and keeps feed hold, and press Cycle Start to continue subsequent program
5.4 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
5.5 To save MDI program in the system disk			
5.5.1	Press 『Save as』		● 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 is saved and a prompt message will be given
5.6 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

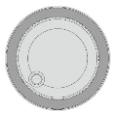
Note:

- 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

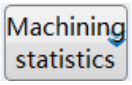
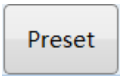
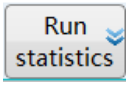
This function controls the machine tool axis to run as per the programmed path through the MPG. It is often used to check whether tool setting is correct when the tool approaches the workpiece in order to avoid damaging the workpiece. This function is valid under automatic mode or single block mode.

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 in auto mode
2	Press [Handle simulation]		● If this function is valid, the indicator light lights up
3	Press [Cycle start]		● The commands run normally before the machine axis moves, such as the spindle rotation CW, and the machine axis does not move at this moment
4	(Rotate the handle)		● If the handwheel rotates clockwise, the machine tool moves forward with the program ● If the handwheel rotates counterclockwise, the machine 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 〖MACH〗		● 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 『Run statistics』		● Export or eliminate machining information file

8 Tool Setting and Machining Setting

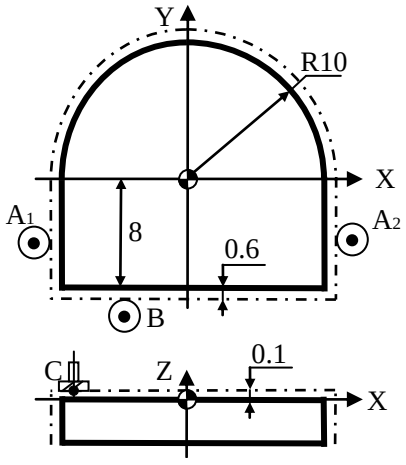
Manual tool setting is completed mainly in "Coordinate system" and "Tool compensation" sub-interfaces. For ease of user operation, the system sets two sub-interfaces in both "Machining" and "Set" function sets to reduce interface switching. The operating steps for both are basically the same.

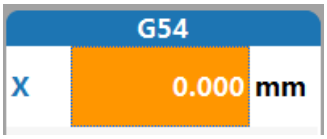
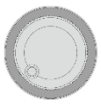
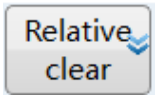
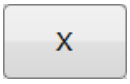
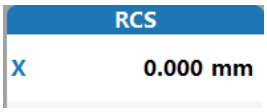
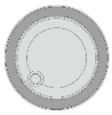
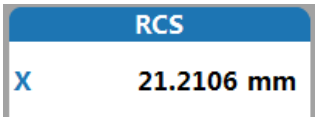
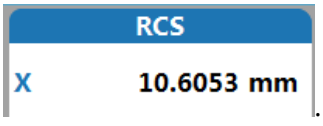
For some specific parts, the system can simplify manual tool setting through "Workpiece measurement" function in "Set" function set. This function includes "Center measurement", "Plane measurement" and "Circle center measurement" and is mainly used to set coordinates of X, Y and Z axes automatically.

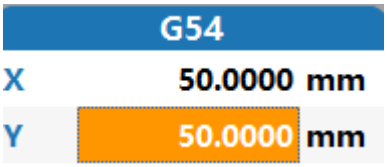
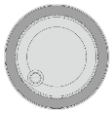
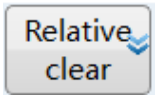
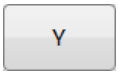
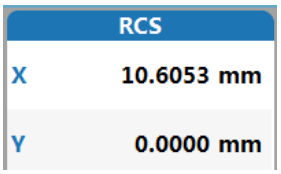
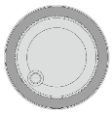
With the popularity of CNC machine tools, the application of tool setter has become more widespread, and automatic tool setting has been increasingly used. The system is configured with "Auto tool setting" function in "Set" function set, which can be used for automatic setup of Z axis coordinates and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

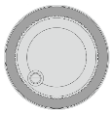
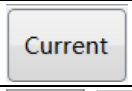
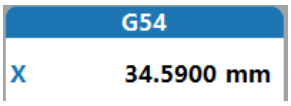
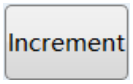
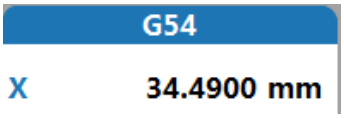
8.1 Manual Tool Setting

Manual tool setting is taken as an example under "Set" function set in this section.

