

HNC-8 System Operating Manual (Lathe)

V2.4 Series

Introduction

The manual may help you to quickly get familiar with the HNC-818 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 "HNC"). Without HNC's authorization or written permission, any units or individuals are not allowed to modify or correct the manual. HNC will not be responsible for any losses thus incurred to customers.






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

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

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



Attention

-  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	i
1 Overview	1
1.1 Basic Operation	1
1.2 Basic function	2
1.3 Basic display interface	3
1.3.1 Machining display interface	3
1.3.2 Program selection and editing interface	5
1.3.3 Machining setup interface	6
1.3.4 Parameter setup interface	7
2 Operating equipment	8
2.1 System host panel (NC panel)	8
2.1.1 System host panel zoning	8
2.1.2 Display interface zoning	9
2.1.3 Definition of host panel keys	10
2.2 Operation panel of machine tool (MCP panel)	13
2.2.1 Operation panel zoning of machine tool	13
2.2.2 Definition of operation panel of machine tool	14
3 Display interface	17
3.1 Display interface selection and menu structure	17
3.1.1 Common operation of interface and menu selection	17
3.1.2 Function menu structure	18
3.2 Display interface and basic operation of "Machining" function set	28
3.2.1 Interface and function of "Machining" function set	28
3.2.1.1 Machining set interface zoning	29
3.2.1.2 Display switch of graphics and G code area	30
3.2.1.3 "Big character coordinate" display setup of coordinate graphics display area ..	30
3.2.1.4 "Joint coordinate" display setup of coordinate graphics display area	30
3.2.1.5 Display switch of machining and commissioning information area	30
3.2.1.6 Display switch of machining information area	31
3.2.2 "Select program" sub-interface	31
3.2.2.1 Select a program in the USB flash disk and load it as current machining program	31
3.2.2.2 Select a program in the directory as current machining program	32
3.2.2.3 Exit from file directory:	32
3.2.2.4 The backstage edits current machining program	32
3.2.2.5 The backstage edits other programs than current machining program	33
3.2.2.6 The backstage edits and creates new programs	33
3.2.3 "Program editing" sub-interface	34
3.2.3.1 Edit current machining program	34
3.2.3.2 Create new programs	35
3.2.3.3 Block operation	35

3.2.4	"Verify program" sub-interface.....	36
3.2.4.1	"Verify program" runs.....	36
3.2.4.2	Exit from "Verify program"	36
3.2.5	"Tool compensation setup" sub-interface.....	37
3.2.5.1	Direct input mode of tool compensation value	37
3.2.5.2	Trial cut input mode of tool compensation value.....	38
3.2.5.3	Overall translation of tool offset	38
3.2.5.4	Exit from input activated state	39
3.2.6	"Parameter configuration" sub-interface.....	39
3.2.6.1	Direct input of parameter configuration value.....	40
3.2.6.2	Direct input of pitch error compensation value	40
3.2.7	"Graphics setup" sub-interface.....	40
3.2.7.1	Graphics workblank setup and zoom.....	41
3.3	"Setup" function set interface and basic operation	42
3.3.1	"Setup" function set interface and function	42
3.3.2	"Coordinate system" sub-interface.....	43
3.3.2.1	Direct input mode of coordinate value.....	44
3.3.2.2	Input mode of current value	44
3.3.2.3	Input mode of increment value	44
3.3.3	"Tool life" sub-interface.....	45
3.3.3.1	Tool life index setup	45
3.3.3.2	Tool life alarm strategy setup.....	46
3.3.4	"Tool measurement sub-interface"	46
3.4	"Program" function set interface and basic operation.....	52
3.4.1	"Program" function set interface and function.....	52
3.4.2	Management of files in system disk, USB flash disk and network disk	53
3.4.2.1	Management program search.....	53
3.4.2.2	Program copy and paste.....	53
3.4.2.3	Program deletion.....	54
3.4.3	Create new programs	54
3.4.4	Program rename	54
3.4.5	Program mark setup	55
3.4.6	Programs are sorted by name and time	55
3.4.7	Program write/read setup	55
3.4.8	Create a new catalogue	55
3.5	"Diagnosis" function set interface and basic operation.....	56
3.5.1	"Diagnosis" function set interface and function.....	56
3.5.2	Alarm history export.....	57
3.5.3	Status record export	57
3.5.4	"Ladder diagram" sub-interface.....	58
3.5.4.1	Ladder diagram monitoring	58
3.5.4.2	Ladder diagram editing.....	59
3.5.4.3	Ladder diagram information	59
3.5.4.4	Ladder diagram signal tracking	60
3.5.5	Display of register status and macro-variable value	63

3.6	"Maintain" function set interface and basic operation	64
3.6.1	"Maintain" function set interface and function	64
3.6.2	Parameter setup.....	66
3.6.3	Parameter validation method and operation.....	67
3.6.4	Classification and switch of management permission	68
3.6.5	System upgrade.....	70
3.6.6	Data management	71
3.6.7	User setup	73
3.6.7.1	Display setup	73
3.6.7.2	Set "P parameter"	74
3.6.7.3	Set "M code"	75
3.6.7.4	Set "PLC switch"	77
3.6.7.5	Communication setup	78
3.6.7.6	Personalized setup.....	85
3.6.8	Process package setup.....	86
4	Power-on, power-off, safety protection, emergency stop	88
4.1	Power-on.....	88
4.2	Power-off.....	88
4.3	Over travel protection and release	89
4.3.1	Over travel protection	89
4.3.2	Hard over travel release	89
4.3.3	Over travel release	90
4.4	Emergency stop	90
4.4.1	Feed hold	90
4.4.2	Reset	90
4.4.3	Emergency stop.....	91
5	Manual operation and speed override	92
5.1	Return to the reference point manually.....	92
5.2	Move coordinate axis by manual feed	93
5.3	Move coordinate axis quickly manually	94
5.4	Move the coordinate axis by increment feed	95
5.5	Move the coordinate axis by handwheel feed.....	96
5.6	Manual control of spindle.....	97
5.7	Other manual operations.....	98
5.8	Speed override	98
5.8.1	Feed speed override	98
5.8.2	Quick traverse speed override.....	99
5.8.3	Selection of increment magnification	99
6	Program editing and management	100
6.1	Program search	100
6.1.1	Search of machining or editing program.....	100
6.1.1.1	Direct search	100
6.1.1.2	Search programs under different disks by "Search" function	101
6.1.1.3	Search programs under the directory by "Search" function.....	101
6.1.2	Search of management program (to be transmitted and deleted).....	102

6.1.2.1	Direct search	102
6.1.2.2	Search programs under different disks by "Search" function	103
6.1.2.3	Search programs under the directory by "Search" function.....	103
6.2	Program editing	104
6.2.1	Create new programs	104
6.2.1.1	Create new programs under "Machining" function set	104
6.2.1.2	Create new programs under "Program" function set.....	105
6.2.2	Modification and editing of program	106
6.2.2.1	Editing and modification of current loading program.....	106
6.2.2.2	Editing and modification of backstage of non-loading program	106
6.2.3	Save program as	107
6.2.3.1	Save program as of "Current loading program"	107
6.2.3.2	Save program of "Non-loading program"	108
6.2.4	Copy and paste of program segment.....	109
6.3	Program management	110
6.3.1	Rename of file directory and program	110
6.3.2	Copy and paste file directory and program.....	110
6.3.3	Program deletion.....	111
6.3.3.1	Program deletion under "Machining" function set.....	111
6.3.3.2	Program deletion under "Program" function set	112
7	Automatic operation	113
7.1	Automatic operation	113
7.1.1	Load machining program.....	113
7.1.1.1	Load a new program as machining program.....	113
7.1.1.2	Load existing programs as machining program.....	114
7.1.2	Program run	114
7.1.3	Program verification	115
7.1.4	Program graphics simulation	115
7.2	Automatic operation control	116
7.2.1	Single-segment operation.....	116
7.2.2	Skip operation.....	116
7.2.3	Run from any line	117
7.2.4	Stop operation.....	119
7.2.5	Select stop.....	119
7.2.6	Suspend operation.....	120
7.2.7	Terminate operation	121
7.3	MDI operation	122
7.4	Handwheel trial cut.....	124
7.5	Machining information query	125
8	Tool setting and machining setup	126
8.1	Directly input tool compensation value	126
8.1.1	Directly input tool compensation value under "Machining" function set	126
8.1.2	Directly input tool compensation value under "Setup" function set	126
8.2	Tool setting for trial cut	127
8.2.1	Tool setting for trial cut under "Machining" function set	127

8.2.2	Tool setting for trial cut under "Setup" function set.....	128
8.3	Overall translation of tool rest	130
8.3.1	Overall translation of tool rest under "Machining" function set	130
8.3.2	Overall translation of tool rest under "Setup" function set	130
8.4	F/S machining setup	131
8.5	Tool measurement.....	132
8.6	Thread repair function	134
9	Machine tool commissioning.....	135
9.1	System upgrade.....	135
9.1.1	System upgrade.....	135
9.1.2	System backup	136
9.2	Batch commissioning.....	137
9.2.1	Batch load commissioning.....	137
9.2.2	Batch backup commissioning	138
9.3	Pitch error compensation	139
9.3.1	Generation of pitch error compensation data file.....	139
9.3.2	Operation of thread repair error compensation sub-interface	140
9.3.3	Import of pitch error compensation data file	141
10	Use and maintenance information	143
10.1	Environmental conditions	143
10.2	Grounding.....	143
10.3	Power supply	144
10.4	Dust removal of filter screen of fan.....	144
10.5	Use after long-time idle	144

1 Overview

HNC-818 CNC turning system includes HNC-808Di-TU, HNC-818Ai-TU and HNC-818Bi-TU. This manual is based on HNC-808Di-TU. In case of discrepancies, please refer to the specification provided by the machine tool manufacturer.

1.1 Basic Operation

HNC-808Di-TU system is used for CNC device of CNC lathe and the MCP panel is furnished with 6 working mode keys "manual, automatic, single segment, MDI, increment/handwheel and reset". During operation of CNC lathe, function description and content of these 6 working modes are shown below.

Working mode	Functional description	Function application
Manual	Control continuous movement of machine tool axis and auxiliary action by Manual key, and so on.	Preparation for parts machining and simple machining, and so on.
Automatic	The machine tool runs continuously and automatically according to the edited program.	Continuous and automatic machining and program verification of parts, and so on.
Single segment	The machine tool runs automatically segment by segment according to the edited program.	Machining position inspection and program verification.
MDI*	The machine tool runs the manually inputted program.	Automatic machining and coordinate setup of simple parts, and so on.
Increment/handwheel	Accurately control axis movement of machine tool by key or handwheel.	Tool setting or manual machining of simple parts, and so on.
Reset	Control the position where all axes of the machine tool return to the reference point.	Verify position of machine tool after start, and so on.

*HNC system is not Di series version and MDI mode is configured as MDI function set

1.2 Basic function

To complete different work under different working modes, corresponding application functions should be used. The NC panel of HNC-808Di-TU CNC device is furnished with 6 function keys "Machining, setup, program, diagnosis, maintenance and user-defined (MDI)". Every function key corresponds to a group of function sets. Every group of function sets can select corresponding functions and interfaces 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	Functional description	Function content
Machining	Functions necessary for automatic machining	Edit new program*, edit current loading program, edit selected program, machining program selection, program verification, tool setting, tool compensation setup*, graphics setup, display switch, user macro, machining information and parameter configuration (user)*
Setup	Relevant functions of tool setting	Tool setting, tool compensation setup*, coordinate setup, tool life management, tool automatic measurement and thread repair
Program	User program management function	Edit new program*, select, copy, paste and delete programs from system disk, USB flash disk and network disk, program rename and sort, set mark
Diagnosis	Fault diagnosis, performance commissioning, intelligent function	1. Fault diagnosis function: Alarm message, alarm history, ladder diagram, PLC status, macro-variable, log and other functions; 2. Performance commissioning function: Servo adjustment 3. Intelligent function: QR code, fault record and screw load inspection, and so on
Maintain	Hardware setup, parameter setup, system upgrade, basic information, data management and relevant maintenance functions	1. System hardware device configuration and configuration sequence setup function: Device configuration 2. Setup function of common parameters: Parameter setup 3. Setup function of user optional parameters: Parameter setup* 4. System upgrade and commissioning function: Batch commissioning, data management, system upgrade, permission management and user setup 5. Registration, basic information and other functions: Registration, machine tool information, system information, process package and time setup
User-defined ** (MDI)	Related functions of manual data input	Pause, 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 products than HNC-818 Di series products, user-defined key is often set as MDI function.

1.3 Basic display interface

HNC-808Di-TU 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 current status and information of system through interface or have a man-machine conversation in the conversation area and realize instruction input, parameter setup and other operations.

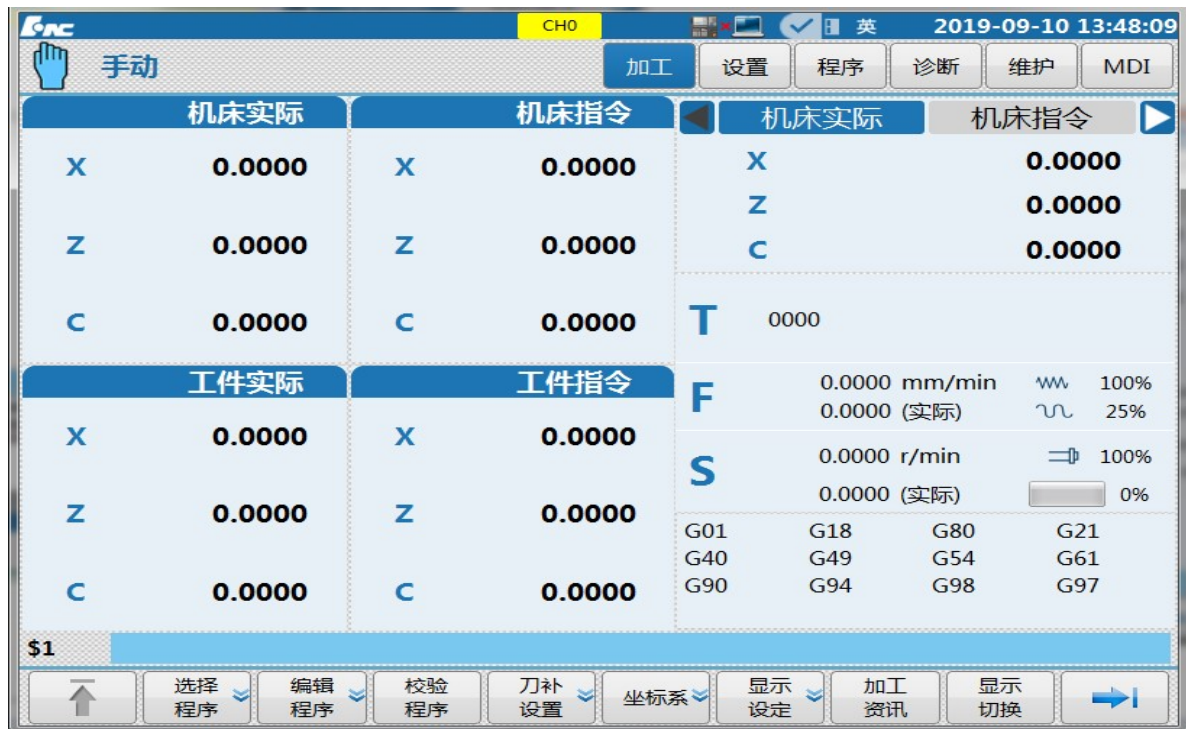
All interfaces are briefly introduced based on HNC-808Di-TU standard configuration.

1.3.1 Machining display interface

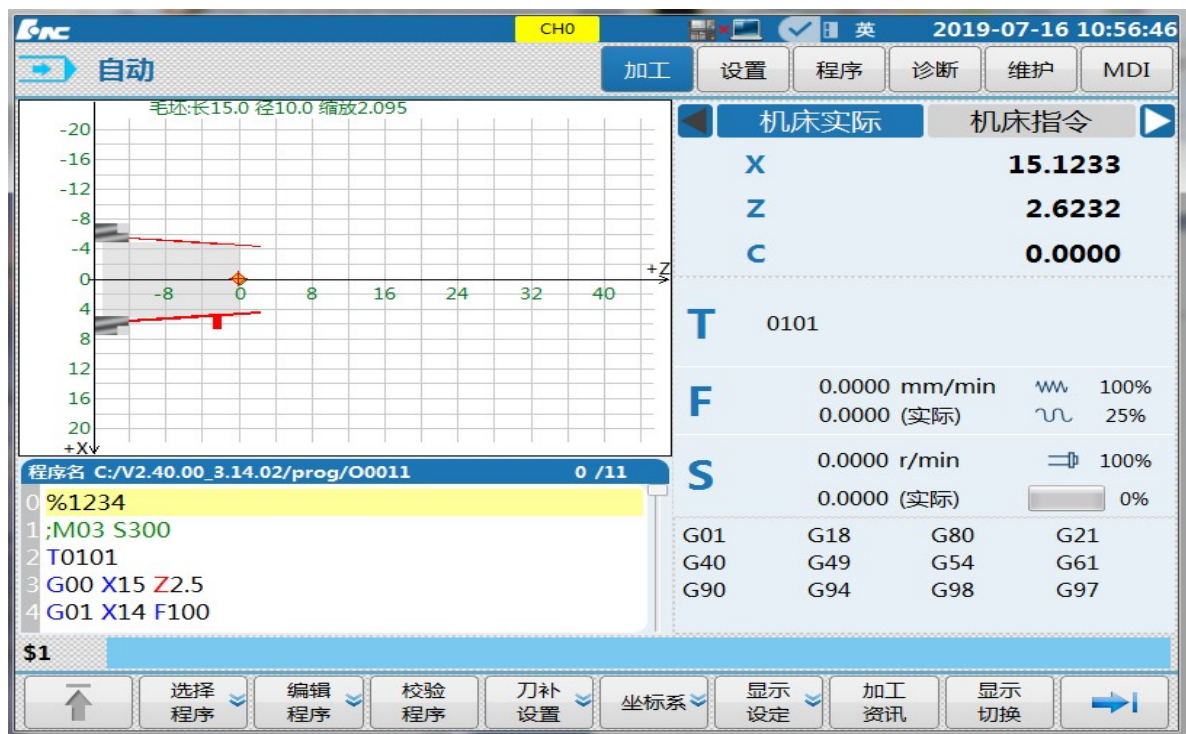
The machining display interface enables the operator to observe the machining process and has 4 display forms, big character coordinate + program, joint coordinate, graphics path + program and program. These 4 interface can be switched through 『Display switch』 soft key.



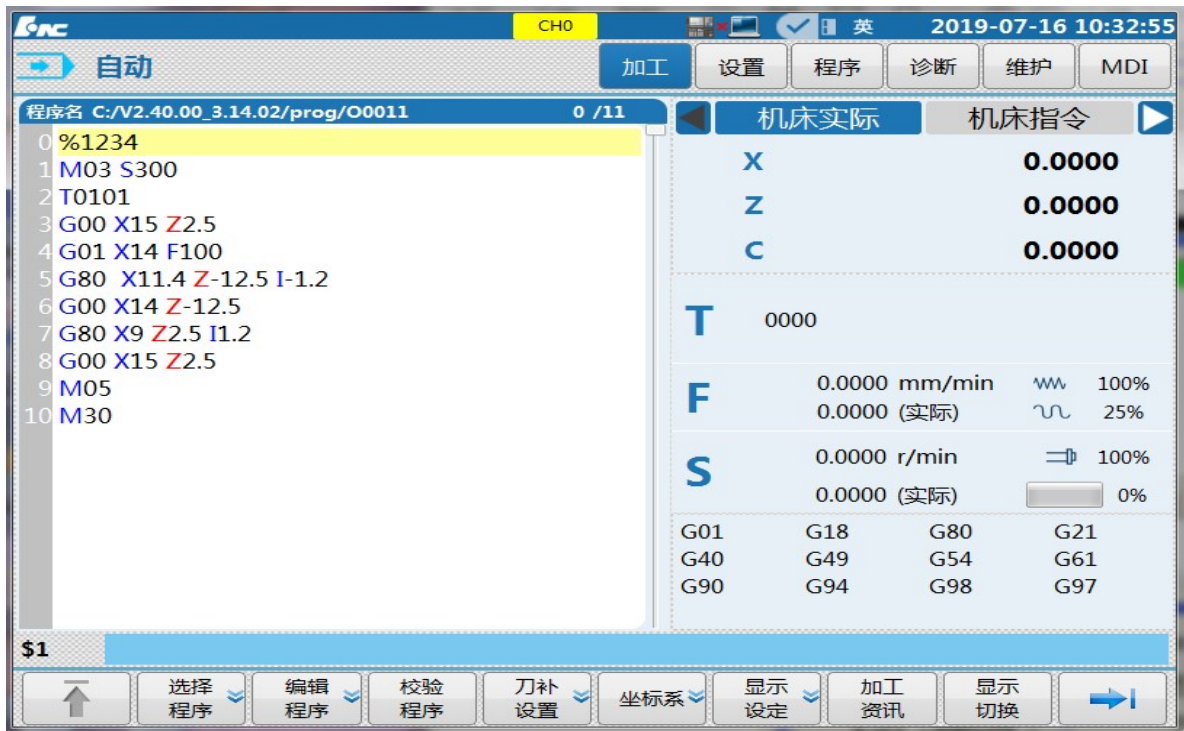
Big character coordinate + program display interface



Joint coordinate display interface



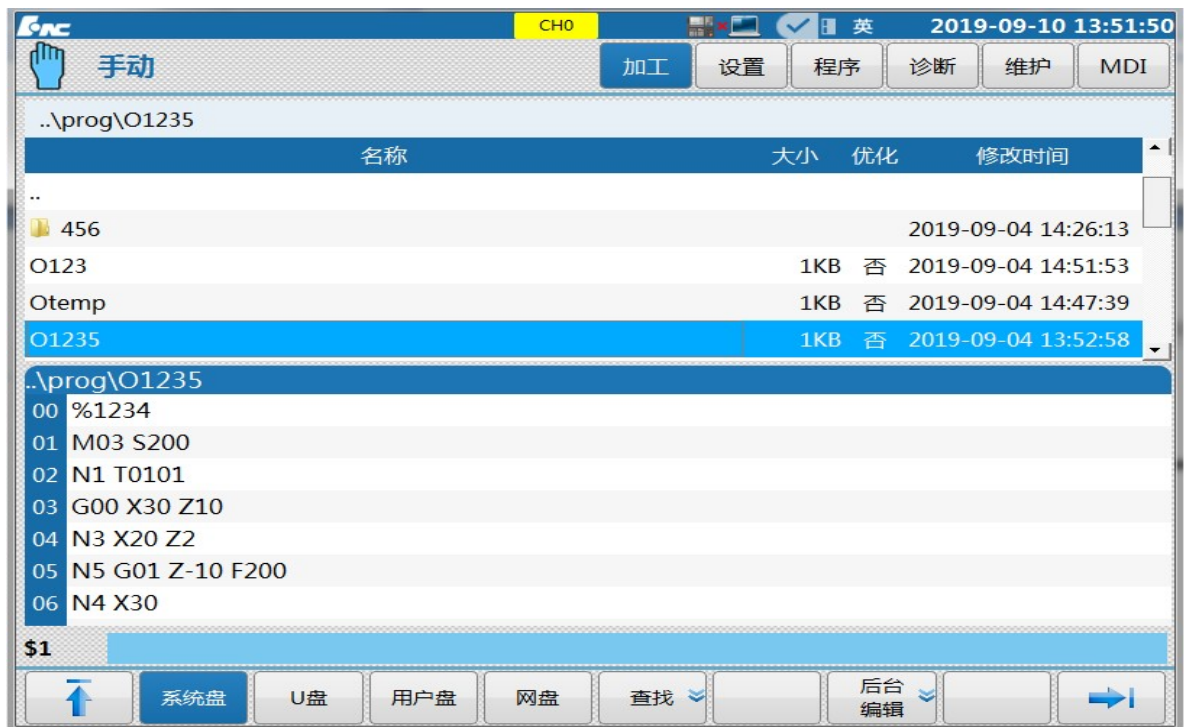
Graphics path + program display interface



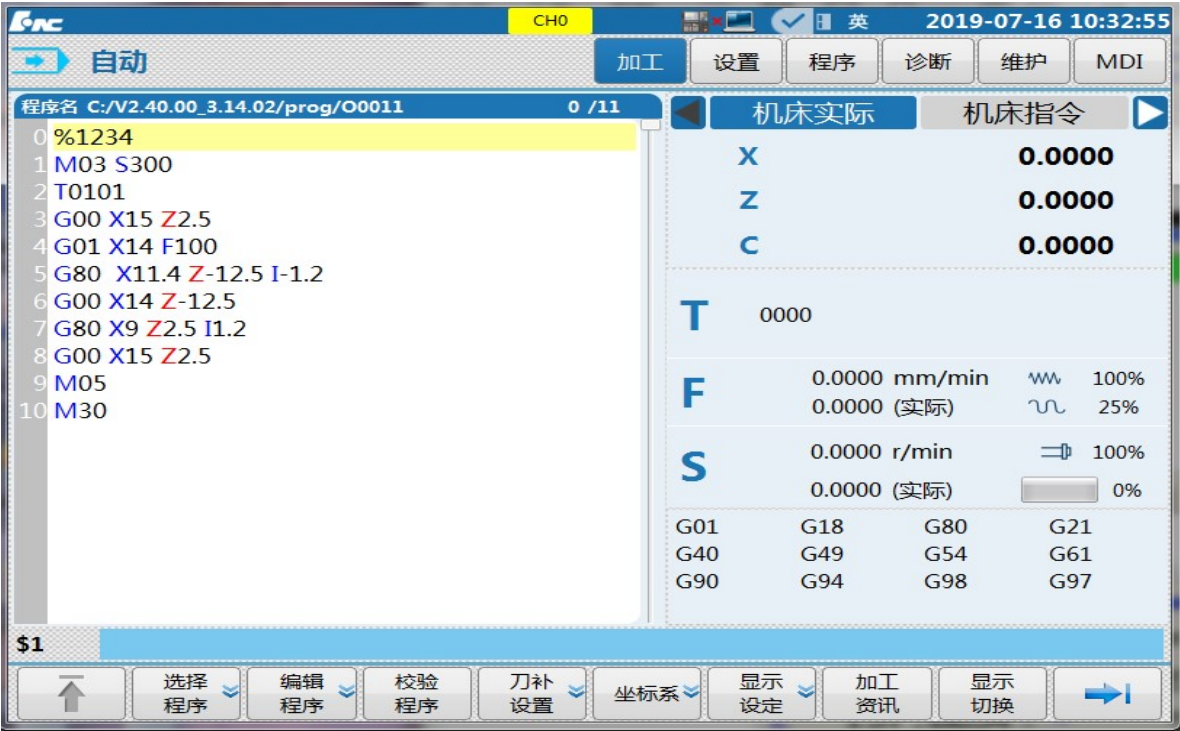
Program display interface

1.3.2 Program selection and editing interface

This system can select programs by cursor. When the cursor selects a program name in the list, the first segments of the program will be displayed in the lower part of the screen in order to confirm the program searched.



Program selection interface



Program editing interface

1.3.3 Machining setup interface



Machining

1.3.4 Parameter setup interface



Parameter setup interface

2 Operating equipment

2.1 System host panel (NC panel)

2.1.1 System host panel zoning

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



- (1)---LOGO
- (2)---USB interface
- (3)---Alphabetical keyboard area
- (4)---Number and character key area
- (5)---Cursor key area
- (6)---Function key area
- (7)---Soft key area
- (8)---Screen display interface area

2.1.2 Display interface zoning

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



(1) ---Title bar

- Machining mode: Working mode of the system can switch among automatic (run), single segment (run), manual (run), increment (run), reset and emergency stop according to status of corresponding keys on the control panel of the machine tool;
- System alarm message;
- Level 0 main menu name: Display currently activated main menu keys;
- USB flash disk connection and network connection;
- System sign, time.

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

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

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

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

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

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

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

2.1.3 Definition of host panel keys

The host panel includes

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

MDI keyboard function

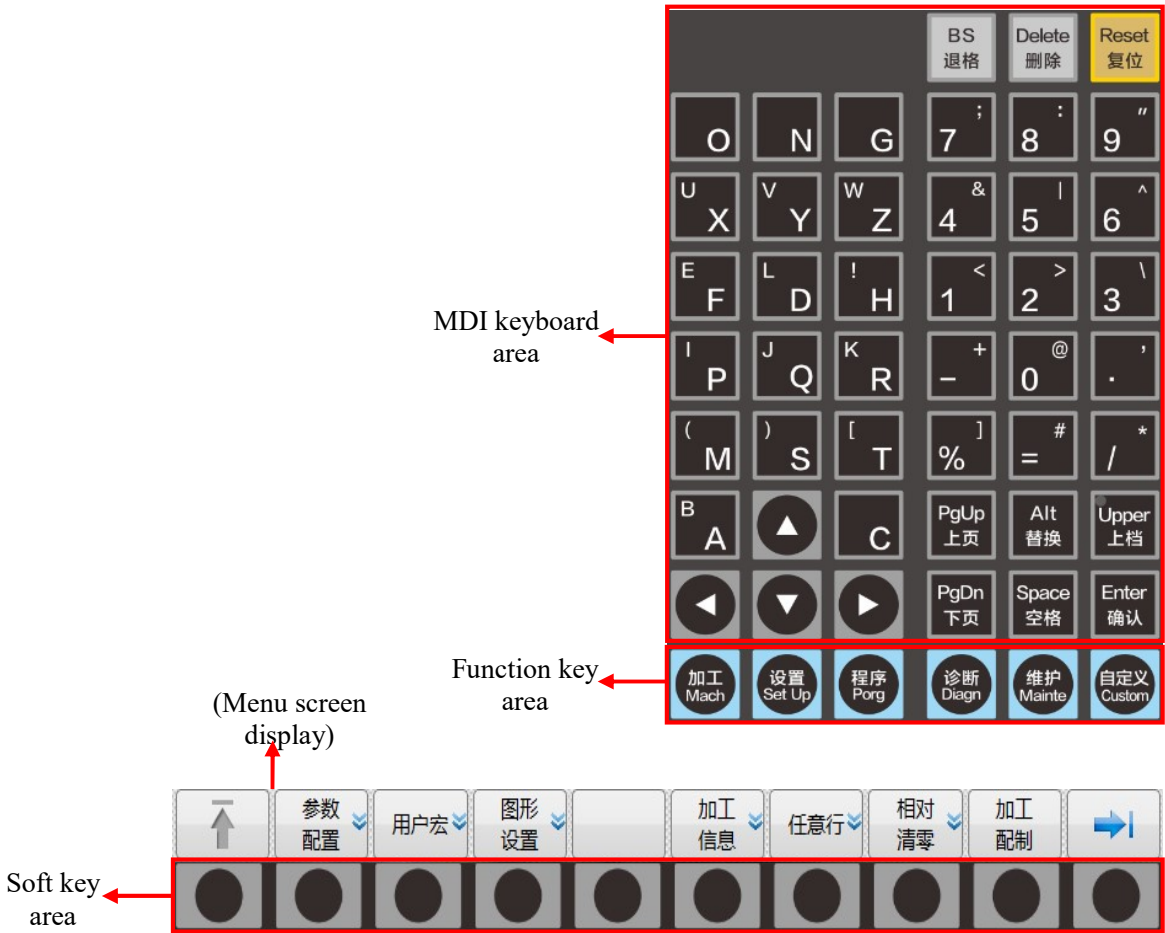
Input and edit instructions by this keyboard. Most keys have functions of upper gear key. Press "Upper gear" key and letter/number key simultaneously to input letter/number of upper gear key.

Function key function

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

Soft key function

There are 10 soft keys below the screen of HNC-808Di-TU system, on which there are no fixed signs. Left and right ends are 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 above the screen and they differ with change of menus (for specific functions, fixed signs, refer to chapter 3).



Function of keys on MDI keyboard

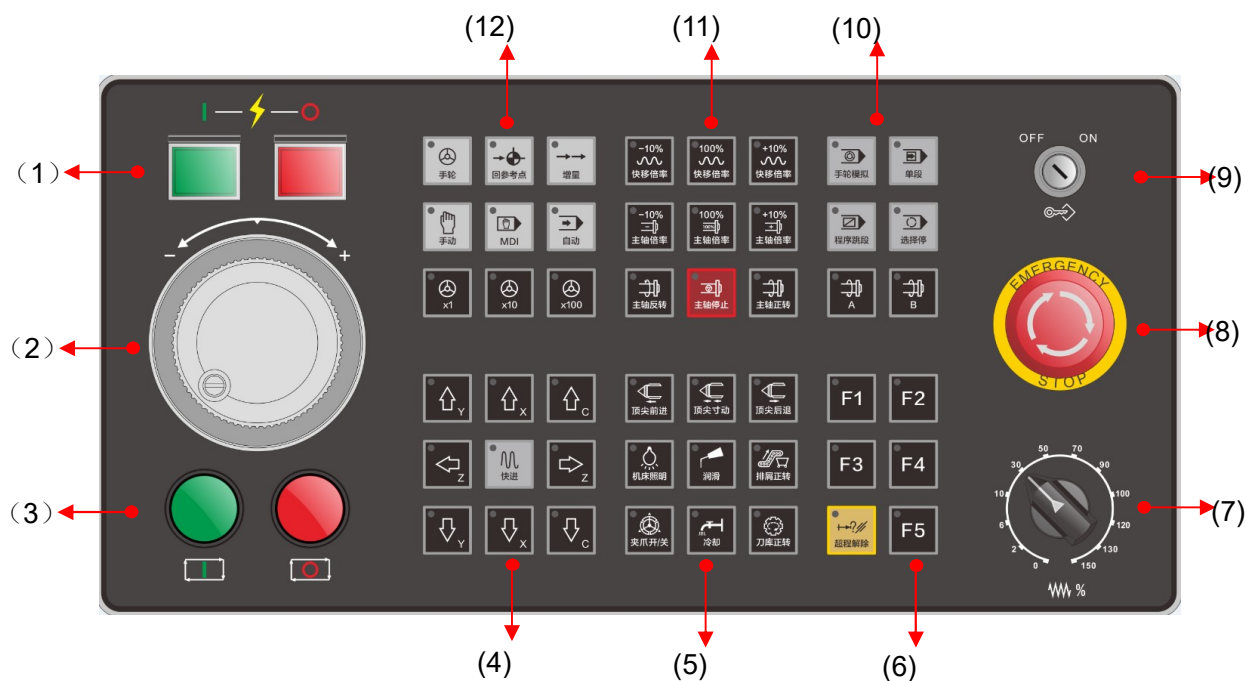
Key	Name/symbol	Functional description
	Character key (letter, number, symbol)/ 「Letter」 (such as 「Y」)	Input letters, numbers and characters. Every key has upper and lower gears. When "Upper and lower gear key" are pressed simultaneously and then press "Character key" to input the above characters; otherwise, input the below characters.
	Cursor shift key/ 「Cursor」	Control the cursor to move horizontally and vertically.
	Program name symbol key/ 「%」	Backspace key/
	Backspace key/ 「Backspace」	Delete characters forward, and so on.
	Delete key/ 「Delete」	Delete current programs and characters, and so on.
	Reset key/ 「Reset」	CNC reset, feed, input stop, and so on.

Alt 替换		Alternate key/ 「Alt」	Press 「Alt」 + 「Cursor」 to switch content of the display frame (position, compensation and current, and so on) on the top right corner of the interface. (Detailed in 3.2.1.5); Press 「Alt」 + 「P」 for screenshot.
Upper 上档		Shift key/ 「Shift」	Press the double address key to switch upper and lower gear keys. Press upper and lower gear keys and double address key and the upper gear key is valid.
Space 空格		Space key/ 「Space」	Leave one blank space backward.
Enter 确认		Confirmation key/ 「Enter」	Open and conform input.
PgUp 上页	PgDn 下页	Page up or page down key/ 「Page up or page down」	Switch previous and next pages in the same display interface.
<div>加工 Mach</div> <div>设置 Set Up</div> <div>程序 Porg</div> <div>诊断 Diagn</div> <div>维护 Mainte</div> <div>自定义 Custom</div>		Function key/ [[Machining]] [[Setup]] [[Program]] [[Diagnosis]] [[Maintain]] [[User-defined]]	<p>Machining: Select function set necessary for automatic machining and corresponding interface.</p> <p>Setup: Select function set relating to tool setting and corresponding interface.</p> <p>Program: Select function set for user program management and corresponding interface.</p> <p>Diagnosis: Select function set for fault diagnosis, performance commissioning and intelligence and corresponding interface.</p> <p>Maintenance: Select relevant maintenance functions such as hardware setup, parameter setup, system upgrade, basic information and data management and corresponding interface.</p> <p>User-defined* (MDI): Select relevant functions of manual data input and corresponding interface.</p>
<div>↑</div> <div>.....</div> <div>→</div>		Soft key/ [[↑]] [[→]] [[“Function”]]	<p>There are 10 unidentified keys below HNC-808Di-TU display screen, namely soft keys. In different function sets or levels, functions correspond to those displayed above the screen. Main functions of soft keys are as follows:</p> <ol style="list-style-type: none"> 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. <p>in 10 soft keys, the leftmost key is return to the previous menu key, arrow is valid when it is blue and it is gray under level 1 menu of function set.</p> <p>In 10 soft keys, the rightmost key is continued menu key and arrow is valid when it is blue. Press this key for cyclic switch among interfaces in menus of the same level (menus of the same level of this system has no more than 2 pages).</p>

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

2.2 Operation panel of machine tool (MCP panel)











2.2.1 Operation panel zoning of machine tool





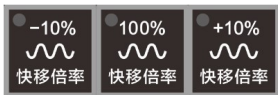



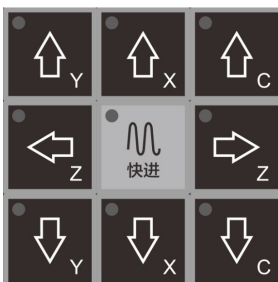



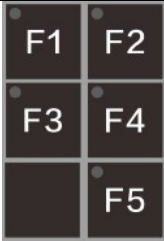


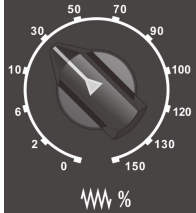
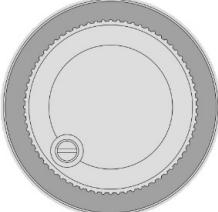


- (1)---Power switch
- (2)---Handwheel pulse generator
- (3)---Cycle start/feed hold
- (4)---Feed axis movement control key area
- (5)---Machine tool control key area
- (6)---Machine control extension key area
- (7)---Feedrate override band switch
- (8)---Emergency stop button
- (9)---Editing lock ON/OFF
- (10)---Operation control key area
- (11)---Speed magnification control key area
- (12)---Working mode selection key area

2.2.2 Definition of operation panel of machine tool

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

Key	Name/symbol	Functional description	Working mode at valid state
	Handwheel Working mode key / 【Handwheel】	Select handwheel mode.	Handwheel
	Reset Working mode key / 【Reset】	Select reset mode key.	Reset
	Increment Working mode key / 【Increment】	Select increment mode.	Increment
	Manual Working mode key / 【Manual】	Select manual mode.	Manual
	MDI Working mode key / 【MDI】	Select MDI mode.	MDI
	Automatic Working mode key / 【Automatic】	Select automatic mode.	Automatic
	Single segment ON/OFF key / 【Single segment】	1) Switch of segment-by-segment operation or continuous operation program. 2) The indicator light lights up when a single segment is valid.	Automatic, MDI (Including single segment)
	Handwheel simulation ON/OFF key / [Handwheel simulation]	1) Switch of whether to enable handwheel simulation function. 2) When this function is enabled, the handwheel controls the tool to run as per the program path. While swinging the handwheel in the positive direction, continue running the subsequent programs; while swinging the handwheel in the negative direction, backspace the run programs in the negative direction.	Automatic, MDI (Including single segment)
	Program skip ON/OFF key /[Program skip]	1) When a program segment is prefixed with "/", switch whether to skip the program segment.	Automatic, MDI (Including single segment)
	Optional stop ON/OFF key / [Optional stop]	1) When a programs run to "M00" instruction, switch whether to stop; 2) If this key has been pressed before program run (indicator light lights up), when the program runs to "M00" instruction, feed is held and press cycle start to continue running the subsequent programs. If this key	Automatic, MDI (Including single segment)

		is not pressed, consistently run the program.		
	Over travel release key / [Over travel release]	1) Cancel limits of machine tool; 2) Press and hold this key to release alarm and run the machine tool.		Handwheel, manual, increment
	Cycle start key / [Cycle start]	Run program and MDI instruction.		Automatic, MDI (Including single segment)
	Feed hold key / [Feed hold]	Suspend program and MDI instruction.		Automatic, MDI (Including single segment)
	Increment/handwheel magnification key / [Increment magnification]	When the handwheel rotates for 1 grid or "Manual control of axis feed key" is pressed once, the movement distance of the machine tool is 0.001mm/0.01mm/0.1mm.		Handwheel, increment, manual, reset, automatic, MDI (including single segment and handwheel simulation)
	Quick traverse speed override key / [Quick traverse override]	Override of quick traverse speed.		
	Spindle magnification key / [Spindle magnification]	Override of spindle speed.		
	Spindle control key / [Spindle positive/negative rotation]	Control positive rotation, negative rotation and stop of spindle.		Handwheel, increment, manual, MDI (including single segment and handwheel simulation)
	Power head control key / [Power head]	1) Control of positive and negative rotation of power head; 2) Press this key to switch between rotation/stop of power head.		
	Manual control of axis feed key / [Axis feed]	1) Control movement and direction of all axes under manual or increment mode; 2) Select handwheel control axis under handwheel mode; 3) When all axes are pressed under manual mode, the axes run as per feed speed. When "Quick traverse" is also pressed, the axes run as per quick traverse speed.		Handwheel, increment, manual
	Machine control key / [Machine control]	Manual control of auxiliary actions of machine tool	Tip advance, inch, retract. Clamping jaw ON/OFF. The magazine tool rotates positively.	Handwheel, increment, manual (and spindle stop)
			Lighting of machine tool. Lubrication. Positive rotation of chip removal. Cooling.	Handwheel, increment, manual, reset, automatic, MDI (including single segment and handwheel simulation)

	Machine tool control extension key / [Machine tool control]	Manual control of auxiliary actions of machine tool.	Machine tool manufacturer is set as needed
	Program protection switch / [Program protection]	Protection program should not be modified arbitrarily.	Handwheel, increment, manual, reset, automatic, MDI (including single segment and handwheel simulation)
	Emergency stop button / [Emergency stop]	In case of an emergency, the system and the machine tool immediately enter halt state and all outputs are closed.	
	Feed rate knob / [Feed rate]	Feedrate override.	Automatic, MDI, manual
	Handwheel / [Handwheel]	Control movement of machine tool. (When handwheel simulation function is valid, it can control the machine tool to move according to program path).	Handwheel
	System power-on / [Power-on]	Control power-on of CNC device.	Handwheel, increment, manual, reset, automatic, MDI (including single segment and handwheel simulation)
	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, return to the previous menu key and continued menu key and are identified using symbols in the following table.

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

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

3 Display interface

3.1 Display interface selection and menu structure

3.1.1 Common operation of interface and menu selection



- 1) There are 6 function keys on the NC panel, which can be used to select corresponding function sets and display interfaces.
- 2) There is a group of function menus in the lower part of every display interface and function menu is selected by soft key.
- 3) Every group of function menus consists of 10 soft keys (space key is often reserved), among which the leftmost key is "Return to the previous menu key" (『↑』), the rightmost key is "Continued menu key" (『→』) and arrow is valid when it is blue.
- 4) The interface displayed when function key is selected for the first time after startup is the default interface of the function set and the function menu below is level 1 main menu. Extension menu of this level can be searched by 『→』.
- 5) Menus of all levels under function set has at most 1 main menu and 1 extension menu. Press 『→』 for cyclic switch and only menu changes, interface does not change.
- 6) Interface selection before function set switch will be memorized. That is, while switching back to this function set, the displayed function menu and interface are the menu and interface upon the previous exit.



- 7) Function sets of this system are at most a level 4 menu structure. Press the right function soft key marked with “≡” to search the lower-level menu. Return to the previous menu by 『↑』.
- 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 user need. The user can configure special needs.
- 9) Generally data input and other man-machine dialog boxes can be opened using corresponding soft keys, but for some data input with high safety requirements, activate the input box using "Enter" (『Enter』) and input data or parameters.
- 10) When the man-machine dialog box does not exit, function sets cannot be switched by function keys.