<table>
<tr><td>Operation name</td><td>Manual tool setting</td><td>Working mode</td><td>Jog, handle</td></tr>
<tr><td>Display interface</td><td>3.2 "Coordinate system" and "Tool compensation" sub-interfaces under "Set" function set</td><td colspan="2" rowspan="2">零件对刀示意图
 
</td></tr>
<tr><td>Basic requirements</td><td>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. A1 and A2 are the touch points that the tool just touches the workpiece when X tool setting;
6. When B is the touch point that the tools just touches the workpiece when Y tool setting;
7. When C is the touch point that the tool just touches the workpiece when Z tool setting.</td></tr>
</table>

SN	Operation steps	Key	Description
1	Press 〖SET〗		● Enter the "Set" 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 A2 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 10.6053 point of "Relative actual" 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』 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 handle		● 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』		● 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		● Select Z axis coordinates of G54 using cursor as below:

	setting axis (Z axis)		
19	Move the tool to C point with handle		● The tool just touches the upper surface of the workpiece (precutting mode)
20	Press 『Current』		● Record the machine coordinate value on Z axis of current tool in the coordinate system .
21	Press 「Cursor」		● Return to Z axis coordinates 
22	Press 『Increment』		● 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	--	

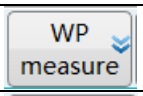
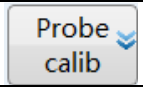
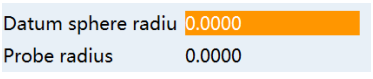
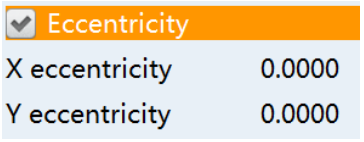
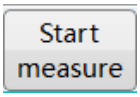
8.2 Workpiece Measurement

"Set" function set supports the workpiece measurement function. This function supports probe calibration, single-point measurement, bevel measurement, plane measurement, rectangle measurement, circle measurement, abnormity measurement, and center measurement.

8.2.1 Probe Calibration

The function is to calibrate the probe eccentricity of measurement gauge. After running calibration measurement automatically, probe eccentricity of measurement gauge can be obtained.

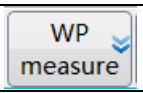
Operation name	Probe calibration	Working mode	Jog, handle
Basic requirements	The system is allowed to run under jog mode and handle mode	Display interface	See Chapter 3 "Workpiece measurement" interface

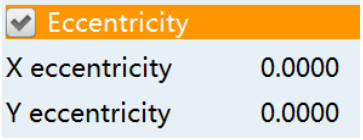
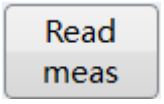
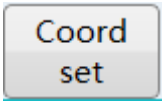
SN	Operation steps	Key	Description
1	Press 〖SET〗		● Enter the main interface of "Set" function set.
2	Press 『Workpiece measurement』		● Enter the "Workpiece measurement" function interface.
3	Press 『Probe calibration』		● Switch to the probe calibration interface.
4	Datum sphere radius and probe radius		● Enter standard cylinder diameter ● Enter probe radius
5	Activate eccentricity		● Select the eccentricity to activate it. Fill the offset amount based on actual situation. No need to fill if there is no offset
6	Start measure		● Read the measured value
7	Coordinate setting		● System calculates measurement result, and assign the value to the selected coordinate system

8.2.2 Single Point Measurement

The measurement is to set coordinates of the tool in the current machine coordinate system as the zero of workpiece coordinate system.

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

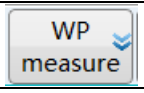
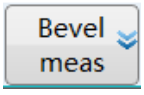
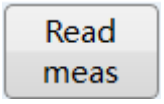
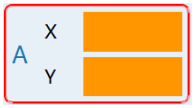
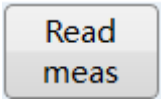
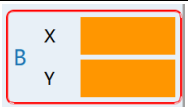
SN	Operation steps	Key	Description
1	Press 〖SET〗		● Enter the main interface of "Set" function set.
2	Press 『Workpiece measurement』		● Enter the "Workpiece measurement" function interface.

3	Press 『Single point measurement』		● Switch to the single point measurement interface.
4	Select coordinate plane		● User Curser and Enter to select and confirm circular plane; G17 G18 G19 ● Display of A, B, C coordinates varies with the set plane
5	Activate eccentricity		● Select the eccentricity to activate it. Fill the offset amount based on actual situation. No need to fill if there is no offset
6	Workpiece coordinates X/Y		● Read the measured value
7	Coordinate setting		● System calculates measurement result, and assign the value to the selected coordinate system

8.2.3 Bevel Measurement

The measurement is to read two coordinate points of current bevel, and calculate angle of the bevel through the two points.

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

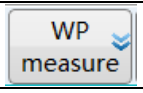
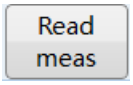
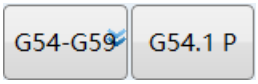
SN	Operation steps	Key	Description
1	Press 〖SET〗		● Enter the main interface of "Set" function set.
2	Press 『Workpiece measurement』		● Enter the "Workpiece measurement" function interface.
3	Press 『Bevel measurement』		● Switch to the bevel measurement interface.
4	Take A coordinates of workpiece bevel		
5	Take B coordinates of workpiece bevel		

6			● Get the angle of bevel measurement

8.2.4 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, handle
Basic requirements	The system is allowed to run under jog mode and handwheel mode	Display interface	See Chapter 3 "Workpiece measurement" interface