11) Exit mode of man-machine dialog box:

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

3.1.2 Function menu structure

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

1) Level 1 menu of all function sets

Function key

Machining

(Level 1 menu)

(Level 1 extension menu)

Select Program	Edit Program	Verify Program	Tool compensation Setup	Coordinate System	Display Setup	Machining Information	Display Switch	⇒	Parameter Configuration	User Macro	Graph Setup	Machining Information	Any line	Relative Reset	Machining Configuration
----------------	--------------	----------------	-------------------------	-------------------	---------------	-----------------------	----------------	---	-------------------------	------------	-------------	-----------------------	----------	----------------	-------------------------

Setup

(Level 1 menu)

(Level 1 extension menu)

Tool compensation Setup	Tool rest Translation	Trial cut Diameter	Trial cut Length	Coordinate System	Tool Life	Relative Reset	⇒	Tool Measurement	Thread Repair
-------------------------	-----------------------	--------------------	------------------	-------------------	-----------	----------------	---	------------------	---------------

Program

(Level 1 menu)

(Level 1 extension menu)

Function key Disk	USB flash disk	Network disk	Create Program	Search	Copy	Paste	Delete	(Level 1 extension menu)	Setup Sign	Rename Name	Name Rank	Time Rank	Create Contents	Writable	read only	User Disk
-------------------	----------------	--------------	----------------	--------	------	-------	--------	--------------------------	------------	-------------	-----------	-----------	-----------------	----------	-----------	-----------

Diagnosis

(Level 1 menu)

(Level 1 extension menu)

Alarm Information	Alarm History	Ladder diagram	Status Display	Axis monitoring	Macro variable	Servo Adjustment	⇒	Self-inspection	Log	Fault Video	Screw Load	QR Code
-------------------	---------------	----------------	----------------	-----------------	----------------	------------------	---	-----------------	-----	-------------	------------	---------

Maintain

(Level 1 menu)

(Level 1 extension menu)

Device Configuration	Parameter Setup	Parameter Configuration	User Setup	Batch Commissioning	Data Management	System Upgrade	Permission Management	⇒	Space Compensation	Time Setup	Process Package	Machine tool Information	System Information	Register
----------------------	-----------------	-------------------------	------------	---------------------	-----------------	----------------	-----------------------	---	--------------------	------------	-----------------	--------------------------	--------------------	----------

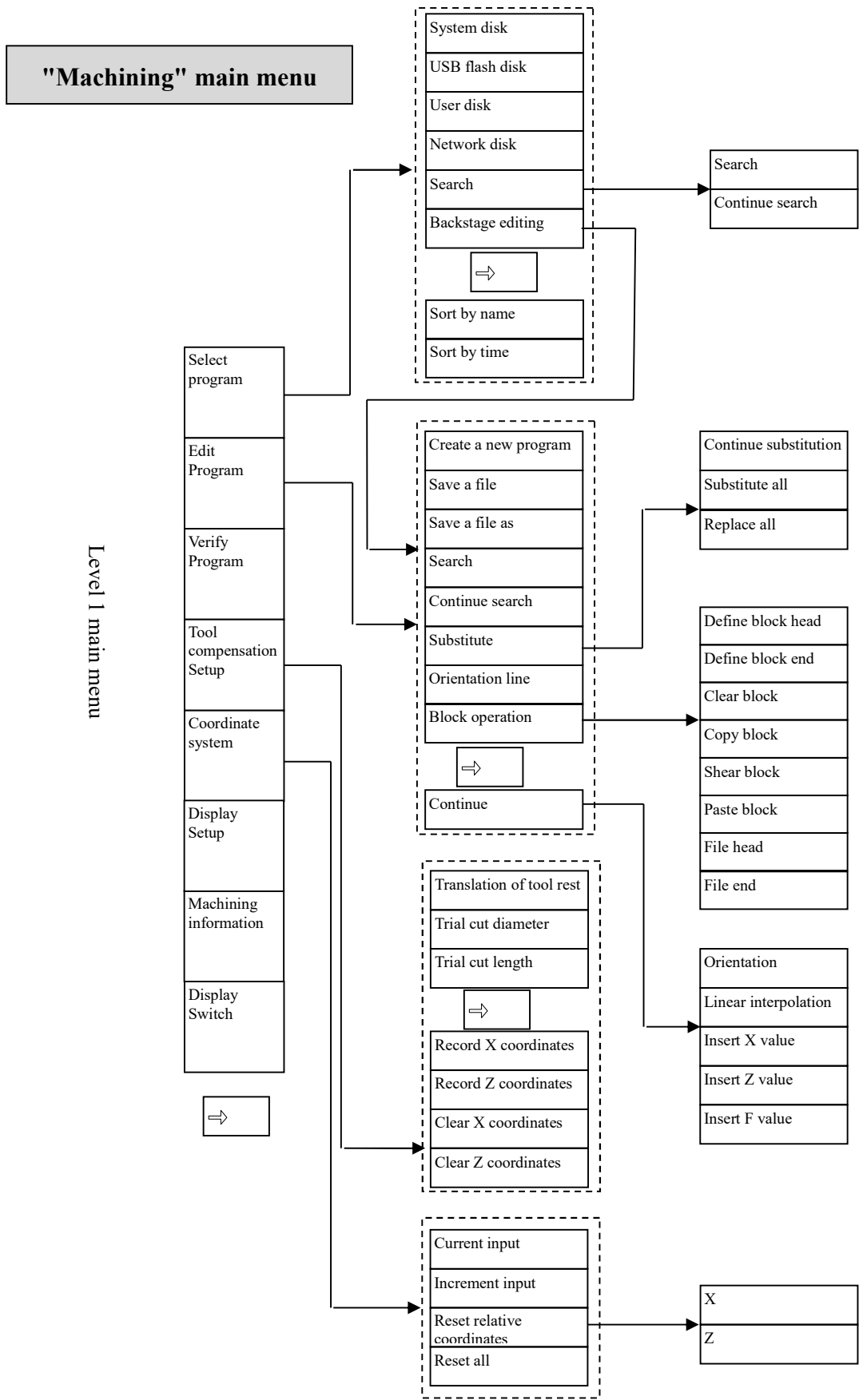
MDI

(Level 1 menu)

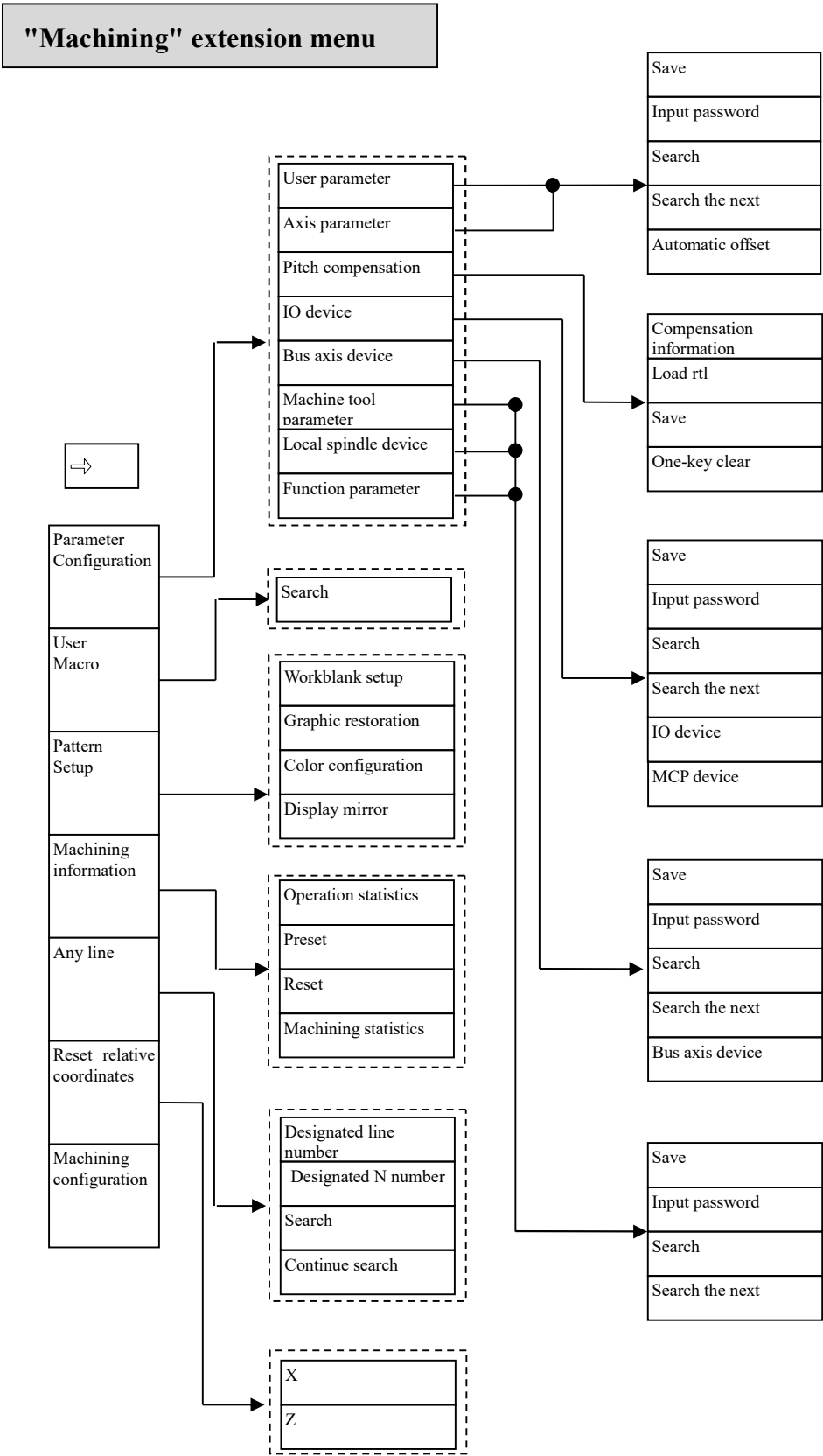
Suspend	Clear	Save	Input
---------	-------	------	-------

2) Menu structure of "Machining" function set

(1) "Machining" main menu

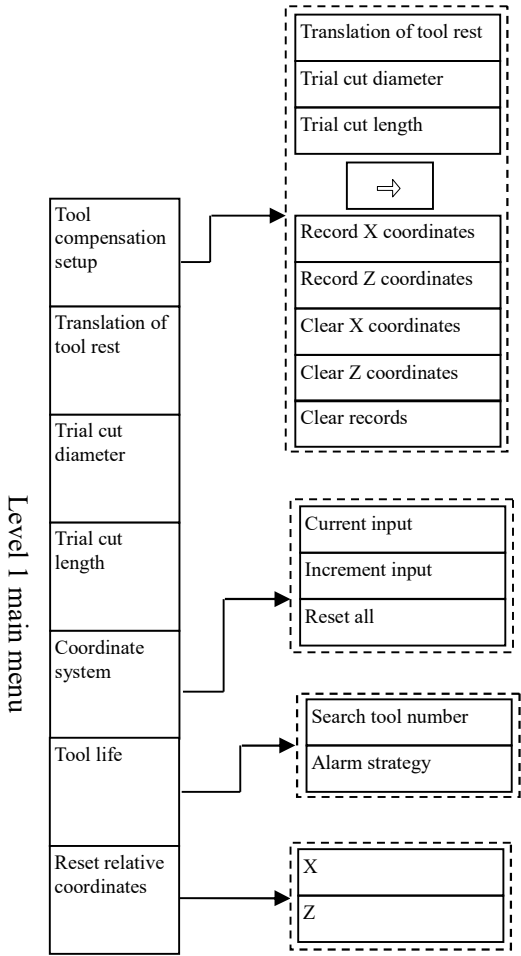


(2) "Machining" extension menu

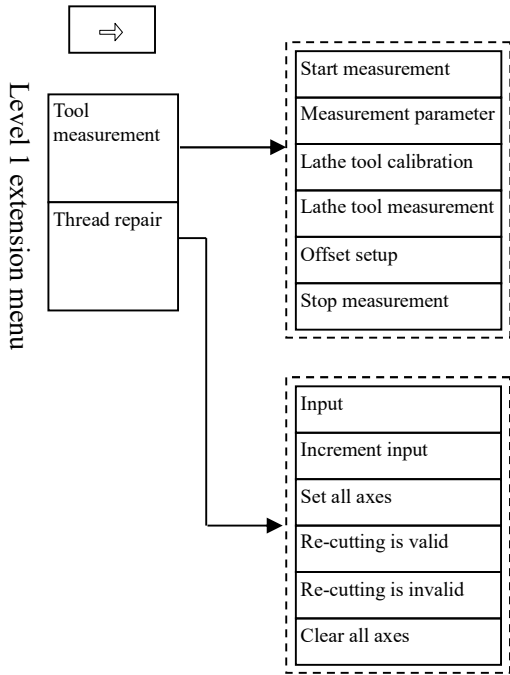


3) Menu structure of "Setup" function set

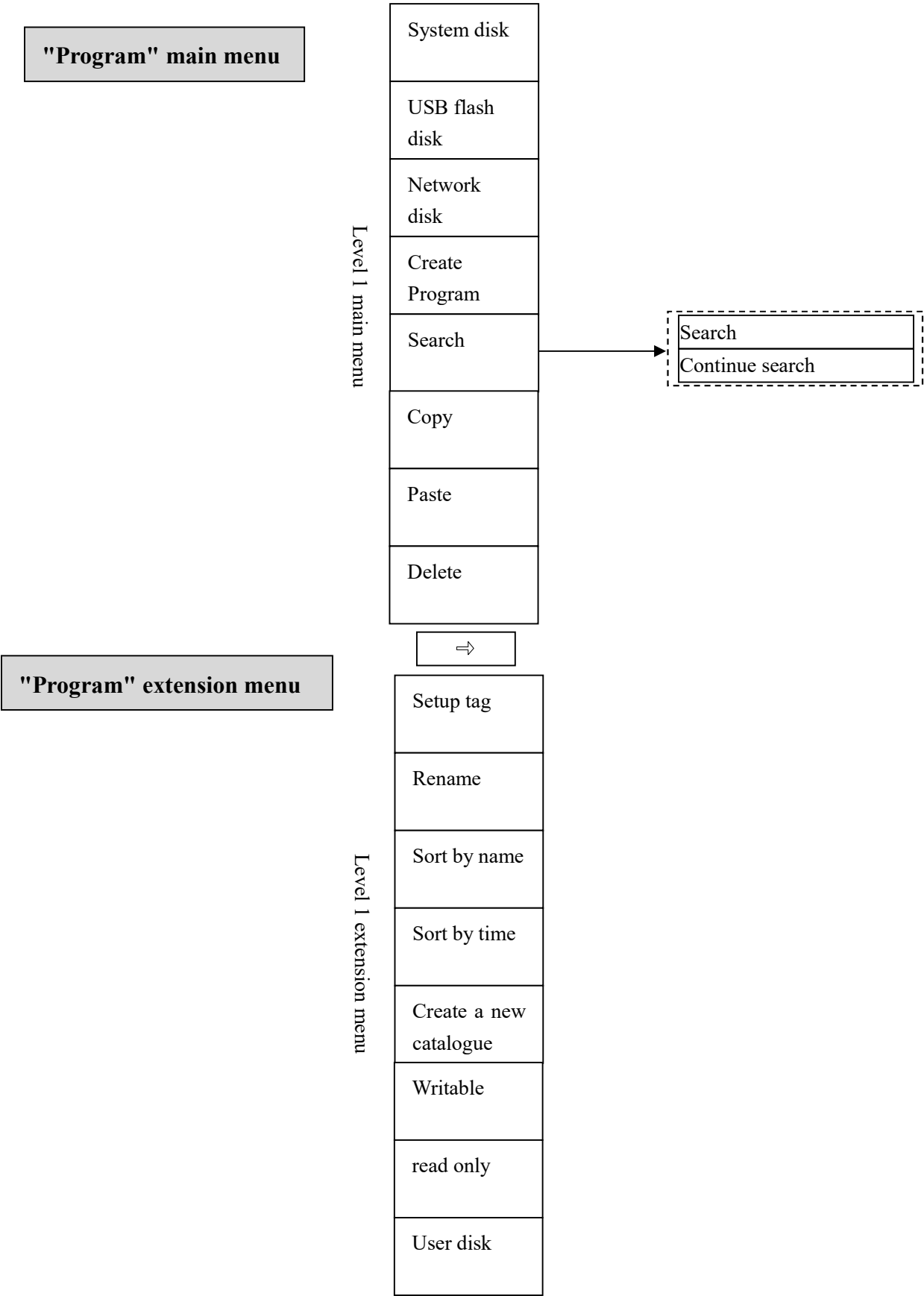
"Setup" main menu



"Setup" extension menu

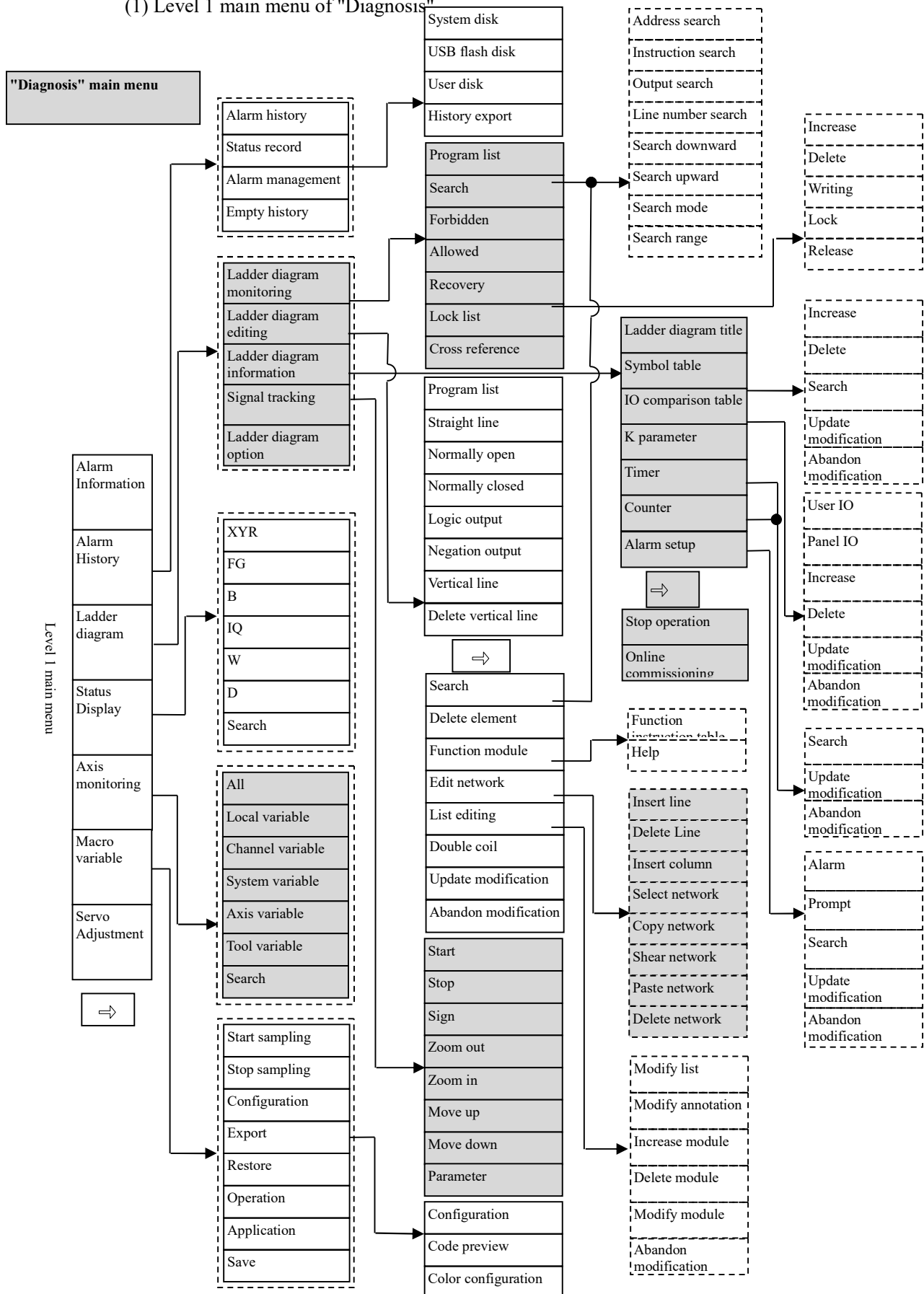


4) Menu structure of "Program" function set

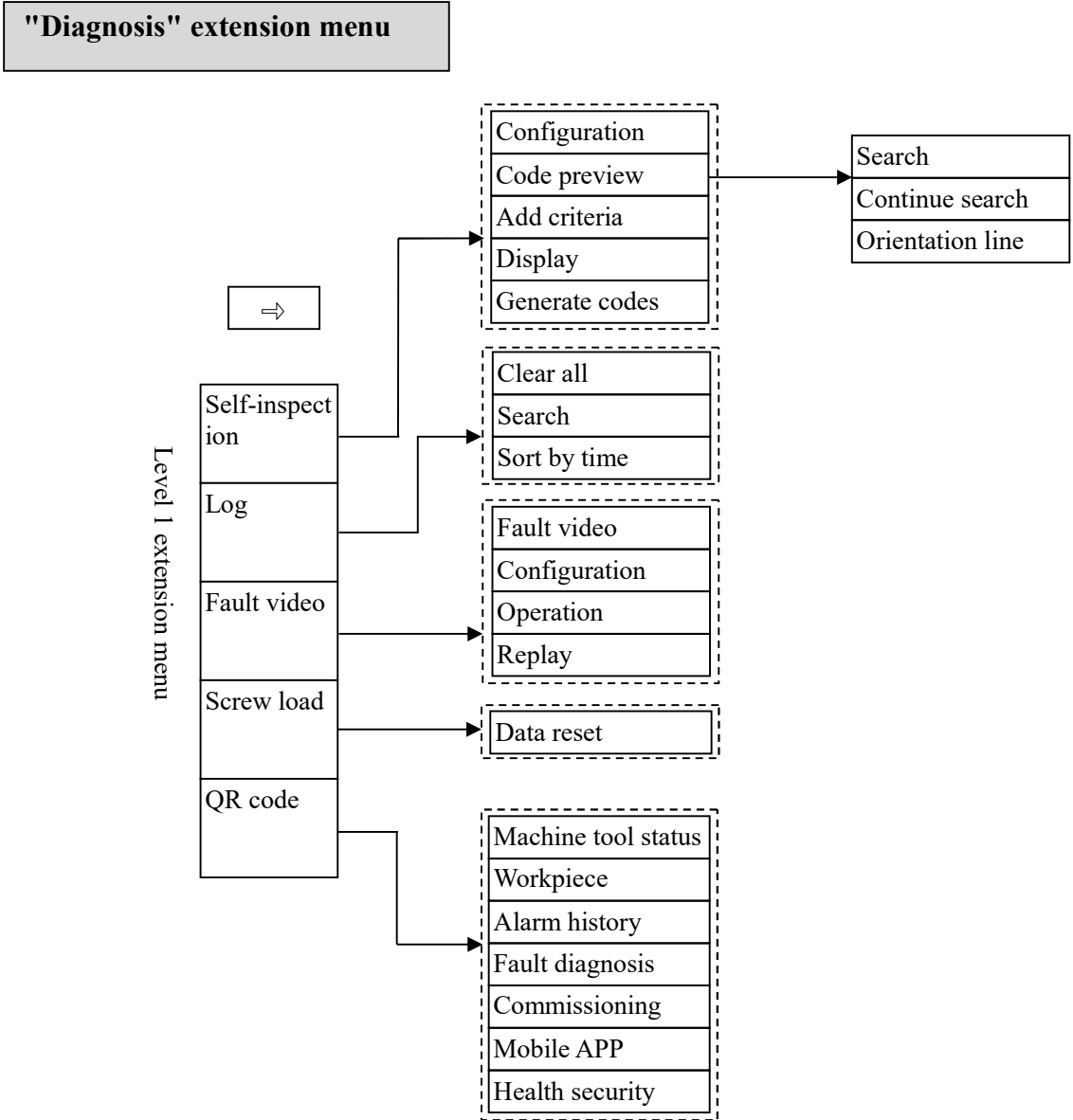


5) Menu structure of "Diagnosis" function set

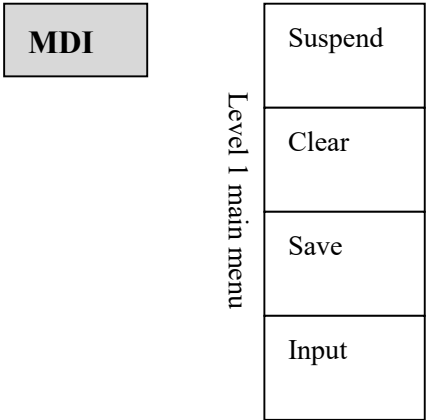
(1) Level 1 main menu of "Diagnosis"



(2) Level 1 extension menu of "Diagnosis"

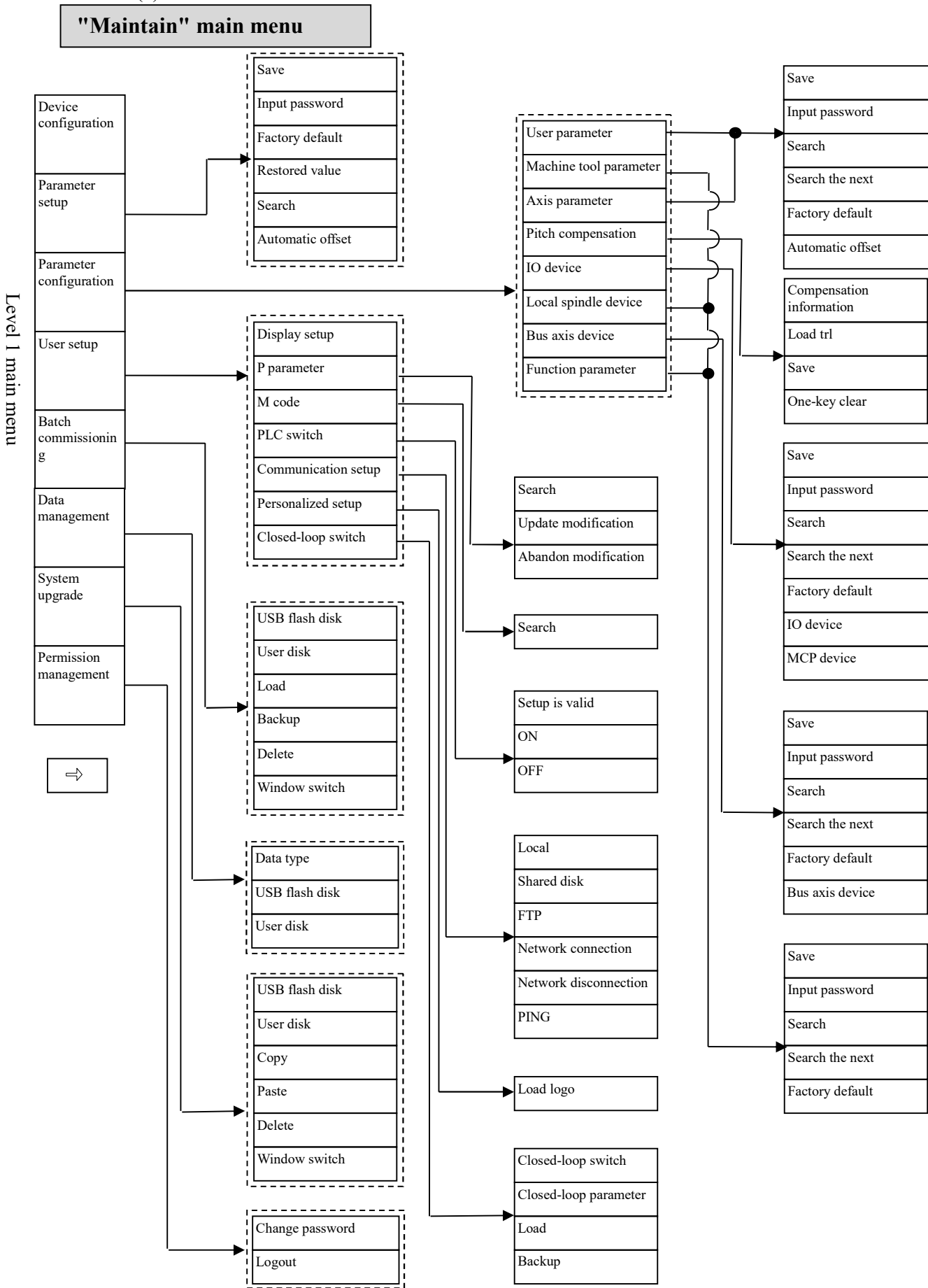


6) Menu structure of "User-defined (MDI)" function set

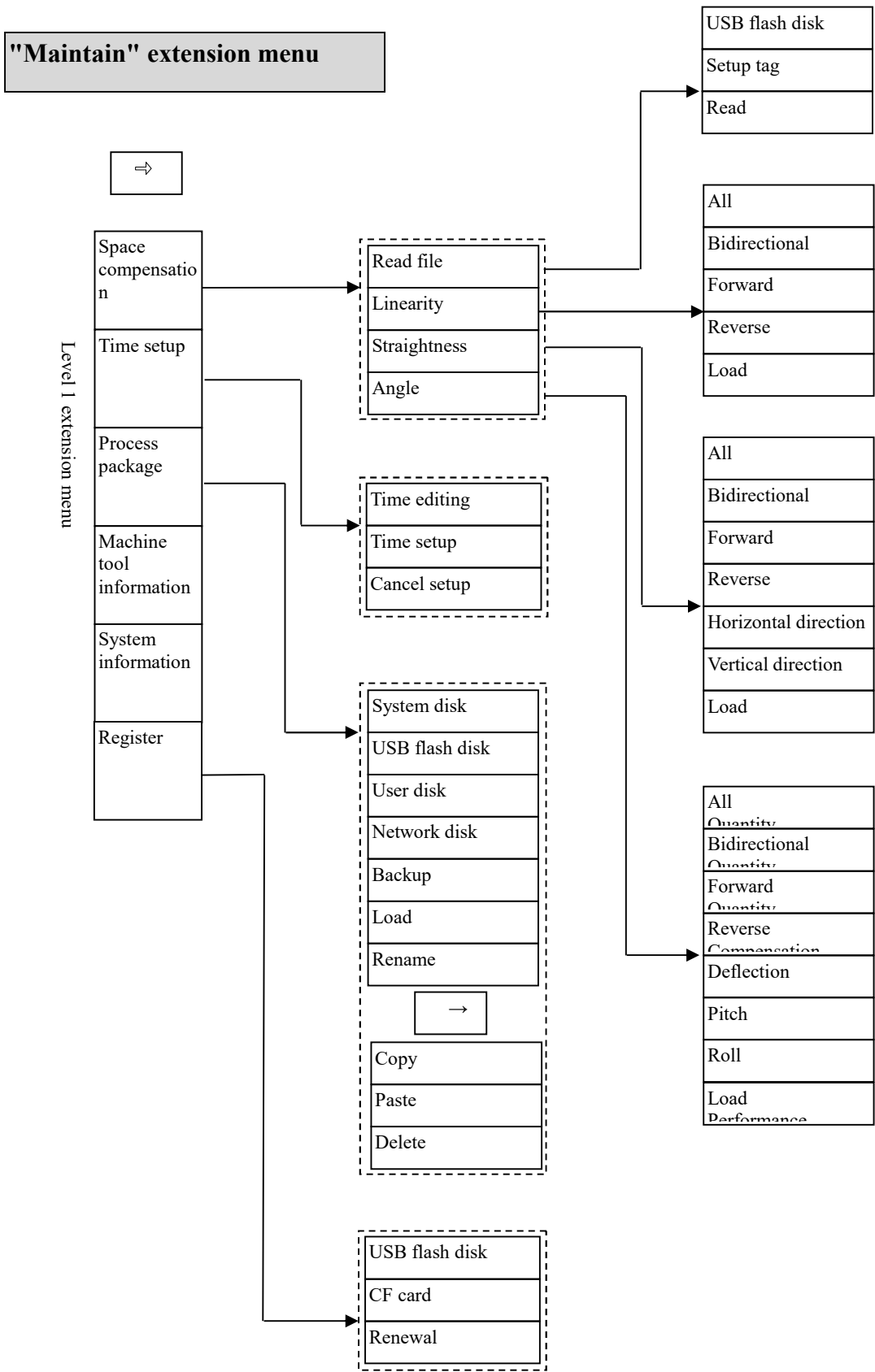


7) Menu structure of "Maintain" function set

(1) Level 1 main menu of "Maintain"



(2) Level 1 extension menu of "Maintain"



3.2 Display interface and basic operation of "Machining" function set

3.2.1 Interface and function of "Machining" function set

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



Select program	Edit program	Verify program	Tool compensation setup	Coordinate system	Display setup	Machining information	⇒
Display switch	Parameter configuration	User macro	Graphics setup	Machining information	Any line	Reset relative coordinates	Machining configuration

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

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

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

Tool compensation setup: Tool offset and wear value can be set under the sub-interface of this function, or tool offset can be set through 『Trial cut diameter』 and 『Trial cut length』 soft keys; tool offset value can be increased or decreased by 『Translation of tool rest』 soft key. Function and operation of "Tool compensation setup" under "Setup" function set are the same as those of "Tool compensation setup" under "Machining" function set.

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

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

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

Machining information: Cyclic switch display: Content of "Machining information" and "G instruction mode"

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

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

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

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

Machining information: Display and setup of machining statistics.

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

Reset relative coordinates: Set relative zero point during tool setting for ease of tool setting calculation.

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

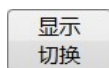
3.2.1.1 Machining set interface zoning

The interface of the first function set selected after startup is the same, called the default interface of the function set. After startup, press [[Machining]] function key to enter the default interface of "Machining" function set, as shown below.



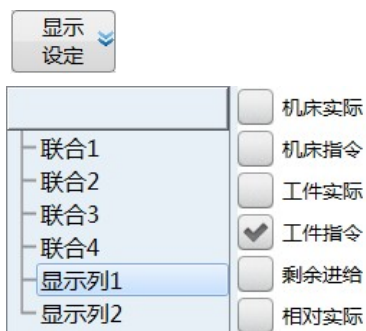
- (1) Area---Title bar
- (2) Area---Coordinates and graphics display window: Coordinate, graph and program display area.
- (3) Area---G code display area: Preview or display code of machining program.
- (4) Area---Input box: Enter information to be inputted in this column.
- (5) Area---Menu instruction bar: Operate system functions through function keys in the menu instruction 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 mode, program progress and workpiece quantity during machining.

3.2.1.2 Display switch of graphics and G code area



For display switch of graphics and G code areas (2) and (3), press 『Display switch』 soft key under the interface 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 coordinate graphics display area (2), press 『Display setup』 soft key under the interface 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 3.6.7.1).

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



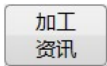
For joint coordinate display setup of coordinate graphics display area (2), press 『Display setup』 soft key under the interface to enter the lower-level menu, select "Joint 1-4" and set 4 coordinate contents in the "Joint coordinate" interface (for details, refer to "User setup" in 3.6.7.1).

3.2.1.5 Display switch of machining and commissioning information area



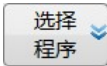
For display switch of machining and commissioning information area (6), press 「Alt」+「Left and right cursors」 on the MDI keyboard to display the following items and values successively: Machine tool actual, machine tool instruction, workpiece actual, workpiece instruction, remaining feed, relative actual,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

3.2.1.6 Display switch of machining information area



For display switch of machining information area (8), press 『Machining information』 soft key under the interface to switch G mode, 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 and edit and create new programs. Existing programs in system disk, USB flash disk and network disk are optional.

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

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



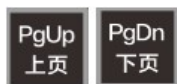
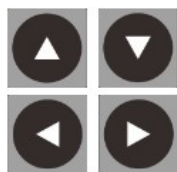
3.2.2.1 Select a program in the USB flash disk and load it as current machining program



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

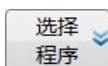
Attention: If error is reported while loading a program, press 「Reset」 to clear it and press 「↑」 to return to level 1 interface);

3.2.2.2 Select a program in the directory as current machining program



- Press 「Select program」 to enter the "Select program" sub-interface;
- Select soft keys of program source disk, namely soft keys 「System disk」, 「USB flash disk」, 「User disk」 and 「Network disk」, and enter corresponding program source disks;
- Press 「Cursor」 or 「PgUp/PgDn」 to select the file directory;
- Press 「Enter」 to activate the selected directory, enter the directory and display program files hereunder.
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor to program file name;
- Press 「Enter」 to load the selected program as current machining program and the interface returns to the previous menu and interface. After that, parts can be processed.

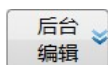
3.2.2.3 Exit from file directory:



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

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

3.2.2.4 The backstage edits current machining program

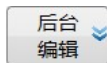


Current machining program cannot be edited when the program runs, but it can be edited using backstage edit function not under running status.

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

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

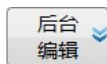
3.2.2.5 The backstage edits other programs than current machining program



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

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

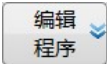
3.2.2.6 The backstage edits and creates new programs



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

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

3.2.3 "Program editing" sub-interface

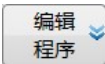


The "Edit program" sub-interface is mainly used to edit current machining program and create and edit new programs.

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



3.2.3.1 Edit current machining program

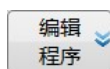


- Press 『Edit program』 soft key under the "Machining" function set and the cursor is in the editing area of current machining program, namely current machining program can be edited.
- After editing or modification, press 『Save file』 soft key and a prompt message Saved will be given, then return to the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

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

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

3.2.3.2 Create new programs



请输入文件名: O temp

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

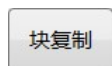
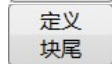
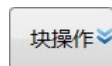
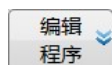
Note: After a new program is saved under "Machining" function set, it will be loaded as current machining program automatically.

3.2.3.3 Block operation

"Block operation" function is often used for copy, paste and other operations of multiple program segments. It defines initial segment and final segment of multiple program segments 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 status of "Backstage editing" function; under "Machining" function set, edit current machining program status of "Program editing" function; under "Program" function set, create program status of "Create" function.

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



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

3.2.4 "Verify program" sub-interface



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

Verification program is valid under automatic mode and single segment mode. Press 『Verify program』 soft key and working mode turns from "Automatic" to "Verification"

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



3.2.4.1 "Verify program" runs

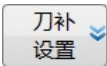
- Load programs under automatic mode;
- Press 『Verify program』 soft key and working mode turns to "Verification";
- Press [Cycle start] to verify programs. (Feedrate override can control verification speed)

3.2.4.2 Exit from "Verify program"



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

3.2.5 "Tool compensation setup" sub-interface



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

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

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

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

Input box →

自动加工 设置 程序 诊断 维护 MDI

刀号		X	Z	R	T
1	偏置	5.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000		
2	偏置	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000		
3	偏置	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000		
4	偏置	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000		
5	偏置	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000		

	机床实际	相对实际	工件实际	记录坐标
X	15.0000	15.0000	15.0000	15.0000
Z	2.4996	2.4996	2.4996	0.0000
C	0.0000	0.0000	0.0000	

\$1 修改成功，下次换刀时或重运行时生效!

刀架 试切 试切 记录X 记录Z 清除X 清除Z
平移 直径 长度 坐标 坐标 坐标 坐标

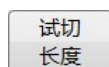
3.2.5.1 Direct input mode of tool compensation value



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

请输入第1号刀具的X: 0.0000

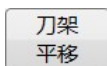
3.2.5.2 Trial cut input mode of tool compensation value



Tool setting is to determine the positional relationship between workpiece coordinate system and machine tool coordinate system. During tool setting for trial cut, measure diameter or length of workpiece so that the operator knows the position of the tool in the workpiece coordinate system (retract by translation). Then, the operator transmit this message to the system by 『Trial cut diameter』 and 『Trial cut length』 soft keys and the system reads the position of the tool in the machine tool coordinate system in order to determine the positional relationship between two coordinate systems.

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

3.2.5.3 Overall translation of tool offset



Specification of workpiece is difference. If only coordinates change, the relative position between tools remains unchanged, but all tools offset. "Translation of tool rest" can meet the requirements.

- Press 『Tool compensation setup』 soft key in the level 1 menu of machining function set to enter the sub-interface;
- Press 『Translation of tool rest』 to activate the input box
- Input “X offset \downarrow Space \downarrow Z offset” (such as 0.1 \downarrow 0.1)
- Press 「Enter」 to confirm input and X/Z offset of all tools in the tool offset table increases by the input value relatively.
- If format of input data is incorrect (if there is no space key), a prompt message that input data is invalid will be given

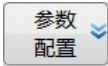
3.2.5.4 Exit from input activated state



During tool compensation MDI input, trial cut diameter input and trial cut length input, due to input errors (such as tool number axis is selected incorrectly) and other reasons, exit from input activated status and do not change the current activated compensation value.

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

3.2.6 "Parameter configuration" sub-interface



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

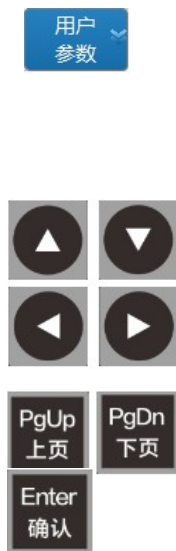
Generally parameter values can be inputted by keys on the panel or backed up and imported. This quarter introduces direct input by panel keys only. For parameter backup, import and other operations, refer to 9 Machine tool commissioning.

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

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

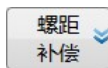


3.2.6.1 Direct input of parameter configuration value



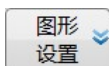
- Press 『User parameter』, 『Machine tool parameter』 and 『Axis parameter』 soft keys to enter different sub-interfaces;
- Press 『Enter password』 to activate the input box;
- Enter user password, such as "NC8";
- Press 「Enter」 to confirm;
- Press 「Cursor」 or 「PgUp/PgDn」 to select the value on the right of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

3.2.6.2 Direct input of pitch error compensation value



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

3.2.7 "Graphics setup" sub-interface



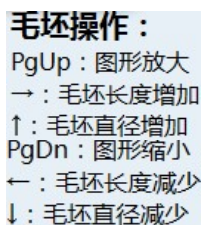
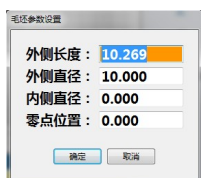
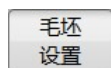
Select simulation display interface through cyclic switch of "Display switch" soft key in the level 1 menu under machining function set. This interface can display such information as workblank graphics, workpiece zero position, tool shape and tool path.

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

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



3.2.7.1 Graphics workblank setup and zoom



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

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

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

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

3.3 "Setup" function set interface and basic operation

3.3.1 "Setup" function set interface and function

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

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



Tool compensation Setup	Tool rest Translation	Trial cut Diameter Program	Trial cut Length	Coordinate System	Tool Life	Relative Reset	⇒
Tool Measurement Switch	Thread Repair Configuration						

- Tool compensation setup: Tool offset and wear value can be set under the sub-interface of this function, or tool offset can be set through 『Trial cut diameter』 and 『Trial cut length』 soft keys; tool offset value can be increased or decreased by 『Translation of tool rest』 soft key.
Function and operation of "Tool compensation setup" under "Setup" function set are the same as those of "Tool compensation setup" under "Machining" function set.
- Translation of tool rest: Press 『Translation of tool rest』 soft key to increase or decrease tool offset under the default interface of the setup function set.
Function and operation of "Translation of tool rest" under "Setup" default interface are the same as those of "Translation of tool rest" under "Tool compensation setup" sub-interface
- Trial cut diameter: Press 『Trial cut diameter』 soft key to set X axis offset of tool and clear tool wear under "Tool compensation setup" sub-interface.
Function and operation of "Trial cut diameter" under "Setup" default interface are the same as those of "Trial cut diameter" under "Tool compensation setup" sub-interface.
- Trial cut length: Press 『Trial cut length』 soft key to set Z axis offset of tool and clear tool wear under "Tool compensation setup" sub-interface.
Function and operation of "Trial cut length" under "Setup" default interface are the same as those of "Trial cut length" under "Tool compensation setup" sub-interface.
- Coordinate system: This function can be used to set values of the workpiece coordinate system through direct input, current value input and increment input mode.
Function and operation of "Coordinate system" under "Setup" function set are the same as those of "Coordinate system" under "Machining" function set.
- Tool life: This function can be used to set tool life management method and strategy.
- Reset relative coordinates: Reset relative coordinates X and Z under this interface.
- Tool measurement: Realize automatic measurement of new tools.

- Thread repair: Repair and process thread of new thread workpiece.
Some functions of "Setup" function set are the same as those of "Machining" function set. Functions introduced in "Machining" function set are not introduced in this quarter.
After startup, press **〔Setup〕** function key to enter the default interface of "Setup" function set, as shown below.