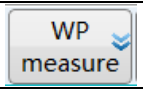
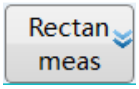
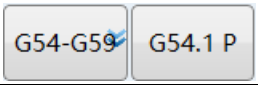
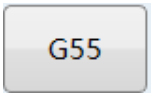
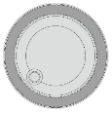
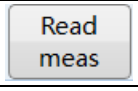
SN	Operation steps	Key	Description
1	Press 〖Setup〗		● Enter the main interface of "Se" function set
2	Press 『Workpiece measurement』		● Enter the default interface of "Workpiece measurement" function
3	Press 『Plane measurement』		● Switch to the plane measurement function interface
4	Take X workpiece coordinate		● Input the cursor on X axis coordinates ● Move the tool to a point on the edge of X axis of the workpiece in handle mode, and acquire coordinates of X axis based on the measured value
5	Take Y workpiece coordinate		● Input the cursor on Y axis coordinates ● Move the tool to a point on the edge of Y axis of the workpiece in handle mode and acquire coordinates of Y axis based on the measured value
6	Take Z workpiece coordinate		● Input the cursor on Z axis coordinates ● Move the tool to a point on the edge of Z axis of the workpiece in handle 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

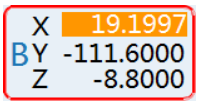
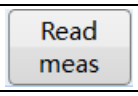
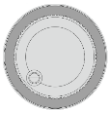
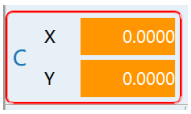
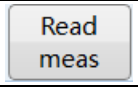
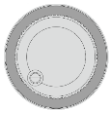
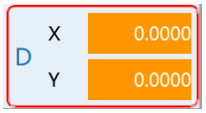
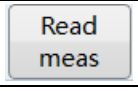
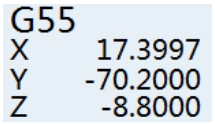
			system

8.2.5 Rectangle Measurement

The measurement is to read machine coordinates when tool contact both ends of workpiece on an axis, and calculate workpiece zero.

Operation name	Plane measurement	Working mode	Jog, handle
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 "Se" function set
2	Press 『Workpiece measurement』		● Enter the default interface of "Workpiece measurement" function
3	Press 『rectangle measurement』		● Switch to the rectangle measurement function interface
4	Select coordinate system		● Select the coordinate system type
5	Press G55		● Select coordinate system
6	Press cursor		● Select tool setting point A
7	Press cursor		● Select X coordinate axis
8	Move tool to the left side of workpiece by handle		● Tool just touches left edge of blank (precutting mode) ● Select A point on left side of workpiece 
9	Read measurement value		● Cursor goes to B automatically on X
10	Move tool to the right side of workpiece by handle		● Tool just touches right side of blank; ● Precutting position error cannot be larger than allowance

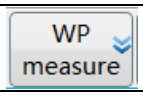
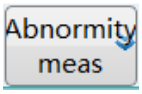
			
11	Read measurement value		● Cursor returns to A automatically
12	Press cursor		● Select Y coordinate axis
13	Move to the rear side of workpiece by handle		● Tool just touches back edge of blank (precutting mode); ● Select A point on back side of workpiece 
14	Read measurement value		● Cursor goes to B automatically on Y
15	Move tool to the front side of workpiece		● Tool just touches front side of blank; ● Precutting position error cannot be larger than allowance 
16	Read measurement value		● Cursor returns to A automatically
17	Press 『Coordinate setup』		● The system calculates measurement results and assign them to the selected coordinate system 

8.2.6 Abnormity Measurement

In the measurement mode, system determines circle center by three points, and set the center as the zero of workpiece coordinate system. (In the description, A, B, C represent the three points on the arc)

Operation name	Abnormity measurement	Working mode	Jog, handle
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

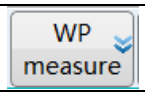
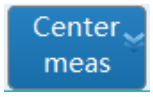
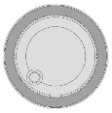
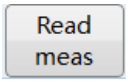
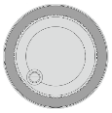
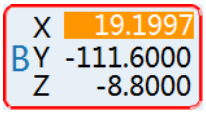
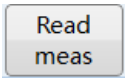
1	Press 〖Setup〗		● Enter the main interface of "Se" function set
2	Press 『 Workpiece measurement』		● Enter the default interface of "Workpiece measurement" function
3	Press 『 Abnormity measurement』		● Switch to abnormity measurement function interface
4	Select plane		● User Curser and Enter to select and confirm circular plane;  ● Display of A, B, C coordinates varies with the set plane
5	Move tool to A		● Manually move tool to point A (tool touches workpiece arc) ● A, B, C can be any different three points on the arc. Uniform distribution is recommended.
6	:Press cursor		● Select point A to set and display
7	Read measurement value		● Read machine coordinate value of A
8	Read B, C coordinates		● Repeat steps 5, 6, 7
9	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

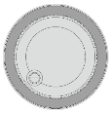
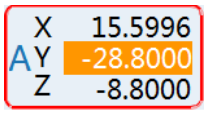
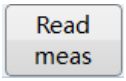
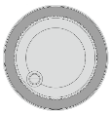
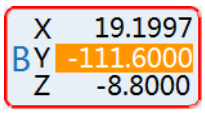
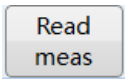
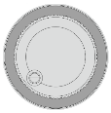
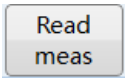
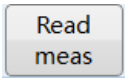
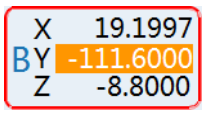
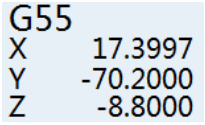
8.2.7 Center Measurement

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.