CNC		CH0		2019-07-29 15:58:08	
手动		加工		设置	
程序		诊断		维护	
MDI					
刀号		X	Z	R	T
1	偏置	5.3000	0.3000	0.0000	0
	磨损	1.0000	0.0000		
2	偏置	15.2033	0.3000	0.0000	0
	磨损	0.0000	0.0000		
3	偏置	15.2033	0.3000	0.0000	0
	磨损	0.0000	0.0000		
4	偏置	0.3000	0.3000	0.0000	0
	磨损	0.0000	0.0000		
5	偏置	0.3000	0.3000	0.0000	0
	磨损	0.0000	0.0000		
		机床实际	相对实际	工件实际	记录坐标
X		0.0000	0.0000	0.0000	-----
Z		0.0000	0.0000	0.0000	-----
C		0.0000	0.0000	0.0000	
\$1					
↑		刀补	刀架	试切	试切
		设置	平移	直径	长度
				坐标系	刀具
					寿命
					相对
					清零
		→			

3.3.2 "Coordinate system" sub-interface

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

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

Where, 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.

坐标系



PgUp
上页

PgDn
下页

CNC		CH0		2019-07-30 11:17:06	
自动		加工		设置	
程序		诊断		维护	
MDI					
外部零点偏移		相对坐标系		机床实际	
相对实际					
X	0.0000 毫米	X	0.0000 毫米	X	-20.0000
Z	0.0000 毫米	Z	0.0000 毫米	Z	0.0000
C	0.0000 度	C	0.0000 度	C	0.0000
G58		G59		G54.1 P1	
G54.1 P2					
X	0.0000 毫米	X	0.0000 毫米	X	0.0000 毫米
Z	0.0000 毫米	Z	0.0000 毫米	Z	0.0000 毫米
C	0.0000 度	C	0.0000 度	C	0.0000 度
\$1					
↑		当前	增量	全部	
		输入	输入	清零	
		→			

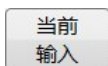
3.3.2.1 Direct input mode of coordinate value

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

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

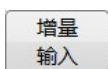
3.3.2.2 Input mode of current value

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



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

3.3.2.3 Input mode of increment value



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

- 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 『Increment input』 to activate the input box;
- Input increment value of the coordinates in the input box;
- Press 「Enter」 to confirm and exit from the input box;
- To abandon input, press 「Reset」 and input is invalid, and exit from the input box

3.3.3 "Tool life" sub-interface

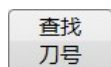


Under the "Tool life" sub-interface (as shown below), 5 tool life criteria can be set by "Setup": Installation times, cutting time, cutting length, cutting power consumption and spindle revolution. To enable this function, set channel parameter 040130 under "Maintain"--> "Parameter setup" menu and parameter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

When the system can reach the designated value according to an index, judge tool life early warning or alarm status, or use the weighted sum of the selected indexes as a criterion for tool life. This strategy is selected by 『Alarm strategy』 soft key under the submenu.



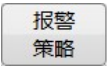
3.3.3.1 Tool life index setup



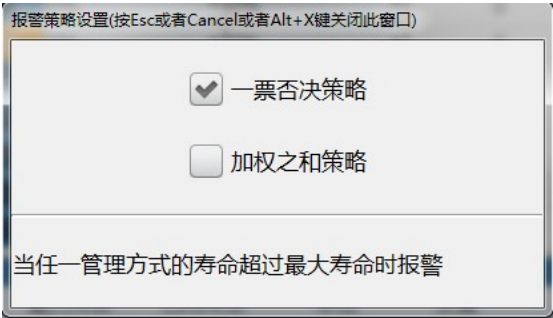
- Press 『Tool life』 soft key under the "setup" interface to enter the sub-interface;
- Press 「Cursor」 or 「PgUp/PgDn」 or 『Search tool number』 soft key to move the cursor to the "Setup" column of the selected tool;
- Press 「Enter」 and the life index setup window will pop up (as shown below);
- Select management mode, life index and weight by cursor;
- Press 「Enter」 to activate input;
- Press 「Enter」 to confirm the inputted values;
- Select "Enter" or "Cancel" by cursor;
- Press 「Enter」 to confirm and exit from the setup window.



3.3.3.2 Tool life alarm strategy setup



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



3.3.4 "Tool measurement sub-interface"



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

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



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

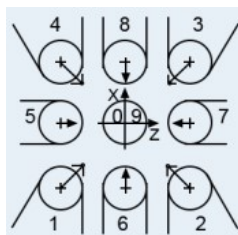
1. Preparation for measurement

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

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

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

- Press 「Cursor」 to select parameters
- Press 「Enter」 to activate the input box
- Input data
- Press 「Enter」 to confirm. Including
 - Length/width of measuring instrument: Length and width value of measurement instrument may not be very accurate, but they have certain influence on the maximum value of offset of tools. When tool offset is greater than 50mm and this function is used for the first time, collision should be avoided.
 - Calibrate tool nose direction: Calibrate tool nose direction of standard tool. Tool nose direction in this quarter is mounting direction number of tool nose, which affects feed and retract direction of standard tool during automatic tool setting (as shown in the left figure).
 - Measurement times: Times that the tool touches the measuring instrument during automatic tool setting, no more than 5 times.
 - Measurement speed: Speed when the tool touches the measuring instrument for the first time during automatic tool setting
 - Trigger speed: Speed when the tool touches the measuring instrument when the value is read. The distance is 2mm by default.



2. Tool calibration

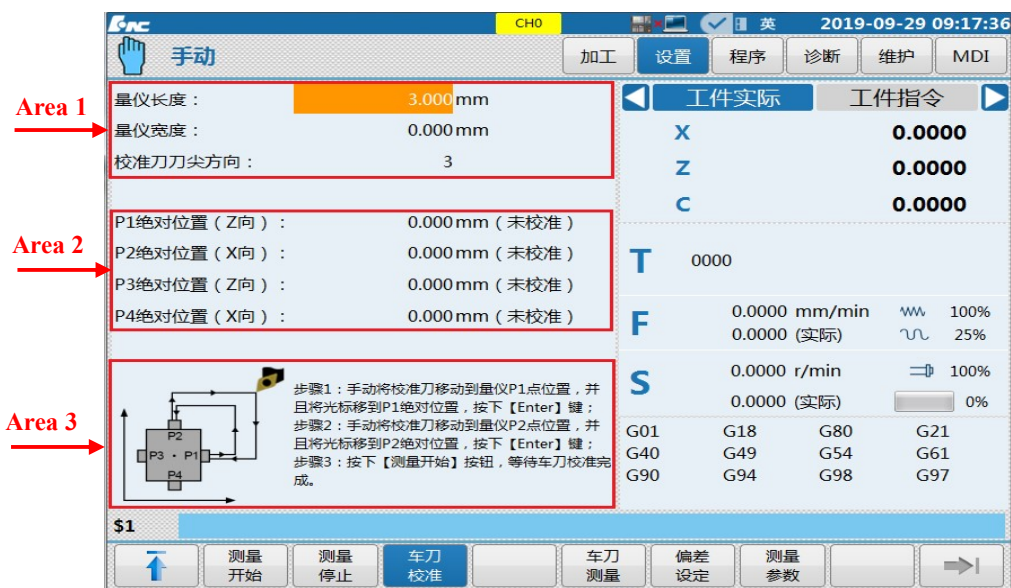
Standard tool calibration function can determine the positional relationship between standard tool and measuring instrument. The tool calibration process includes manual calibration and automatic calibration.

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

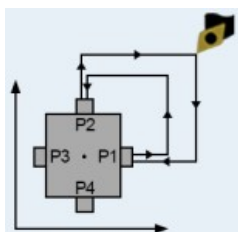
The below area 1 is used to set size of measuring instrument and tool nose direction number of standard tool movement direction. If this parameter has been set in "Measurement parameter", the parameter value is displayed accordingly (for ease of inspection).

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

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



1) Manual calibration of "Standard tool"



- Set working mode as **【Manual】**
 - Press 「Cursor」 to select the calibration position, such as "P1 absolute position";
 - Move the standard tool to P1 point (as shown in the left figure);
 - Press 「Enter」 to enter and display "P1 absolute position". This value is the Z axis coordinate value of the standard tool in the machine tool coordinate system (namely Z axis coordinate value of P1 point of the measuring instrument in the machine tool coordinate system).
 - Likewise, enter "P2 absolute position", namely X axis value;
- (The tool corresponding to every tool number should be calibrated just at 2 points. e.g.: 3# tool corresponding to tool nose direction number should be calibrated at P1 and P2 points.)

2) Automatic calibration of "standard tool"

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

3. Automatic measurement

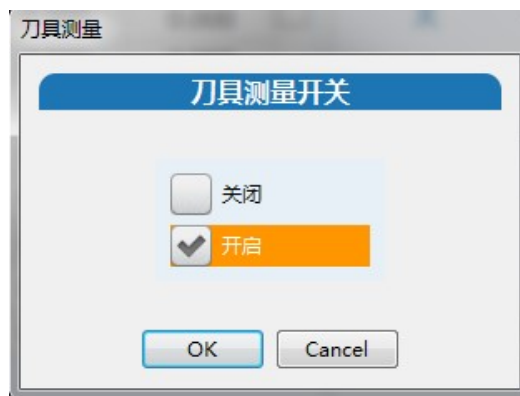
1) Tool selection

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



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

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



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

2) Automatic tool setting

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

- Set working mode as **【Automatic】** or **【Single segment】** ;
- Complete preparation for measurement under the "Tool measurement" interface;
- Press 『Start measurement』 soft key under the "Tool measurement" interface;
- Press **【Cycle start】** and every tool whose measurement function has been enabled will be measured automatically.

Attention:

- Press **【Cycle start】** and a prompt message: Execute G code and wait for user intervention may occur when the measurement program runs. After the operator confirms that there is no measurement error, press **【Cycle start】** and the measurement program will continue running.
- When tool offset is greater than 50mm, collision between the tool and the measuring instrument should be avoided.

4. Offset setup

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

Press 『Offset setup』 soft key under the "Tool measurement" interface to enter the "Offset setup" sub-interface (as shown below). Where, area 1 realizes "Offset setup", area 2 realize standard tool setting and area 3 displays offset of the workpiece zero point in X axis and Z axis



Specific steps for "Offset setup":

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

Attention:

Press **【Start measurement】** before lathe tool calibration and lathe tool measurement. If an emergency occurs in the measurement process, press **【Feed hold】** to stop the program and **【Stop measurement】** to end the measurement process. Whereas the measurement process cannot be interrupted, press **【Stop measurement】** to finish measurement.

3.4 "Program" function set interface and basic operation

3.4.1 "Program" function set interface and function



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

System Disk	USB flash disk	Network disk	User Disk	Search	Setup Sign Copy	Copy	Paste	⇒
Create Program	Create Contents	Delete	Rename Name	Name Rank	Time Rank	Writable	read only	

- System disk, USB flash disk, network disk: Source disk or target disk of program during program file management.
- New program: Create new programs with the same functions of 『Create』 soft key under the 『Machining』 function set.
- Search: Search programs in the source disk of program files.
- Copy, paste: Copy programs in the source disk of program files and paste them to the target disk.
- Delete: Delete program files in the source disk of programs.
- Set mark: Mark programs in the source disk of programs in order to copy or paste multiple programs.
- Rename: Rename programs in the source disk of programs.
- Sort by name, sort by time: Sort programs in the source disk of programs by alphabetical order in name or modification time sequence.
- Create a new directory: Create a new program directory in the target disk of programs.
- Writable, readable: Set program files as writable or readable.
- User disk: This system divides storage card (CF card) into operating system disk area, CNC system disk area and user disk area, among which user disk is used for backup, storage and other uses and is unrelated to operation of machine tool.

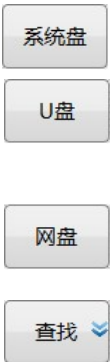
After startup, press 『Program』 function key to enter the default interface of "Program" function set, as shown below.

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



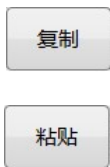
3.4.2 Management of files in system disk, USB flash disk and network disk

3.4.2.1 Management program search



- Select the areas where the program searched may be located under the "Program" default interface, namely 『System disk』, 『USB flash disk』 and 『Network disk』;
- If the program to be searched is in the file directory, press 『Enter』 to open it;
- Press 『Search』 soft key, activate the input box and a prompt message Input the file to be searched will be given;
- 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 『Search』 or 『Cursor』 and 『PgUp/PgDn』 under the "Program" default interface to select the program to be copied and pasted;
- Press 『Copy』 soft key and the input box will give a prompt message: Copy succeeds;
- Press 『System disk』, 『USB flash disk』 and 『Network disk』 to select the target areas
- If the program to be pasted is in the file directory, select the file directory and press 『Enter』 to open it;
- Press 『Paste』 soft key and the input box will give a prompt message: Paste succeeds;

3.4.2.3 Program deletion

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

3.4.3 Create new programs

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

Note 1: "Machining" function set and "Program" function set have 『Create』 function.

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

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

3.4.4 Program rename

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

3.4.5 Program mark setup

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

3.4.6 Programs are sorted by name and time



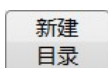
- Press 『System disk』, 『USB flash disk』 and 『Network disk』 under the "Program" default interface to select the areas where a program is marked;
- If the program to be marked is in the file directory, select the file directory and press 「Enter」 to open it;
- Press 「Cursor」 or 「PgUp/PgDn」 to move the cursor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

3.4.7 Program write/read setup



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

3.4.8 Create a new catalogue



- Press 『System disk』, 『USB flash disk』 and 『Network disk』 under the "Program" default interface to select the areas where a new directory is built;
- 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 built;
- Press 『→』 under the "Program" default interface to switch to extension menu page of the "Program" interface;

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

3.5 "Diagnosis" function set interface and basic operation

3.5.1 "Diagnosis" function set interface and function



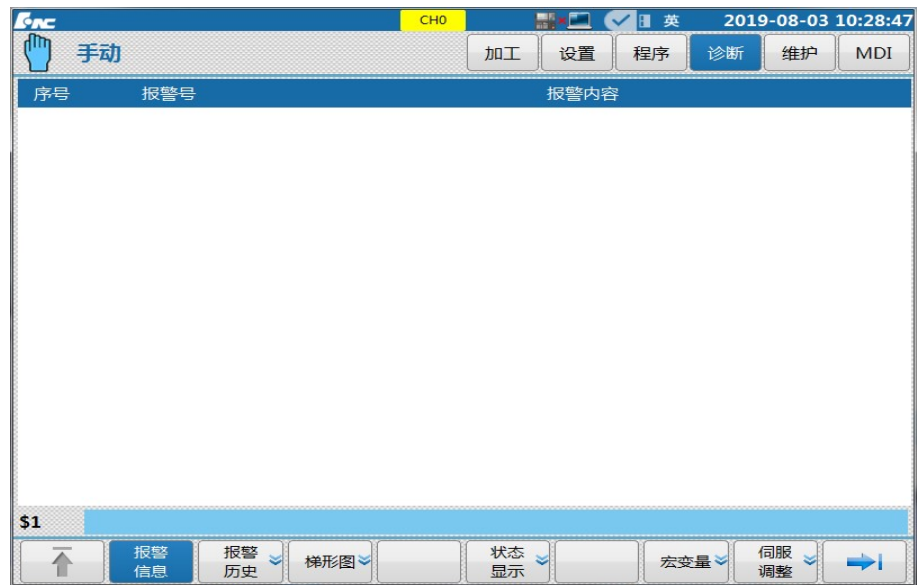
"Diagnosis" function set mainly integrates such functions as fault alarm, fault diagnosis and machine tool commissioning. Level 1 main menu and level 1 extension menu of soft key function of "Diagnosis" function set are shown below.

Alarm Information	Alarm History	Ladder diagram	Display Status	Axis monitoring	Macro variable	Servo Adjustment	⇒
Self-inspection	Log	Fault Video Program	Screw Load	QR Code			

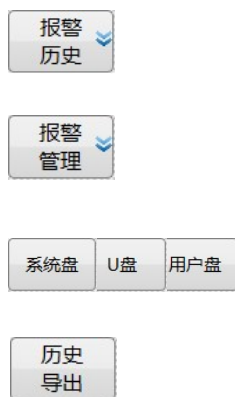
- Alarm message: Display current alarm message
- Alarm history: Save information of recent alarms and export historical alarms to USB flash disk and system disk using "History export" soft key under the lower-level menu "Alarm management".
- Ladder diagram: Track or record PLC signal by period or trigger; configure operation, inspection and valuation method of PLC through ladder diagram option.
- Status display: Display and view the status of all registers.
- Axis monitoring: Display and view status of axis, namely coordinate position, pulse value, breakpoint position, compensation value and load current of axis.
- Macro-variable: Display and view the value of macro-variables.
- Servo adjustment: Adjust servo parameters to realize commissioning of optimal speed, position, roundness, tapping, noise and gantry synchronization of machine tool through sampling result.
- Self-inspection: Record relevant health indexes during long-term operation of machine tool in order to prejudge health condition of machine tool.
- Log: Record the operation process completed by the system when the machine tool runs.
- Fault video: Record related data of fault within 10s before a fault occurs. Related data of fault can be preset as position, speed and current of all axes.
- Screw load: Record operation frequency of all areas of screw during long-term operation of machine tool in order to judge screw wear and other statuses.
- QR code: This function can be used to acquire data of machine tool such as machining status, alarm history and fault diagnosis, upload them to cloud data center and realize full life circle management of machine tool.

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

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

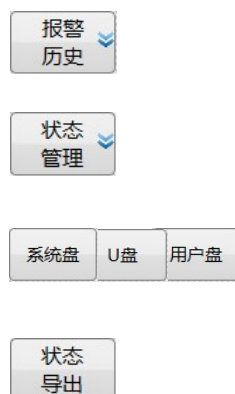


3.5.2 Alarm history export



- Press 『Alarm history』 soft key under the "Diagnosis" function set default interface;
- Press 『Alarm history』 soft key under the 『Alarm history』 sub-interface;
- Press 『Alarm management』 soft key;
- Select 『System disk』, 『USB flash disk』 and 『User disk』 soft keys;
- Press 『History export』 soft key to export corresponding information to the selected disk.

3.5.3 Status record export



- Press 『Alarm history』 soft key under the "Diagnosis" function set default interface;
- Press 『Status record』 soft key under the "Alarm history" sub-interface;
- Press 『Status management』 soft key;
- Select 『System disk』, 『USB flash disk』 and 『User disk』 soft keys;
- Press 『Status export』 soft key to export corresponding information to the selected disk.

3.5.4 "Ladder diagram" sub-interface

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



Attention: This interface should be entered with workshop manager permission or above. For specific permission content, please refer to 3.6.4.

3.5.4.1 Ladder diagram monitoring

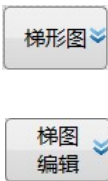


This function is used for monitoring system PLC

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

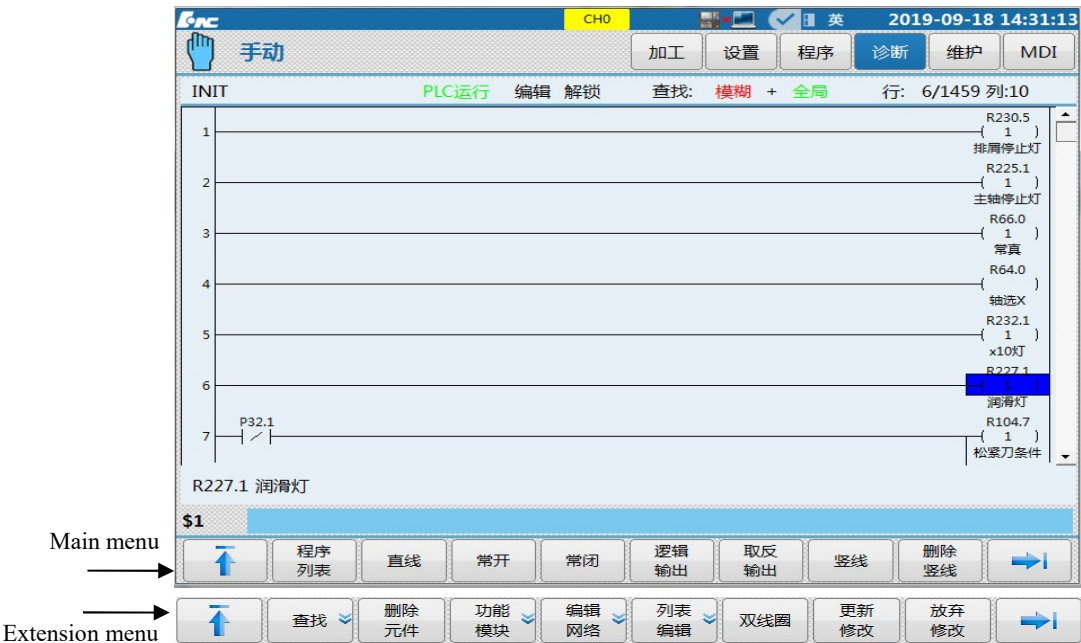


3.5.4.2 Ladder diagram editing



This function is used for PLC modification

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



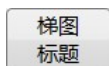
3.5.4.3 Ladder diagram information



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

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

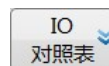


1. Ladder diagram title

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

2. Symbol table

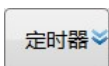
Chinese name setup table of X, Y, F, R and G registers is mainly used to view point location information.

3. IO comparison table

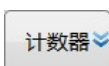
2.4 version uses I register and Q register to replace output of X register and Y register in PLC. If different machine tools are configured, only a set of PLC can be used in order to reduce problems of PLC. While configuring different machine tools, PLC needs not modified but IO comparison table with change of IO point location. Different XY registers correspond to IO registers and IO point location remains unchanged in PLC.

4. K parameter

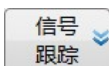
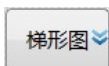
Function and P parameter are the same, but 0 or 1 point location can be modified directly in this interface.

5. Timer and counter

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

**6. Alarm setup**

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

3.5.4.4 Ladder diagram signal tracking

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

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



For example, take samples of “Run allowed R10.0” and a flash point location R72.6, fill two registers in corresponding box, click on Start sampling and change of two point locations will be displayed by status of histogram.



Press Zoom out and Zoom in to control the interval time between two sampling points. e.g.: the below figure is the scaled-down version of the above figure.



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



The parameter interface can control enable conditions for sampling function/ e.g.: start condition is changed to trigger start in the above example and trigger condition is changed to X483.7, click on Start and sampling will not be started. Press X483.7 key once to start sampling. If stop is changed to trigger stop and stop condition is X483.6, press stop key and X483.6 key to stop sampling.