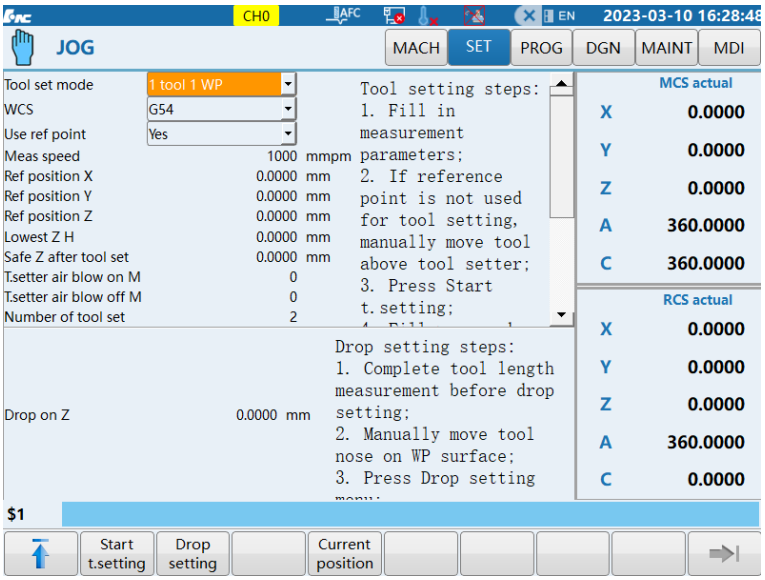
Operation name	Center measurement	Working mode	Jog, handle
Basic requirements	The system is allowed to run under jog mode and handwheel mode	Display interface	See Chapter 3 "Workpiece measurement" interface

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. 
14	Press 『Read measurement』		● 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. 
16	Press 『Read measurement』		● 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』		● The cursor skips to B point automatically on (Z) axis; ● Keep the tool position unchanged and execute subsequent operations. 
20	Press 『Read measurement』		● The cursor returns to A point automatically and the coordinate axis remains unchanged 
21	Press 『Coordinate setup』		● The system calculates measurement results and assign them to the selected coordinate system. 

8.3 Automatic Tool Setting

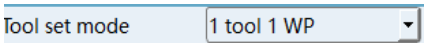
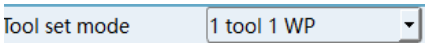
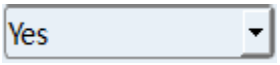
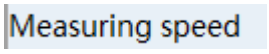
"Auto tool setting" submenu under "Set" function set can measure tool length automatically. The "Auto tool setting" function includes single tool single workpiece measurement mode, single tool multiple workpiece measurement mode and multiple tools multiple workpiece measurement mode. The function interface 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.

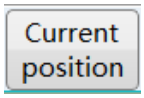
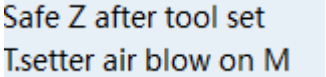
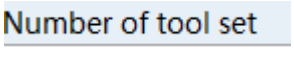


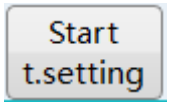
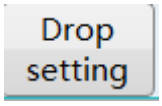
8.3.1 Single-Tool Single-Workpiece Measurement

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

SN	Operation steps	Key	Description
I. Preparation for tool setting			
1	Press 〖Set〗		● Enter the default interface of "Set" function set and.
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 "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 of 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	Handle or Jog 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.

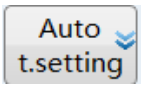
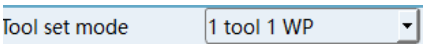
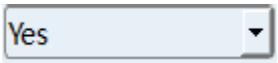
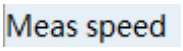
11	Press 「Cursor」		● Move the cursor to "Reference point coordinates in X direction"
12	Press 『Current position』		● The system sets X axis position of current tool as the X axis position of the tool setter in the machine coordinate system; ● Then, the positions of Y and Z axes remain unchanged; ● The 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 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 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 commands of tool setter air blowing on and air blowing off		● Set if the tool setter has air blowing function; otherwise, 0 is set by default.
19	Number of tool setting		● Refer to the number of slow-speed tool setting. The collision of the tool with the tool setter at the first rapid drop is not counted.
V. Automatic tool setting and drop setting			

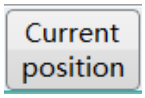
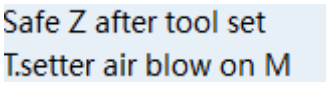
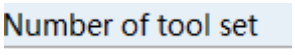
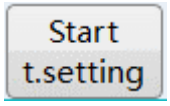
20	Press 『Start tool setting』		● Start tool setting; ● After tool setting, save the measured value in the workpiece coordinate system; ● If an alarm is given in the process of measurement, the measurement stops, and the measurement is performed again after check and debugging.
21	Move the tool nose to the position where Z axis of the workpiece coordinate system is 0 in handle mode	--	● "Drop setting" 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 setting』		● Drop is set when no machine tool runs. After calculation, drop of Z axis is set in the external zero offset coordinates; ● If the tool cannot reach the workpiece coordinates Z0, 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 set external zero offset.

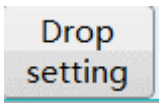
8.3.2 Single-Tool Multiple-Workpiece Measurement

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

SN	Operation steps	Key	Description
I. Preparation for tool setting			
1	Press 〖Set〗		● Enter the default interface of "Set" function set and level 1 menu.

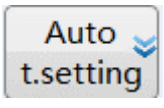
2	Press 『Auto tool setting』		● Enter the "Auto tool setting" function interface.
3	Press 「Cursor」		● Move the cursor to "Tool setting mode" column. 
4	Press 「Enter」		● Activate "Tool setting mode" drop-down box.
5	Select "Single tool multiple workpiece"		● Select "Single tool multiple workpiece".
6	Press 「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	Handle or Jog 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";

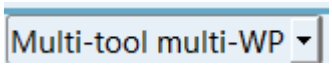
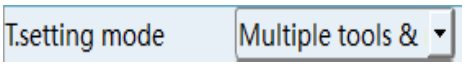
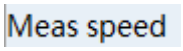
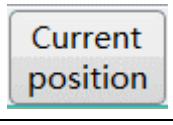
11	Press 『Current position』		● The system sets X axis position of current tool as the X axis position of the tool setter in the machine tool coordinate system; ● Then, the position of Y and Z axes remains unchanged; ● The value also can be set manually as per steps 3, 4 and 6.
12	Set reference point coordinates in Y and Z directions	--	● Repeat steps 10 and 11 respectively
III. Safe position presetting			
13	Press 「Cursor」		● Move the cursor to "Machine coordinates H of the lowest point on Z axis"
14	Input machine 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 Z safe point after tool setting	--	● 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 commands of tool setter air blowing on and air blowing off		● Set if the tool setter has blowing function; otherwise, it is 0 by default.
18	Input tool setting times		● Refer to the number of slow-speed tool setting. The collision of the tool with the tool setter at the first rapid drop is not counted.
V. Automatic tool setting and drop setting			
19	Press 『Start tool setting』		● Start tool setting; ● After tool setting, save the measured value in the external zero offset Z; ● If an alarm is given in the process of

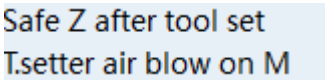
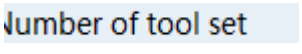
			measurement, the measurement stops, and the measurement is performed again after check and debugging.
20	Select workpiece coordinate system		● Select the set workpiece coordinates of the tool; ● Select and conform as per the same steps as 3, 4 and 6.
21	Move the tool nose to the position where Z axis of the workpiece coordinate system is 0 in handle mode	--	● "Drop setting" is to set the drop between the upper surface of tool setter and zero position of workpiece coordinate Z; ● 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 setting』		● 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.

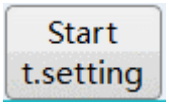
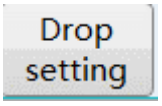
8.3.3 Multiple Tools Multiple Workpiece Measurement

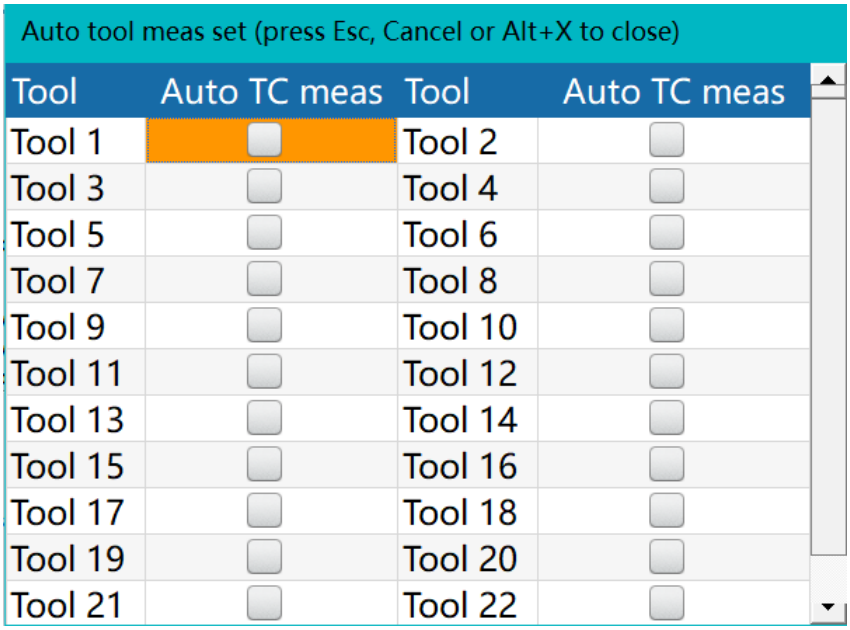
Operation name	Multiple tools multiple workpiece measurement	Working mode	Jog, handwheelle
Basic requirements	The machine tool is equipped with the environment of tool setter test	Display interface	"Auto tool setting" sub-interface