3.5.5 Display of register status and macro-variable value

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

- Press 『Diagnosis』 function key to enter the default interface of function set;
- Press 『Status display』 or 『Macro-variable』 soft key to display status of X, Y, F, G, R and B registers or value 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. Level 1 main menu and level 1 extension menu of soft key function of "Maintain" function set are shown below.

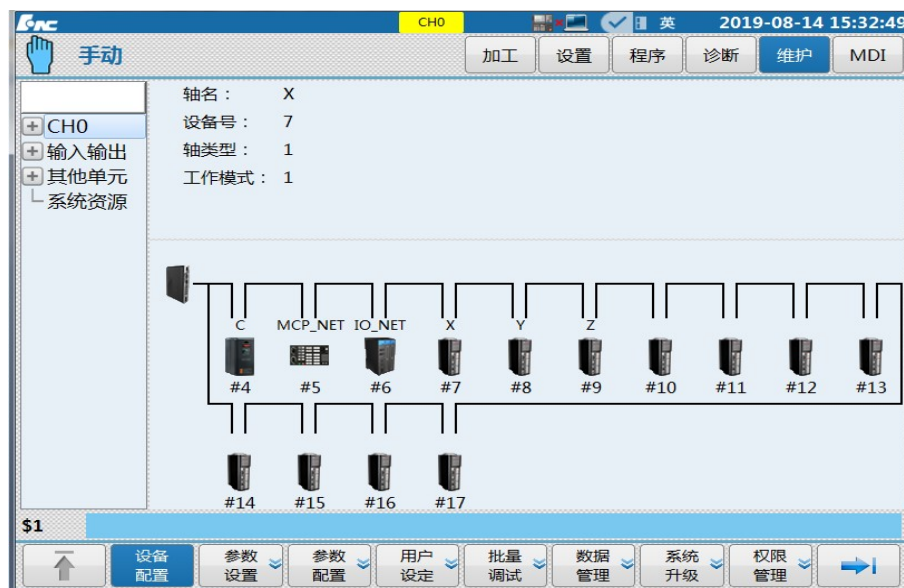
Device configura tion	Paramete r setup	Parameter configura tion	User setup	Batch commissi oning	Data managem ent	System upgrade	Permissi on managem ent	⇒
Space compens ation	Time setup	Process package	Machine tool informati on	System informati on	Register			

- Device configuration: View number of driver, I/O, panel and other hardware devices and connection sequence of bus;
- Parameter setup: This interface includes all system parameters, under which NC parameter, machine tool user parameter, channel parameter, coordinate axis parameter, error compensation parameter, device interface parameter and data sheet parameter can be set;
 - ✧ NC parameters: Common parameters of the CNC system (such as interpolation period and resolution, and so on).
 - ✧ Machine tool user parameters: Common parameters relating to machine tool and user (type of measuring instrument and maximum number of channels).
 - ✧ Channel parameters: Common parameters of CNC system of all channels (such as small line segment parameter)
 - ✧ Axis parameter: Related parameters of logical axis (electronic gear ratio and 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: Interface parameters relating to physical device and system connection (such as device type and device ID)
 - ✧ Data sheet: Data sheet storing corresponding compensation values of error compensation parameters.
- Parameter configuration: Common user parameters under this interface, which are classified according to application type for ease of user setup; Parameters under this interface include user parameter, axis parameter, pitch compensation, IO device, local spindle device, bus axis device and function parameter;
- User setup: Setup relating to user application. This interface includes display setup, P parameter, M code, PLC switch, communication setup, personalized setup and closed-loop switch;
- Batch commissioning: Loading and backup of PLC, parameter, fixed cycle, G code and other files;
- Data management: Loading and backup of various types of data;
- System upgrade: System upgrade and backup;
- Permission management: Setup of administrators of different permissions.

Different permissions have certain influence on the structure of interfaces and menus;

- Space compensation: Space error compensation setup;
- Time setup: System time setup;
- Process package: Load or back up process package file;
- Machine tool information: Edit or display machine tool information;
- System information: Display information of this system;
- Registration: Display registration code and related information of the machine tool;

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



3.6.2 Parameter setup

- 1) Parameter selection
 - Press 『 Device configuration 』 , 『 Parameter classification 』 and 『Parameter setup』 to select parameter set;
 - Press 「 Left and right cursors 」 to move the cursor and select parameter classification column or parameter setup column
- 2) Parameter input activation
 - When the cursor moves to parameter classification column, press 「 Enter 」 to open categories of the selected parameter
 - When the cursor moves to parameter setup column, press 「 Enter 」 to activate "Input box"
- 3) Parameter input
 - After input box is activated and corresponding values are inputted, press 「 Enter 」 to confirm input.
- 4) Exit from input
 - To abandon input after input box if activated, press 「 Reset 」 to exit from input and retain the original value.

Parameter classification column

NC参数

机床用户参数

通道参数

通道0

通道1

通道2

通道3

坐标轴参数

误差补偿参数

设备接口参数

数据表参数

Parameter setup column

参数号	参数名	参数值	生效方式
040000	通道名	CH0	重启
040001	X坐标轴轴号	0	重启
040002	Y坐标轴轴号	-1	重启
040003	Z坐标轴轴号	2	重启
040004	A坐标轴轴号	-1	重启
040005	B坐标轴轴号	-1	重启
040006	C坐标轴轴号	-2	重启
040007	U坐标轴轴号	-1	重启
040008	V坐标轴轴号	-1	重启

最大值：127

默认值：-1

最小值：-3

说明：用于配置当前通道内X轴的轴号，实现通道进给轴与逻辑轴之间的映射。
0~127：指定当前通道进给轴的轴号。
-1：当前通道进给轴没有映射逻辑轴，为无效轴。
-2：当前通道进给轴保留C/S轴切换，切换后在位置方式下轴类型为旋转轴
-3：当前通道进给轴保留C/S轴切换，切换后在位置方式下轴类型为直线轴

\$1

请输入参数值: 0

Input box

3.6.3 Parameter validation method and operation

There are 5 validation method for parameters of this system: immediate, save, reset, restart and curing (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 validated immediately.

2) Save to validate parameters

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

3) Reset to validate parameters

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

4) Restart to validate parameters

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

Notices:

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

3.6.4 Classification and switch of management permission

Specific to different application requirements for CNC machine tool, this system sets 5 operation permissions, operator, workshop manager, machine tool manufacturer, CNC manufacturer and system administrator. Management functions of various permissions are roughly as follows:

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

"CNC manufacturer": Product manufacture and quality inspection with permissions for system upgrade, system parameter setup, PLC program editing and limited-time shutdown setup.

"Machine tool manufacturer": Machine tool commissioning with permissions for partial system parameter modification, error compensation data entry and limited-time shutdown setup.

"Workshop manager": Machining commissioning with permissions for partial user parameter modification, editing parts program and editing tool compensation data.

"Operator": Machining operation with permissions for editing tool compensation data and selecting program.

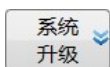
Operation function \ Permission type		System administrator	System manufacturer	Machine tool manufacturer	Workshop manager	Operator
Parameter	User parameter	Yes	Yes	Yes	No	No
	Machine tool parameter	Yes	Yes	Yes	No	No
	Axis parameter	Yes	Yes	Yes	No	No
Quantity	Pitch compensation	Yes	Yes	Yes	No	No
Classification	I/O device	Yes	Yes	No	No	No
	Local spindle device	Yes	Yes	No	No	No
Yes	Bus axis device	Yes	Yes	No	No	No
	Function parameter	Yes	Yes	Yes	No	No
System upgrade		Yes	Yes	No	No	No
Permission management		Yes	Yes	Yes	Yes	Yes
Batch commissioning		Yes	Yes	Yes	No	No
User setup (except display setup)		Yes	Yes	Yes	Yes	No
Data management		Yes	Yes	Yes	Yes	No
Space compensation		Yes	Yes	Yes	No	No
Time setup		Yes	Yes	Yes	No	No
Process package		Yes	Yes	Yes	No	No
Register		Yes	Yes	Yes	No	No
Alarm history		Yes	Yes	Yes	Yes	No
Ladder diagram	Ladder diagram monitoring	Yes	Yes	Yes	Yes	No
	Ladder diagram editing	Yes	Yes	Yes	No	No
	Ladder diagram information	Yes	Yes	Yes	No	No
	Signal tracking	Yes	Yes	Yes	No	No
Status display		Yes	Yes	Yes	Yes	No
Macro-variable		Yes	Yes	Yes	Yes	No
User macro		Yes	Yes	Yes	Yes	No
Servo adjustment		Yes	Yes	Yes	No	No
Log clearing		Yes	No	No	No	No

Program Function Set	Select program, search, sort	Yes	Yes	Yes	Yes	No
	Verify, any line	Yes	Yes	Yes	Yes	Yes
	Reset relative coordinates	Yes	Yes	Yes	Yes	Yes
	Display mode, track switch	Yes	Yes	Yes	Yes	Yes
	Machining statistics	Yes	Yes	Yes	Yes	Yes
	Machining optimization	Yes	Yes	Yes	Yes	No
	Program editing	Yes	Yes	Yes	Yes	No
	Read only, writable attribute setup	Yes	Yes	Yes	Yes	No
	User macro	Yes	Yes	Yes	Yes	No
Setup Function Set	Tool compensation	Yes	Yes	Yes	Yes	Yes
	Coordinate system	Yes	Yes	Yes	Yes	Yes
	Workpiece measurement	Yes	Yes	Yes	Yes	Yes
	Magazine tool, tool life setup	Yes	Yes	Yes	Yes	No
	Broken tool detection	Yes	Yes	Yes	Yes	No
	Automatic tool setting	Yes	Yes	Yes	Yes	No
Machining Function Set	Parameter configuration	Yes	Yes	Yes	No	No
	User macro	Yes	Yes	Yes	Yes	No
	Edit program	Yes	Yes	Yes	Yes	No
	Select program (except "System disk")	Yes	Yes	Yes	Yes	No
	Other operations	Yes	Yes	Yes	Yes	No

This system can set permission of "Operator" or "Workshop manager" as default permission through parameter 000359 in NC parameter list and other permissions can be switched through entering a password after startup. Permission switch is as below:

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

3.6.5 System upgrade



"System upgrade" function is available for system manufacturer only. Thus, permissions should be set under the "Maintain" function set after startup (such permissions are not saved only after startup).

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



- Press 『Window switch』 soft key to select the "Upgrade selection" window above 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 user disk);
- Press 『Window switch』 soft key to select the upgrade patch file source selection window below the interface (the default upgrade source disk is USB flash disk);
- Press 「Up and down cursors」 to select the upgrade patch file (upgrade patch file name must be suffixed with . BTF);
- Press 「Enter」 to confirm and start upgrade

Notices:

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

3.6.6 Data management



The data management interface can load/back up parameters, PLC, fixed cycle, log, compensation, oscilloscope and other files. With load/back up system parameter files as an example, other files are loaded/backed up according to the same operation steps (except error compensation file).

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



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



- To load data in USB flash disk or user disk to system disk, press 『Window switch』 soft key to move the red box to USB flash disk or user disk below 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 data file;
 - Press 『N』 or 『Reset』 to abandon loading data file.
- To back up data in system disk in USB flash disk or user disk, press 『Window switch』 soft key to move the red box to USB flash disk or user disk above the above figure;
 - Press 『Enter』 to open the file directory and press 『Cursor』 to select the data file to be backed up;
 - Press 『Backup』 soft key and the input box gives a prompt message "Whether to back up the selected file?"
 - Press 『Y』 to back up data file;
 - Press 『N』 or 『Reset』 to abandon backing up data file.

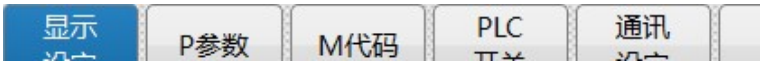
Attention Power failure is strictly prohibited during backup or loading.

3.6.7 User setup

用户
设定

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

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

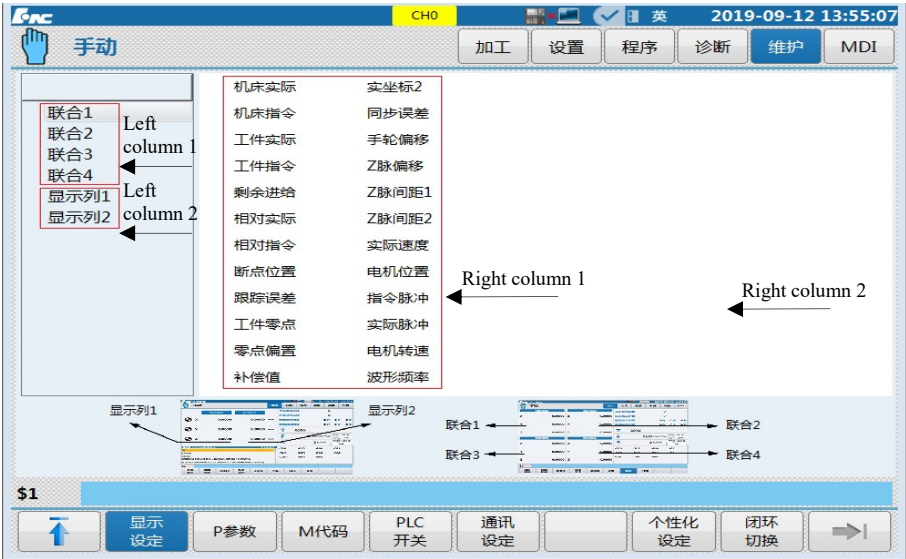


3.6.7.1 Display setup

显示
设定

It is introduced in the machining interface that press 『Display switch』 soft key under the "Machining" function set for cyclic switch among 4 interfaces, "Big character coordinate + program", "Joint coordinate", "Graph+ program" and "Program", among which "Big character coordinate + program" and "Joint coordinate" interfaces can be set here.

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



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

手动		加工	
机床实际		机床指令	
X	0.0000	X	0.0000
Z	0.0000	Z	0.0000
C	0.0000	C	0.0000
剩余进给		跟踪误差	
X	0.0000	X	0.0000
Z	0.0000	Z	0.0000
C	0.0000	C	0.0000

Joint coordinate display

手动		加工	
实坐标2		相对指令	
X	0.0000	0.0000 mm	
Z	0.0000	0.0000 mm	
C	0.0000	0.0000 deg	
程序名 ..\prog\O1235 0 /14			
%1234			
M03 S200			
N1 T0101			
G00 X30 Z10			
N3 X20 Z2			

Big character coordinate + program display

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

3.6.7.2 Set "P parameter"



P parameter is a parameter relating to machining and system operation. All PLC subprogram switches and PLC judgments 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 parameters after machine tool user parameter 010300 and correspond to the same memory address together with such user parameters. 『P parameter』 soft key is a shortcut key.

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

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

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

3.6.7.3 Set "M code"



M code table is setup of M codes of system and it is mainly used to set judgments such as M code validation method, validation sequence when M codes and G00 are in the same line and whether M codes are identified when any line is scanned.

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

mainwindow

CH0

 手动

加工

设置

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

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

M3 means positive rotation of spindle 1;
M4 means negative rotation of spindle 1;
M5 means stop of spindle 1;

a) Front or rear

When M code and G code are on the same line, to validate M code earlier, M code should be set as front. To validate it latter, it should be set as rear. To validate M code and G code simultaneously, they should be set as synchronous.

b) Any line scanning mode

Scanning mode refers to before the system runs a program, scanning all mode information in front of any line and the designated line inherits such information as coordinates, tool and compensation from front statement.

Scanning mode includes scanning without Z axis return and scanning with Z axis return, which are controlled by parameter 040113. It will take a long time to scan mode of a very large program. Meanwhile, subprogram scanning is supported.

Non-scanning mode refers to that the system does not calculate mode information in front of any line and it is default status of the system.

Note: 040113 parameter value

0: Non-scanning mode (default)

1: Scanning, without Z axis return

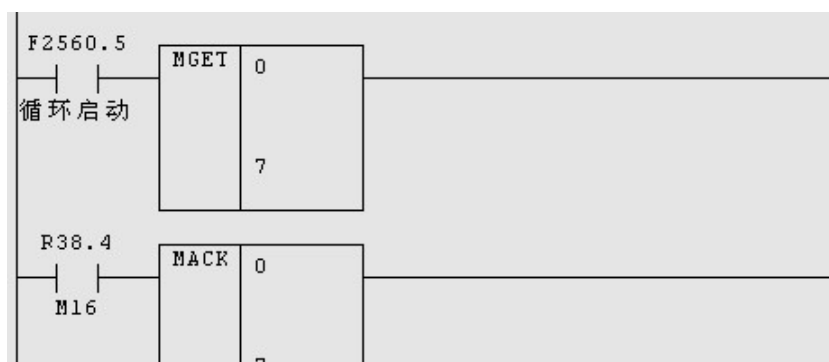
2: Scanning, with Z axis return

c) G00 synchronization

It should be noted that synchronization of M instruction and G code on the same line is synchronous attribute setup of front and rear columns, but it is a special situation that M instruction and G00 are on the same line and it should be set separately.

With M7 as an example, when M7 and G00 are on the same line, it is synchronous M code. When M7 and G01 are on the same line, it is front M code.

Attention, M code with G00 synchronous attribute should be immediately answered in PLC. In order to enable wait function of M7 when it is on the same line with G01, plc should be modified, as shown below



d) Spindle control

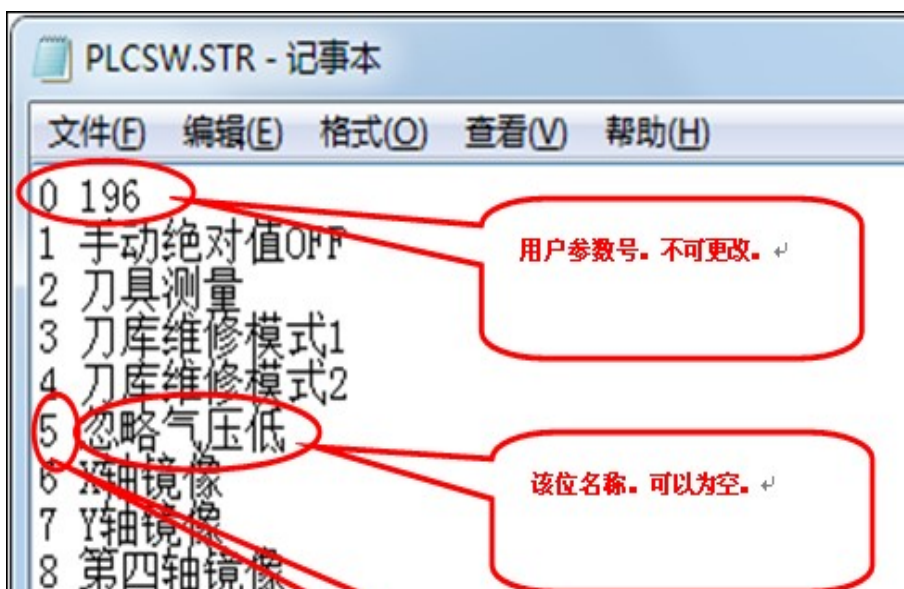
When the spindle undergoes automatic C/S switch (position mode/speed mode), PLC program should define axis name and axis number switched as well as start and stop of axis through M code. This setup can mark M code defined in PLC.

Attention: When there are multiple spindles, be sure to mark M codes controlled by newly increased spindles.

3.6.7.4 Set "PLC switch"



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



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



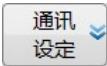
Press "Maintain→ User setup→ PLC switch" menu to enter the PLC switch function interface and operate the designated P parameter by bit, as shown below:



索引号	名称		索引号	
1	安全门锁有效	<input type="checkbox"/>	17	夹具1检测有
2	气压检测有效	<input type="checkbox"/>	18	夹具2检测有
3	气压报警停程序	<input type="checkbox"/>	19	测量头行程限
4	MST.LOCK	<input type="checkbox"/>	20	主轴G01保护
5	手摇急停有效	<input type="checkbox"/>	21	主轴油冷节能
6		<input type="checkbox"/>	22	主轴过载保护
7		<input type="checkbox"/>	23	X轴负向键回
8	手轮中断	<input type="checkbox"/>	24	
9	第四轴有效	<input type="checkbox"/>	25	
10	四轴无松紧到位	<input type="checkbox"/>	26	
11	进给倍率限制100	<input type="checkbox"/>	27	12 TOOL

Effective setup: Protect “ON” and “OFF” menus, which can be operated only through pressing "Effective setup" menu;
ON: Set bit of focus as ON;
OFF: Set bit of focus as OFF;
Setup result is saved in the designated user P parameter. As shown below, user P parameter in this example is designated by PLC.SW.STR file.

3.6.7.5 Communication setup



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

Attention: Network can be connected only after NC parameter 000050 【Whether to start network】 is opened

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 segment,

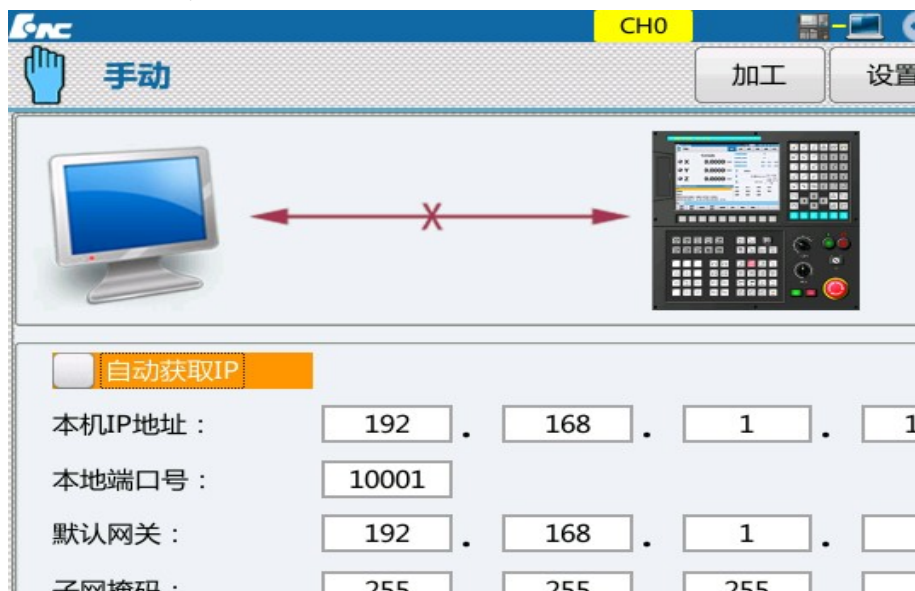


ip of the CNC system is 192.168.1.113 by default. ipv4 of the computer can be set as 192.168.1.XXX during connection and default gateway and subnet mask of the computer are consistent with those of the CNC system. Specific steps:

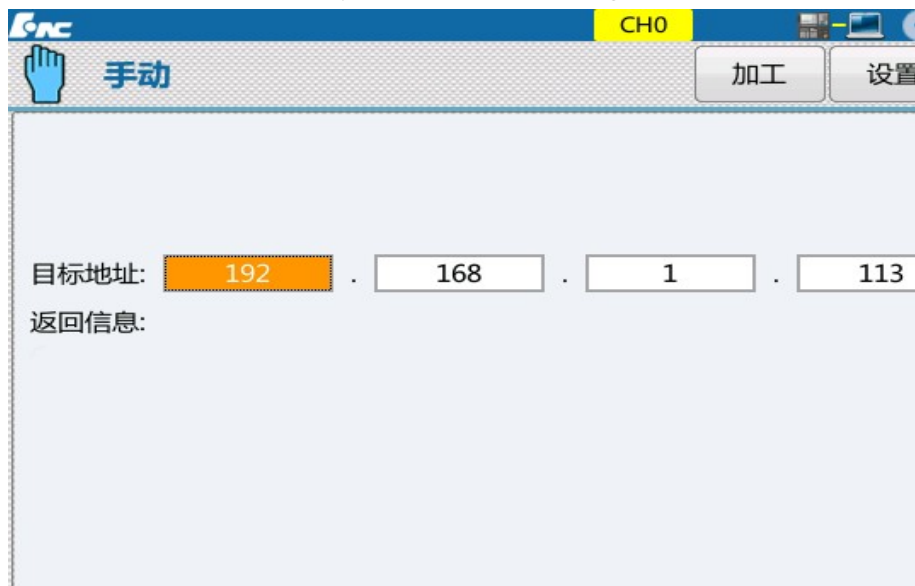
- Press 『User setup』 soft key under "Maintain" function set to enter the sub-interface;
- Press 『Communication setup』 soft key to enter the communication setup sub-interface;
- Press 『Local』 soft key to enter the "Local" connection sub-interface (as shown below);
- Move the cursor to where "Server IP address" is set and set default

IP address of the system, namely 192.168.1.113.

- Move the cursor to "Local port number", "Default gateway" and "Subnet mask" of the system and set them as "Local port number", "Default gateway" and "Subnet mask" of the computer.
- Connect Internet access of computer and that of system IPC using network cable. It should be noted that it cannot be connected to M3 of system IPC or EtherCAT Internet access.



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



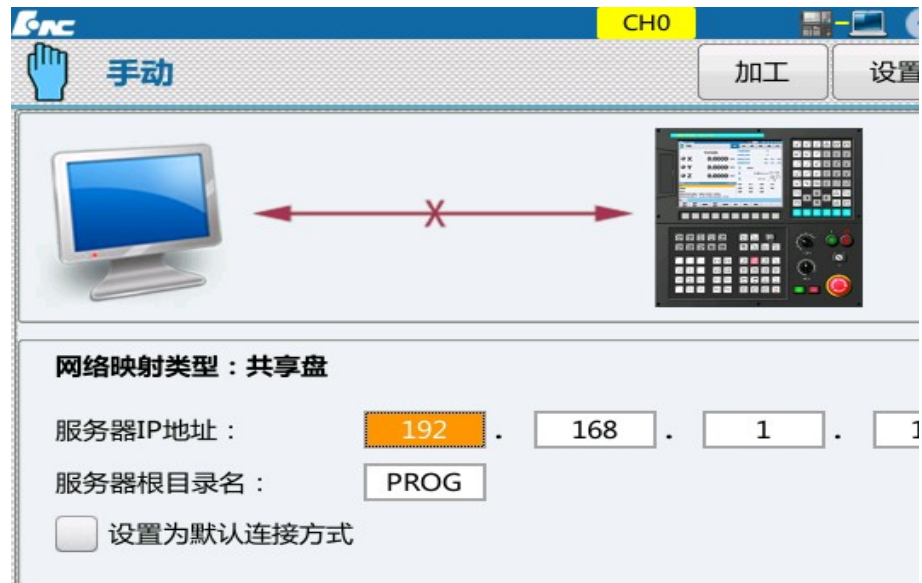
- While PING computer in the system, fill ip address of the computer in PING interface and click on 『PING start』 soft key;
- To terminate PING, press 「Reset」.

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

b) Shared disk communication

共享盘

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



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

c) FTP

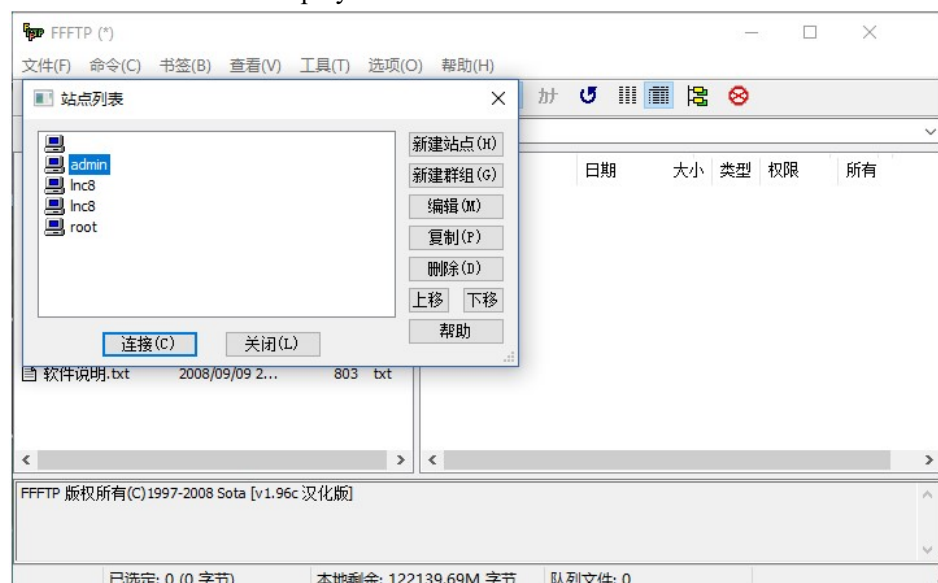
FTP

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

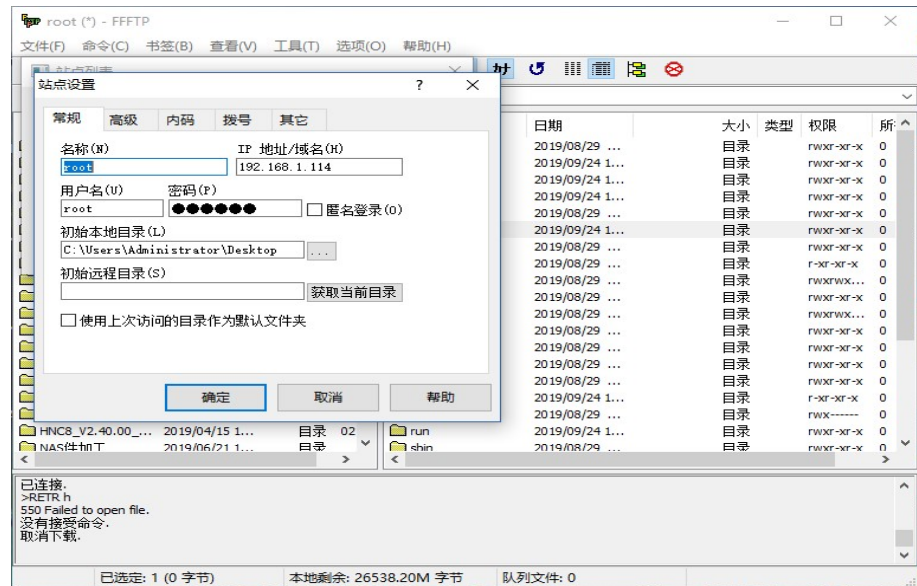
Before FTP is used, ensure smooth network according to the previous quarter

Then, install software FFFTP on computer (download from the official website of HNC CNC or Dingpan, see the right icon)

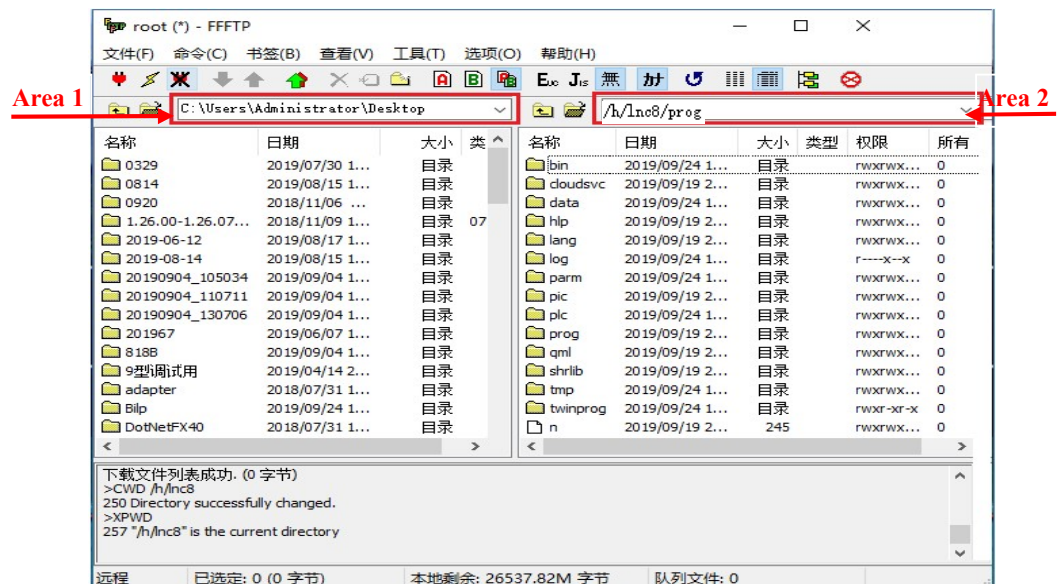
After ping the system and the computer, click on FFFTP icon and the interface is displayed as below:



We can select root login from the above interface station list. There is no necessary station, build a new station (such as root) and the station setup interface is shown below:



- ip address is ip of the system;
- Select the set 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.

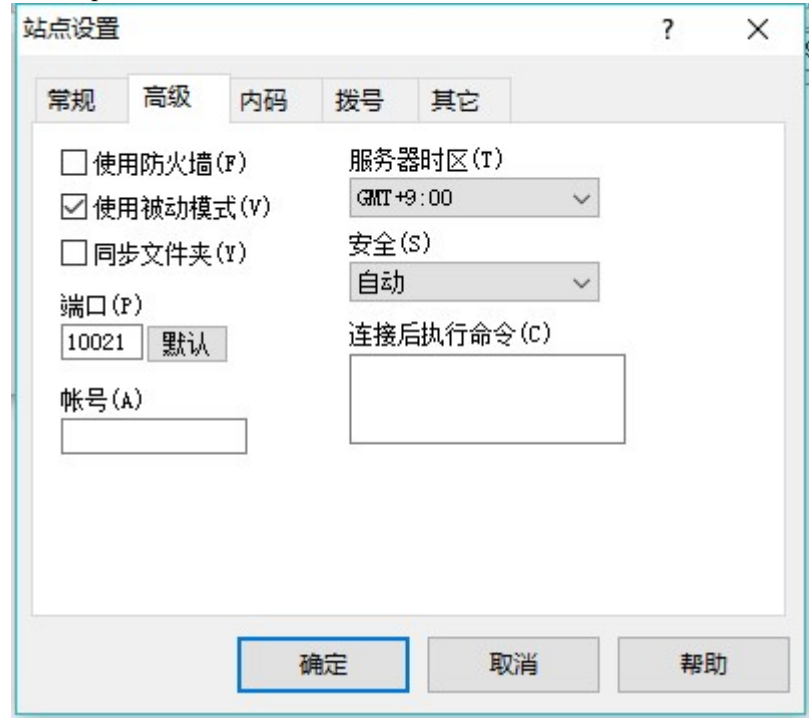


The transmission path of the system can be set in the above interface area 2 and system is often written in /h/lnc8/prog. The computer path can be set in area 1.

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

Attention: If fftp cannot be connected, inspect the following setups

1. Whether parameter 00050 starts the network and whether it is opened
2. Whether the computer and the system are connected to ping
3. Whether port number is correct, including port number 10001 in the CNC system interface and port number under the advanced menu of fftp.



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.
- A suitable adapter is needed,

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

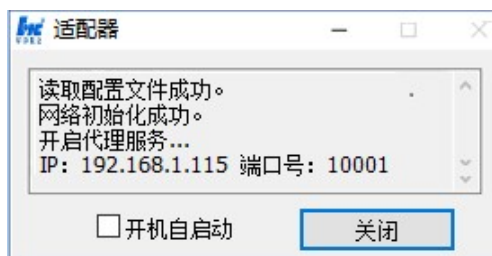
- (1) Turn on the adapter and find ServerWindowD.exe and ServerIp.xml

	mfc140d.dll	2015/6/25 23:34	应用程序扩展	8,044 I
	msvcpl40.dll	2015/6/25 23:34	应用程序扩展	430 I
	msvcpl40d.dll	2016/6/9 22:46	应用程序扩展	733 I
	msvcrt.dll	2011/12/16 16:46	应用程序扩展	620 I
	ServerIp.xml	2019/9/6 15:37	XML 文件	1 I
	ServerIp.xml.bak	2019/1/25 18:40	BAK 文件	1 I
	ServerWindowD.exe	2017/5/16 15:20	应用程序	3,974 I
	ucrtbase.dll	2015/6/10 19:10	应用程序扩展	863 I
	ucrtbased.dll	2015/7/9 22:33	应用程序扩展	1,479 I

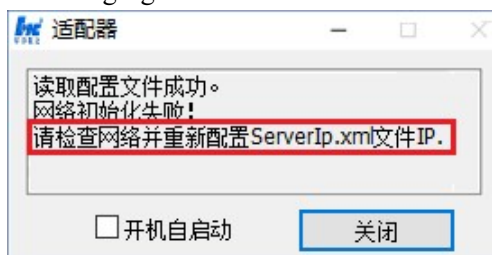
- (2) Open ServerIp.xml and replace LocalIP with ip with this computer, which is 192.168.1.115 in this example. Port number must be consistent with that on the system and should be edited and saved.

ServerIp.xml - 记事本
 文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)
 <ROOT> <LocalIP> 192.168.1.115 </LocalIP> <LocalPort> 10001 </LocalPort> </ROOT>

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

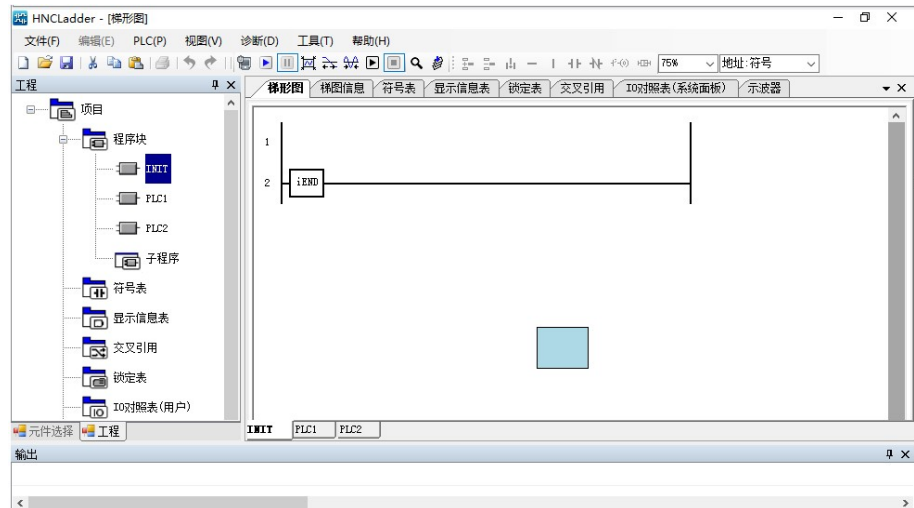


If ServerIp.xml is not modified, an alarm will be given as shown in the following figure



- (4) Find online commissioning in the second page of ladder diagram-ladder diagram information menu under the system diagnosis interface and click on it, the system will give a prompt message that PLC is being commissioned online.
- (5) Open ladder diagram editing tools of version V2.0 and click on Network setup under the tools menu



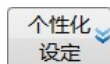


Correctly fill in ip of the computer and the system and click on Connect

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



3.6.7.6 Personalized setup



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



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

3. Skin setup

Skin setup supports black and blue. Subsequent versions will support gray as well.



3.6.8 Process package setup

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

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



"Process package" of this system is shown below:

- Backup

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

- Load

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

- Rename

The user 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

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

Note:

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

```

1  <?xml version="1.0" encoding="GB2312"?>
2  <CRAFTBAG version="1.0">
3      <item type="parm" no="000029" val="0"/>
4      <item type="parm" no="000032" val="20000"/>
5      <item type="parm" no="000067" val="60"/>
6      <item type="parm" no="000069" val="0"/>
7      <item type="parm" no="000077" val="0"/>
8      <item type="parm" no="010103" val="0x0"/>
9      <item type="parm" no="040087" val="0.0000"/>
10     <item type="parm" no="040088" val="0"/>
11     <item type="parm" no="040107" val="1"/>
12     <item type="parm" no="040089" val="0.0000"/>
13     <item type="parm" no="040158" val="0"/>
14     <item type="parm" no="040199" val="0.0000"/>
15     <item type="parm" no="040216" val="0.0000"/>
16     <item type="parm" no="040334" val="0"/>
17     <item type="parm" no="302154" val="0"/>
18 </CRAFTBAG>
19

```

version is 1.0;

encoding is GB2312;

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

- type is type, it is lowercase “parm” if it is parameter;





no is parameter number;

val is parameter value;

4 Power-on, power-off, safety protection, emergency stop




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

4.1 Power-on

Operation name	Power-on	Working mode	Emergency stop
Basic requirements	(1) Inspect whether machine tool status is normal; (2) Inspect whether power voltage is consistent with the requirements; (3) Inspect whether connection is correct and secure.		
SN	Operation steps	Key	Description
1	Press [Emergency stop]		<ul style="list-style-type: none"> ● Safety protection
2	Turn on [Air switch of machine tool]		<ul style="list-style-type: none"> ● Power on the machine tool
3	Press [System power-on]		<ul style="list-style-type: none"> ● Power on the system
4	Release [Emergency stop]		<ul style="list-style-type: none"> ● Rotate right to release [Emergency stop] ● System reset

Attention: 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]		<ul style="list-style-type: none"> ● Safety protection
2	Press [System power-off]		<ul style="list-style-type: none"> ● System power-off
3	Turn off [Air switch] of the machine tool		<ul style="list-style-type: none"> ● Power off the machine tool

Attention: To power on after power off, hold power-off for above 20s.



4.3 Over travel protection and release

4.3.1 Over travel protection

There is a limit switch on both ends of the travel of servo axis used to prevent from damaging servo axis due to collision. When the servo axis touches the travel limit switch, hard over travel protection will occur. When an axis undergoes hard over travel protection (the indicator light in "**Over travel release**" lights up), the system shuts down appropriately and the machine tool stops operation.

This system also can set soft over travel 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 Hard over travel release

Operation name	Over travel release		Working mode	Manual, 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 【Manual】 or 【Handwheel】			● Set effective working
2	Press [Over travel release] and [Axis feed]			<ul style="list-style-type: none"> ● Press [Over travel release] and [Axis feed] simultaneously ● Select [Axis feed] in the negative direction of over travel axis

Notices:

- Under manual (handwheel) mode, enable the axis to exit from over travel status in the negative direction;
- While operating the machine tool to exit from over travel status, please be sure to note movement direction and movement speed in order to avoid collision;
- If "Over travel release" is released, "Error" in the operating state column changes to "Normal operation" and it means to return to normal work and operation can be resumed.

4.3.3 Over travel release

Operation name	Over travel release		Working mode	Manual, handwheel
Basic requirements	(1) In case of over travel of an axis of the machine tool, all axes must not move and a prompt message is given.			
SN	Operation steps	Key		Description
1	Press 【Manual】 or 【Handwheel】			● Set effective working mode
2	Press [Axis feed]			● Press [Axis feed] in the negative direction of over travel axis
3	Press 「Reset」			● Clear alarm

4.4 Emergency stop

4.4.1 Feed hold



Press [Feed hold] when the machine tool runs the program automatically to suspend the machining program. But, it cannot be stopped immediately when a thread program runs.

4.4.2 Reset



When the system is at alarm status, in case of abnormal action and abnormal output of coordinate axis or to exit from input, press 「Reset」 to reset the system. "Reset" status of the system is as below:

- All axes stop running (except during thread machining);
- M and S function output is invalid;
- Stop automatic operation and hold mode function.

Besides, validation method of some parameters is "Reset". Press Reset to validate such parameters after they are inputted and saved.

4.4.3 Emergency stop



When the machine tool is running, in case of danger or an emergency, press "**Emergency stop**" and the CNC system enters emergency stop status and servo feed and spindle rotation stop immediately (feed driving power in the control cabinet is cut off); release "**Emergency stop**" (rotate this button to right and it jumps up automatically) and the system enters reset status.

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






Notices:

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

5 Manual operation and speed override

5.1 Return to the reference point manually

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	Return to the reference point manually		Working mode	Return to the reference point
Basic requirements	With the reference point as the boundary, ensure opposite direction of the feed axis of the machine tool stipulated by parameter "Direction of returning to the reference point"			
SN	Operation steps	Key		Description
1	Press 【Return to the reference point】			● Set effective working
2	Press [Axis feed]	 Or  Or  Or 		● Stipulate [Axis feed] of direction of returning to the reference point

Notices:

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

5.2 Move coordinate axis by manual feed

Movement of the coordinate axis can be controlled by continuous mode. Generally it is used for machining of simple parts. Press **【Manual】** working mode key, [Axis feed] key and [Feedrate override] key on the control panel of the machine tool to move coordinate axis of the machine tool manually.