SN	Operation steps	Key	Description
I. Preparation for tool setting			
1	Press 〖Setup〗		● Enter the default interface of “Set”
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 point		● When the position of tool setter has never been set, select the reference 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		● 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	Handle or jog 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 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 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 coordinates H of the lowest point on Z " column.
14	Input machine coordinates H of the lowest point on Z	---	● 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 Z safe point after tool setting	--	● 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 of tool setter air blowing on and blowing off		● Set if the tool setter has blowing function; otherwise, it is 0 by default.
18	Input number of tool setting		● Refer to the number of slow-speed tool setting. The collision of the tool with the tool setter at the first rapid drop is not counted.
V. Automatic tool setting and drop setting			
19	Set tool number for automatic tool setting		● 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』		● 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		● 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 workpiece coordinate system is 0 in handle mode	--	● "Drop setting" is to set the drop between the upper surface of tool setter and zero position of workpiece coordinate Z; ● This operation requires that the tool reaches Z0 of workpiece coordinates accurately; ● If the tool cannot reach Z0 of workpiece coordinates, treat it after "Drop setting".
23	Press 『Drop setting』		● 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.

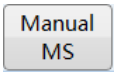


8.4 F/S machining Setting

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 spindle speed.

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 〖MACH〗		● Switch to the machining interface
2	Press 『Manual MS』		● F and S setting menus pop up
3	(Set machining F and S values)		● If there is no F or S in machining program, this value prevails ● Set the default S (speed in jog mode)

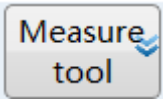
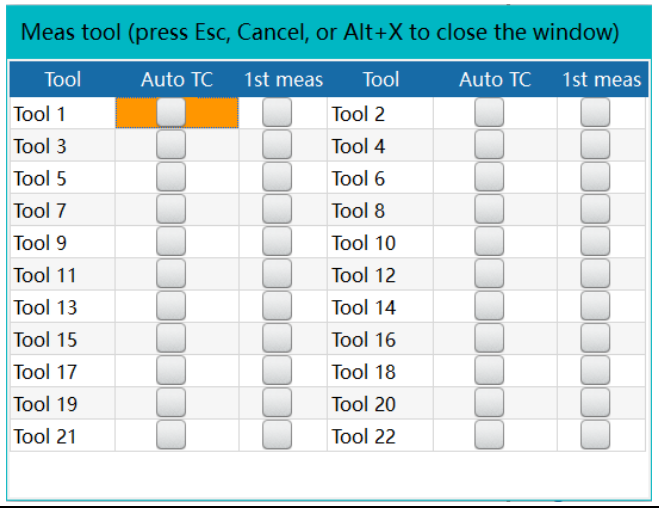
Note:

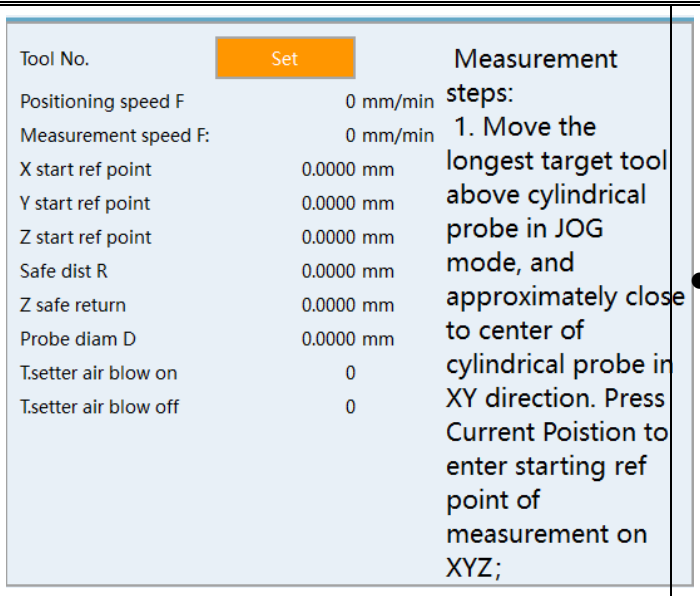
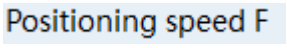
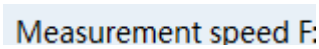
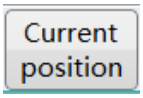
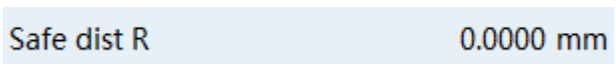
- If there is no F/S value in the program, the value in the status bar can be validated immediately.
- If F/S has been set in the program, manual MS is invalid and data in

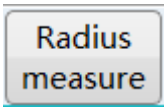
manual MS will be modified by the program.

8.5 Tool Measurement

Operation name	Tool measurement	Working mode	Jog, handle
Basic requirements	The machine tool is equipped with tool setter	Display interface	"Automatic tool setting" sub-interface

SN	Operation steps	Key	Description
I. Preparation for tool setting			
1	Press 〖Set〗		● Enter the default interface of "Set" function set.
2			● Enter Sub-menu of “Set”
3	Press 『 Measure tool』		● Enter the "Measure tool" function interface.
4	Press Set		● Click Set to enter tool test setting interface
5	--		● To set the tool to be measured. Check the tool which needs auto measurement. If "1st measurement" is checked, the measured tool value will be written into the tool offset; if "1st measurement" is not checked, the measured tool value will be written into the tool wear. A maximum of 32 tools can be selected

6	--		● Refer to the function page..
7	Positioning speed		● Set the positioning speed based on actual situation.
8	Measurement speed		● Select an appropriate measurement speed based on height of reference level Z; ● Set as F50 when tool contacts tool setter;
II. Initial position presetting for tool setting			
9	Move the tool to the preset position of tool setter manually	Handle or Jog key	● Move the tool with the largest length above the cylindrical probe in jog mode, close to the probe center on X and Y
10	Press 「Cursor」		● Move the cursor to "Reference point coordinates on X";
11	Press 『 Current position』		● Press the key to enter starting reference point of the measurement on X, Y, Z
12	Safe distance R		● Move the tool in jog mode to make the tool can be measured by the outer surface. Enter the safe distance R.

13	Press 「Cursor」		● Move the cursor to "Z safe return" Safe dist R 0.0000 mm
14	Input Z safe point after tool setting	--	● Move the tool the safe point on Z after measurement in jog mode, and enter the position;
15	Input the commands of tool setter air blowing on and air blowing off	Safe Z after tool set T.setter air blow on M	● Set if the tool setter has blowing function; otherwise, it is 0 by default.
16	Radius measurement key		● Radius measurement starts

Note: 1. No measuring of too radius in machining;

1. When no tool number is set on tool radius measurement interface, no starting of tool radius measurement;

2. Positioning traverse speed F cannot exceed 2000mm/min;

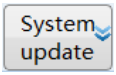
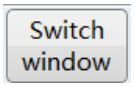
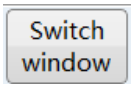
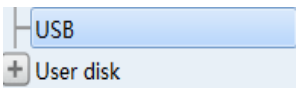
3. No using of tool measurement function in single block mode; otherwise, an alarm is issued.

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 『Switch window』		● 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 『Switch window』		● Switch to the upgrade package file source selection window on the lower part of the interface. 
7	Press 『USB』/『User disk』		● Select the upgrade package file from USB by default ● The upgrade package file in the user disk also can be selected
8	Press 「Cursor」		● Select the upgrade file package ● The upgrade package file name must be of BTF

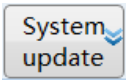
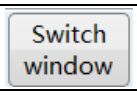
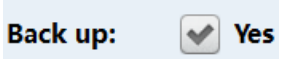
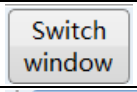
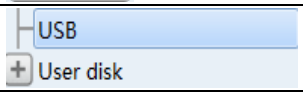
9	Press 「Enter」		● Start system upgrade ● Do not power off before the upgrade is completed

Note

- **The upgrade should be conducted with the permission of system administration, 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 『Switch window』		● Switch to the upgrade package 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 selection
8	Press 『Switch window』		● Switch the cursor to the backup target disk window
7	Press 『USB/ 『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 V2.42.00 software is about 200MB.**

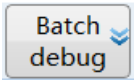
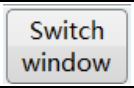
9.2 Batch Commissioning

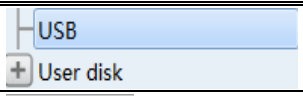
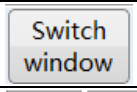
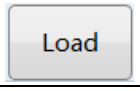
This function is limited by permission. This function can load/back up PLC, canned cycle, parameter, G code, parameter configuration and other files required for commissioning separately/in batches.

Operation mode and object of "Batch commissioning" function are similar to those of "Manage data" function. There are more "Manage data" files and only a single file can be operated.

9.2.1 Batch Load Commissioning

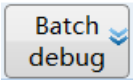
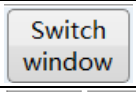
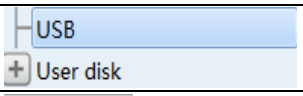
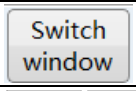
Operation name	Batch load commissioning	Working mode	Emergency stop
Basic requirements	The machine must be at "Emergency stop" state Files loaded in batches must be of .tar	Display interface	3.3 "Maintain" function set interface

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 『Switch window』		● 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/『User disk』		● Select the source disk of the loaded file
7	Press 『 Switch window』		● 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 "Load file XXXX.tar or not?(Y/N)"
10	Press 「 Y 」 or 「Enter」	 or 	● A prompt message "File loaded, power off and reastart!”

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/『User disk』		● Select a target disk to back up files
7	Press 『 Switch window』		● 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 "Enter 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 “Backed up”

Note: Do not power off during backup or loading.

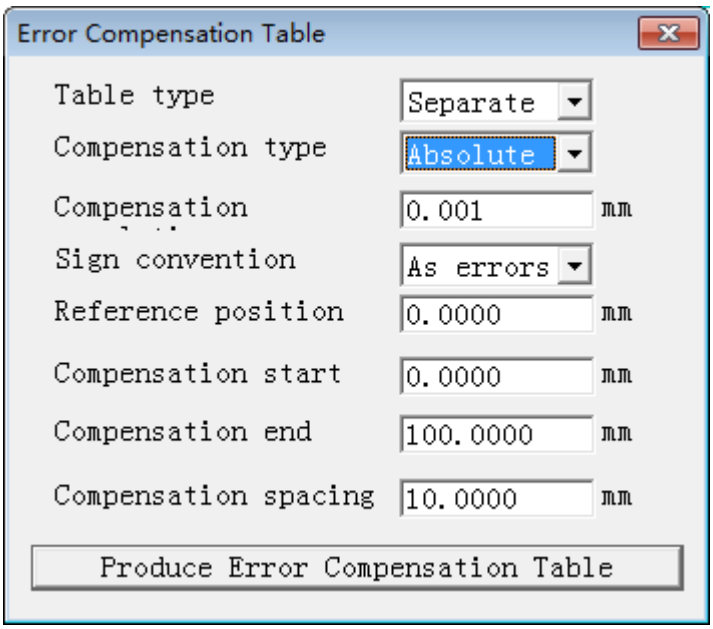
9.3 Pitch Error Compensation

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-848 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).



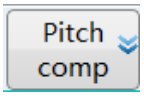
- "Chart type" **must** select "Separate compensation";
- "Compensation type" **must** select "Absolute";

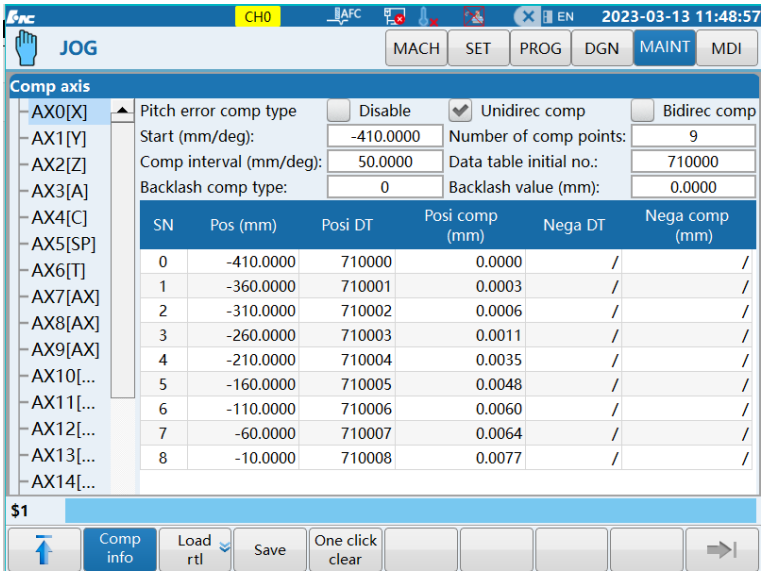
- "Compensation resolution" must be "1"um;
- "Conversion of plus and minus symbols (+/-)" **must** select "Compensation value";
- "Reference point " **must be "0"**;
- The "Compensation start point" is the machine coordinate position of the compensation start point, **it must be 0**;
- The "Compensation 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-848 system can enter this function sub-interface under "Maintain" function sets.

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

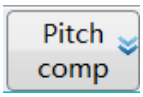
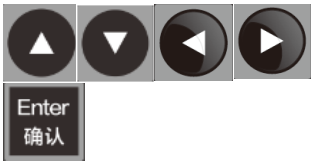
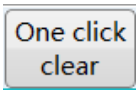
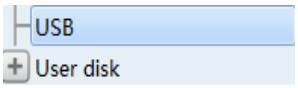
SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● Enter “Maintenance” menu
2	Press 『 Parameter category』		● Enter the "Parameter category" sub-interface
3	Press 『 Pitch compensation』		● Enter the pitch compensation sub-interface (as shown below)



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 pitch error compensation file generated by Renishaw software).

Operation name	Import of pitch error compensation file	Working mode	Auto, single block, jog, increment
Basic requirements	Pitch error compensation data file has been generated correctly	Display interface	3.2 "Tool compensation " sub-interface

SN	Operation steps	Key	Description
1	Press 〖Maintain〗		● "Maintain" main menu
2	Press 『 Parameter category』		● "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 ● 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 pitch 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
8	Press 『USB』/『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 pitch compensation data)		● Check pitch error compensation type, start point, number of compensation points, 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: Options on the interface

- Selection of compensation axis: Axis 0, axis 1 and axis 2.....;
- Pitch error compensation type: Disable, unidirectional compensation, bidirectional compensation;
- Start point : The same as "Renishaw error compensation table";
- Compensation interval: The same as "Renishaw error compensation table";
- Number of 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.