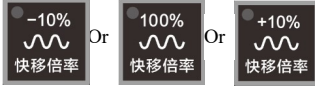





Operation name	Move coordinate axis by manual feed	Working mode	Manual
Basic requirements	(1) The need for continuous movement of machine tool		
SN	Operation steps	Key	Description
1	Press 【Manual】		<ul style="list-style-type: none"> ● Set effective working mode
2	Select [Feedrate override]		<ul style="list-style-type: none"> ● The product of default speed and feedrate override
3	Press [Axis feed]	 Or  Or  Or 	<ul style="list-style-type: none"> ● If the key is released, feed stops

Notices:

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

5.3 Move coordinate axis quickly manually

Move the coordinate axis quickly manually. Press **【Manual】** working mode, [Quick traverse magnification] and [FF]+[Axis feed] on the control panel of the machine tool to complete this operation.

Operation name	Move coordinate axis quickly manually		Working mode	Manual
Basic requirements	(1) The need for quick movement of machine tool			
SN	Operation steps	Key	Description	
1	Press 【Manual】		<ul style="list-style-type: none"> ● Set effective working mode 	
2	Select [Quick traverse magnification]		<ul style="list-style-type: none"> ● The product of default speed and quick traverse magnification 	
3	Press [Quick traverse] and [Axis feed]	    	<ul style="list-style-type: none"> ● Press [FF] and [Axis feed] simultaneously ● If the key is released, feed stops 	



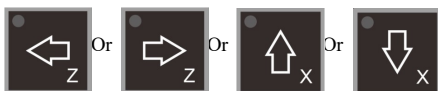
Notices:

- Set 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 quick traverse override rate as per 10%;
- Under other modes than "Manual" mode, [FF] is invalid.

5.4 Move the coordinate axis by increment feed

Increment feed can move the coordinate axis quantitatively by pulse mode. It is often used for tool setting or magazine tool commissioning and other operations to control accurate positioning of machine tool.

Press **【Increment】** working mode, [Magnification] and [Axis feed] on the control panel of handheld unit and machine tool to move coordinate axis of the machine tool by increment feed.

Operation name	Move the coordinate axis by increment feed		Working mode	Increment
Basic requirements	(1) For manual accurate positioning			
SN	Operation steps	Key	Description	
1	Press 【Increment】		● Set effective working mode	
2	Press [Magnification]		● The product of magnification and 0.001mm	
3	Press [Axis feed]		● Press once to move to corresponding axis and direction for an increment distance.	










Notices:

- Press [Axis feed] once and the axis moves for a fixed increment distance. Release it and press it again and the axis moves for another fixed increment distance. Press and hold it and the axis does not move;
- The increment distance is the product of 001mm and magnification;
- HNC-808Di-TU system has three kinds of magnification: $\times 1$, $\times 10$ and $\times 100$;
- There are interlocking keys between [Magnification] keys, namely others cannot be validated when on is valid;

5.5 Move the coordinate axis by handwheel feed

Move the coordinate axis by handwheel feed and continuously and quantitatively move the coordinate axis. It is often used for tool setting or magazine tool commissioning and other operations to control accurate positioning of machine tool.









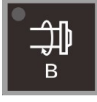
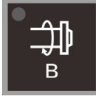

Press **【Handwheel】** working mode, [Magnification] and [Axis feed] on the control panel of handheld unit and machine tool to move coordinate axis of the machine tool by handwheel feed.

Operation name	Move the coordinate axis by handwheel feed		Working mode	Handwheel
Basic requirements	(1) The need for continuous accurate movement of machine tool			
SN	Operation steps	Key	Description	
1	Press 【Handwheel】		● Set effective working mode	
2	Press [Magnification]	 Or  Or 	● The product of magnification and 0.001mm	
3	Press [Axis feed]	 Or  Or  Or 	● While swinging the handwheel for a grid, it moves to corresponding axis and direction for an increment value.	
4	Swing [Handwheel]		● Continuously and accurately move the machine tool	










Notices:

- When the handwheel rotates, the movement distance per grid is the product of 0.001mm and magnification;
- HNC-808Di-TU system has three kinds of magnification: $\times 1$, $\times 10$ and $\times 100$;
- There are interlocking keys between [Magnification] keys, namely others cannot be validated when on is valid;
- The handwheel should rotate at speed of no greater than 5r/s. If the handwheel rotates too fast, the movement distance is not equal to pulse count of handwheel or the feed axis cannot stop immediately when the handwheel stops rotating. The former is by default for HNC-808Di-TU system.

5.6 Manual control of spindle

SN	Operation name	Start operation	Stop operation	Description	Effective working mode
1	The spindle rotates positively	Press [Positive rotation of spindle] 	Press [Spindle stop] or [Reset]  	1. [Positive rotation of spindle], [Negative rotation of spindle] and [Spindle stop] are interlocked; 2. While spindle control is needed for automatic operation, switch to manual mode and then switch back to automatic mode.	Handwheel, increment, manual
2	The spindle rotates negatively	Press [Negative rotation of spindle] 	Press [Spindle stop] or [Reset]  		
3	Spindle stop	Press [Spindle stop]	Press [Reset]		
4	Positive rotation/stop of power head	Press [A] 	Press [A] 	1. Press [A] to control positive rotation or stop of power head cyclically.	
5	Negative rotation or stop of power head	Press [B] 	Press [B] 	1. Press [A] to control negative rotation or stop of power head cyclically.	
6	Speed override of spindle	Press [Spindle magnification] 		1. Based on 100% of default speed, increase and decrease quick traverse override rate as per 10%; 2. Override range: 0%-150%.	Handwheel, increment, manual, automatic, MDI

5.7 Other manual operations

SN	Operation name	Start operation	Stop operation	Description	Effective working mode
1	Tip forward	Press [Tip forward]	 顶尖前进	Press this key again	Press [Tip forward] to cyclically control forward and stop of tip. It is stop by default.
2	Tip inch	Press [Tip inch]	 顶尖寸动	Stop after inch once	Press [Tip inch] once and the tip can advance for a fixed distance.
3	Tip backward	Press [Tip backward]	 顶尖后退	Press this key again	Press [Tip backward] to cyclically control backward and stop of tip. It is stop by default.
4	Clamping jaw ON/OFF	Press [Clamping jaw ON/OFF]	 夹爪开/关	Press this key again	Press Clamping jaw ON/OFF to cyclically control ON or OFF of clamping jaw. It is OFF by default.
5	Positive rotation of magazine tool	Press [Positive rotation of magazine tool]	 刀库正转	Stop after rotating for a cutter bit	Press [Positive rotation of magazine tool] once and the magazine tool rotates positively for a tool bit.
6	Lighting ON/OFF	Press [Machine tool lighting]	 机床照明	Press this key again	Press [Machine tool lighting] to cyclically control ON or OFF of light. It is OFF by default.
7	Lubrication start/stop	Press [Lubrication]	 润滑	Press this key again	Press [Lubrication] to cyclically control ON or OFF of lubricating pump. It is OFF by default.
8	Positive rotation of chip removal	Press [Chip removal magazine tool]	 排屑正转	Press this key again	Press [Positive rotation of chip removal] to control positive rotation or stop of chip removal motor cyclically. It is OFF by default.
9	Cooling ON/OFF	Press [Cooling] key	 冷却	Press this key again	Press [Cooling] to control start and stop of cooling pump cyclically. It is stop by default.

5.8 Speed override

5.8.1 Feed speed override

Under automatic mode or MDI operation mode, when the feed speed of F instruction programming is too high or too low, rotate the feedrate override band switch to adjust the feed speed in the override program. Override range: 0%-120%.



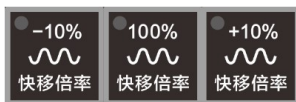
If feed speed is not designated in the automatic operational program and "Machining preparation" function is not enabled, the system runs as per the set value of channel parameter "Default feed speed" 040030. If "Machining preparation" function is enabled, the system runs as per the machining preparation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

Under manual continuous feed mode, the band switch can regulate manual feed rate and the system runs as per the set value of axis parameter "Slow speed jog speed" 100032 (X axis), 101032 (Y axis) and 102032 (Z axis).

Notices:

- When feed override knob is set as 0, feed override rate is 0 and quick traverse override rate changes to 0 temporarily. If feed override rate is changed to not 0, quick traverse override rate recovers to the original value;
- During thread machining, feed override is invalid and the system runs as per the original speed;

5.8.2 Quick traverse speed override



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

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

Quick traverse segment in manual quick traverse, G00 quick traverse and fixed cycle and quick traverse segment in G28 and G29 can increase and decrease by 10% by "Quick traverse magnification" and the override range is 0%-100%.

Notices:

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

5.8.3 Selection of increment magnification



Under increment and handwheel mode, move a fixed distance after an instruction pulse is issued (the handwheel rotates for a grid or [Axis feed] is pressed once) and the value of fixed distance is controlled by increment magnification.

HNC-808Di-TU device has three increment magnification buttons "×1", "×10" and "×100". The corresponding relationship between increment magnification key and distance value is shown below:

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

Notices

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







6 Program editing and management

6.1 Program search





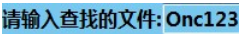


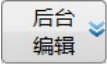
HNC-808DiM system has program search function under "Machining" function set and "Program" function set, but usage of programs searched under two function sets is different. Programs searched under "Machining" function set are used for machining program loading, program backstage editing and other operations;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

6.1.1 Search of machining or editing program




6.1.1.1 Direct search









Operation name	Search of machining or editing program	Working mode	Automatic, single segment, manual, MDI
Basic requirements	Programs to be searched exist	Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description
1	Press 『Machining』		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface and menu
3	Press 『System disk』 and 『USB flash disk』, and so on		● Select system disk, USB flash disk, network disk and user disk
4	Press 「Cursor」 or 「PgUp/PgDn」		<ul style="list-style-type: none"> ● Complete search and move the cursor to the program name to be searched ● Searched programs can be used for two purposes
5.1	Press 「Enter」		● Searched programs are used for loading machining program and run
5.2	Press 『Backstage editing』		● Searched programs are used for entering program editing status

6.1.1.2 Search programs under different disks by "Search" function

Operation name	Search of machining or editing program	Working mode	Automatic, single segment, manual, MDI
Basic requirements	Programs to be searched exist	Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description
1	Press 『Machining』		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface, level 2 menu
3	Press 『System disk』 and 『USB flash disk』, and so on		● Select system disk, USB flash disk, network disk and user disk
4	Press 『Search』		● 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 run
7.2	Press 『Backstage editing』		● Searched programs are used for entering program editing status









6.1.1.3 Search programs under the directory by "Search" function

Operation name	Search of machining or editing program	Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs to be searched	Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description
1	Press 『Machining』		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface, level 2 menu
3	Press 『System disk』 and 『USB flash disk』, and so on		● Select system disk, USB flash disk, network disk and user disk






4	「Cursor」 or 「PgUp/PgDn」	   	<ul style="list-style-type: none"> ● Move the cursor to the selected file directory name
5	Press 「Enter」		<ul style="list-style-type: none"> ● Confirm and open the directory
6	Press 「Search」		<ul style="list-style-type: none"> ● Prompt: Enter a file name
7	(Enter a file name)	---	<ul style="list-style-type: none"> ● Prompt: Enter a file name
8	Press 「Enter」		<ul style="list-style-type: none"> ● Complete searching programs and move the cursor to the program to be searched; ● Searched programs can be used for two purposes
9.1	Press 「Enter」		<ul style="list-style-type: none"> ● Searched programs are used for loading machining program and run
9.2	Press 「Backstage editing」		<ul style="list-style-type: none"> ● Searched programs are used for entering program editing status

6.1.2 Search of management program (to be transmitted and deleted)






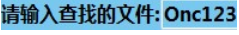

6.1.2.1 Direct search

Operation name	Search of management (copy, paste) program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs to be searched		Display interface	3.4 "Program" function set interface
SN	Operation steps	Key	Description	
1	Press 「Program」		<ul style="list-style-type: none"> ● Default interface, main menu 	
2	Press 「System disk」 and 「USB flash disk」, and so on	  	<ul style="list-style-type: none"> ● Select system disk, USB flash disk, network disk and user disk 	
3	Press 「Cursor」 or 「PgUp/PgDn」	   	<ul style="list-style-type: none"> ● Complete search ● Move the cursor to the program name to be searched 	

6.1.2.2 Search programs under different disks by "Search" function

Operation name	Search of management (copy, paste) program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs to be searched		Display interface	3.4 "Program" function set interface
SN	Operation steps	Key	Description	
1	Press 【Program】		● Default interface, main menu	
2	Press 【System disk】 and 【USB flash disk】, and so on		● Select system disk, USB flash disk, network disk and user disk	
3	Press 【Search】		● Prompt: Enter a file name	
4	(Enter a file name)	---	● e.g.: Onc123 	
5	Press 【Enter】		● Complete search ● Move the cursor to the program name to be searched	

6.1.2.3 Search programs under the directory by "Search" function

Operation name	Search of management (copy, paste) program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs to be searched		Display interface	3.4 "Program" function set interface
SN	Operation steps	Key	Description	
1	Press 【Program】		● Default interface, main menu	
2	Press 【System disk】 and 【USB flash disk】, and so on		● 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 【Search】		● Prompt: Enter a file name	
6	(Enter a file name)	---	● e.g.: Onc123 	
7	Press 【Enter】		● Complete search ● Move the cursor to the program name to be searched	

6.2 Program editing




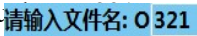


This system enters program editing status in 4 ways. 1. "Create new programs" under "Machining" function set; 2. "Program editing" under "Machining" function set; 3. "Backstage editing" under "Machining" function set; 4. "Create new programs" under "Program" function set.

6.2.1 Create new programs








HNC-808DiM system has "Create new programs" function under "Machining" function set and "Program" function set. Although two "Create new programs" is operated in different ways, their functions are basically identical. Configuration of two "Create new programs" can simplify operator's use.

6.2.1.1 Create new programs under "Machining" function set



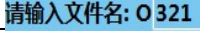

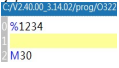

1) Create new programs under "Program editing" sub-interface

Operation name	Create new programs under "Machining" function set	Working mode	Automatic, single segment, manual, MDI
Basic requirements	The new program name cannot be the same as existing program names	Display interface	3.2.3 "Edit program" sub-interface
SN	Operation steps	Key	Description
1	Press 【Machining】		● Default interface, main menu
2	Press 【Edit program】		● Enter current loading program editing status
3	Press 【Create】		● Prompt: Please enter a file name
4	(Enter a program name)	---	● Input program name su 
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

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

Operation name	Create new programs under "Machining" function set	Working mode	Automatic, single segment, manual, MDI
Basic requirements	The new program name cannot be the same as existing program names	Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description
1	Press [Machining]		<ul style="list-style-type: none"> Default interface, main menu
2	Press [Select program]		<ul style="list-style-type: none"> "Select program" sub-interface, level 2 menu
	Press [Backstage editing]		<ul style="list-style-type: none"> Enter the "Backstage editing" sub-interface, level 3 interface
3	Press [Create]		<ul style="list-style-type: none"> Prompt: Please enter a file name
4	(Enter a program name)	---	<ul style="list-style-type: none"> Input program name such as 
5	Press [Enter]		<ul style="list-style-type: none"> 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)	---	<ul style="list-style-type: none"> Complete program editing
7	Press [Save file]		<ul style="list-style-type: none"> A prompt message Program has been saved will be given The new program is immediately loaded as machining program




6.2.1.2 Create new programs under "Program" function set

Operation name	New programs created under "Program" function set	Working mode	Automatic, single segment, manual, MDI
Basic requirements	The new program name cannot be the same as existing program names	Display interface	3.4 "Program" function set interface
SN	Operation steps	Key	Description
1	Press [Program]		<ul style="list-style-type: none"> Default interface, main menu
2	Press [Create]		<ul style="list-style-type: none"> A prompt message "Please enter a file name" will be given 
3	(Enter a file name)	---	<ul style="list-style-type: none"> e.g.: Onc321
4	Press [Enter]		<ul style="list-style-type: none"> Confirm file name and switch to "Machining" function set Enter the editing area 
5	(Edit program)	---	<ul style="list-style-type: none"> Complete program editing
6	Press [Save file]		<ul style="list-style-type: none"> A prompt message Program has been saved will be given The new program is not loaded as machining program

6.2.2 Modification and editing of program

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





6.2.2.1 Editing and modification of current loading program


Operation name	Editing and modification of current loading program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing loading programs		Display interface	3.2.3 "Edit program" sub-interface
SN	Operation steps	Key	Description	
1	Press 【Machining】		● Default interface, main menu	
2	Press 【 Edit program】		● Enter current loading program editing status	
3	(Edit program)	---	● Edit and modify existing loading programs	
4	Press 【 Save file】		● Program has been saved	

Notices:

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

6.2.2.2 Editing and modification of backstage of non-loading program

Operation name	Editing and modification of backstage of non-loading program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs to be modified		Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description	
1	Press 【Machining】		● Default interface, main menu	
2	Press 【 Select program】		● "Select program" sub-interface	
3	Press 【 Cursor】		● Select current programs to be edited and modified	
4	Press 【 Backstage editing】		● Enter program editing status	

5	(Edit program)	---	<ul style="list-style-type: none"> Edit and modify existing loading programs
6	Press 『Save file』		<ul style="list-style-type: none"> Program has been saved


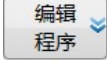
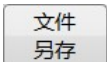



Notices:

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










6.2.3 Save program as

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

6.2.3.1 Save program as of "Current loading program"

Operation name	Save program as of "Current loading program"	Working mode	Automatic, single segment, manual, MDI
Basic requirements	The saved file name cannot be the same as existing program names	Display interface	3.2.3 "Edit program" sub-interface
SN	Operation steps	Key	Description
1	Press 『Machining』		<ul style="list-style-type: none"> Default interface, main menu
2	Press 『 Edit program』		<ul style="list-style-type: none"> Enter current loading program editing status
3	Press 『Save a file as』		<ul style="list-style-type: none"> Storage target selection dialog box "System disk, USB flash disk or user disk" can be selected The file directory of all disks can be selected
4	Press 『Cursor』		<ul style="list-style-type: none"> Move the cursor to the selected target disk or file directory name
5	Press 『O』		<ul style="list-style-type: none"> Activate file name input box
6	(Input the saved file name)		<ul style="list-style-type: none"> If renamed, the original program will be covered
7	Press 『Enter』		<ul style="list-style-type: none"> Current loading program is saved in the target position Exit from storage target selection dialog box The saved file is current editing program

6.2.3.2 Save program of "Non-loading program"





Operation name	Save program of "Non-loading program"		Working mode	Automatic, single segment, manual, MDI
Basic requirements	The saved file name cannot be the same as existing program names		Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description	
1	Press [Machining]		● Default interface, main menu	
2	Press [Select program]		● "Select program" sub-interface	
3	Press [Cursor]		● Select current program to be saved ●	
4	Press [Backstage editing]		● Enter program editing status	
5	Press [Save a file as]		● Storage target selection dialog box  ● "System disk, USB flash disk or user disk" can be selected ● The file directory of all disks can be selected	
6	Press [Cursor]		● Move the cursor to the selected target disk or file directory name ●	
7	Press [O]		● Activate 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 from storage target selection dialog box ● The saved file is current editing program	

6.2.4 Copy and paste of program segment


Operation name	Copy and paste of program segment		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Enter program editing status		Display interface	3.2 "Machining" function set interface 3.3 "Program" function set interface
SN	Operation steps	Key	Description	
1	(Edit program)		<ul style="list-style-type: none"> ● This system enters program editing status in 4 ways: <ul style="list-style-type: none"> ✧ "Create new programs" function under "Machining" function set ✧ "Edit programs" function under "Machining" function set ✧ "Backstage editing" function under "Machining" function set ✧ "Create new programs" function under "Program" function set 	
2	『Block operation』	块操作	● Enter "Block operation" sub-interface	
3	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the head of the copied segment	
4	『Definition block head』	定义块头	---	
5	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to the end of the copied segment	
6	『Definition block end』	定义块尾	---	
7	『Block copy』	块复制	---	
8	「Cursor」 or 「PgUp/PgDn」		● Move the cursor to where current program or other programs are pasted	
9	『Block paste』	块粘贴	● Paste succeeds	
10	『Save file』	保存文件	● Exit and save programs	




6.3 Program management

6.3.1 Rename of file directory and program

Operation name	Rename of file directory and program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs can be searched		Display interface	3.4.4 "Program rename" sub-interface
SN	Operation steps	Key	Description	
1	Press 『Program』		● Default interface, main menu	
2	(Search directory and program)	---	● Move the cursor to the directory and program to be renamed according to 6.1.2 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	


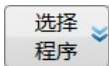


6.3.2 Copy and paste file directory and program

Operation name	Copy and paste file directory and program		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Existing programs can be searched		Display interface	3.4.2.2 "Program copy and paste" sub-interface
SN	Operation steps	Key	Description	
1	Press 『Program』		● 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: Paste succeeds

6.3.3 Program deletion

6.3.3.1 Program deletion under "Machining" function set

Operation name	Deletion of program (under "Machining" function set)	Working mode	Automatic, single segment, manual, MDI
Basic requirements	Programs to be deleted can be searched	Display interface	3.2.2 "Program selection" sub-interface
SN	Operation steps	Key	Description
1	Press 『Machining』		● Default interface, main menu
2	Press 『Select program』		● "Select program" sub-interface
3	(Search directory and program)	---	● Move the cursor to the program name to be deleted according to 6.1.2 search program of "Program" function set
4	Press [Delete]		● Prompt: "Conform to delete the selected file? (Y/N)"
5	Press 「Y」 or 「N」		<ul style="list-style-type: none"> ● Press 「Y」 to complete deletion ● Press 「N」 to abandon deletion

6.3.3.2 Program deletion under "Program" function set

Operation name	Deletion of program (under "Program" function set)		Working mode	Automatic, single segment, manual, MDI
Basic requirements	Programs to be deleted can be searched		Display interface	3.4 "Program" function set interface
SN	Operation steps	Key	Description	
1	Press 【Program】		<ul style="list-style-type: none"> ● Default interface, main menu 	
2	(Search the program to be deleted)	---	<ul style="list-style-type: none"> ● Move the cursor to the program name to be deleted according to 6.1.2 search program of "Program" function set 	
3	Press [Delete]		<ul style="list-style-type: none"> ● Prompt: "Conform to delete the selected file? (Y/N)" 	
4	「Y」 or 「N」		<ul style="list-style-type: none"> ● Press 「Y」 to complete deletion ● Press 「N」 to abandon deletion 	







7 Automatic operation

7.1 Automatic 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	Automatic, single segment, manual, MDI
Basic requirements	Create new programs under "Machining" function set		Display interface	3.2.3 "Edit program" sub-interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		● Maintain the original interface	
2	Press 【Machining】		● Default interface, main menu	
3	Press 【Edit program】		● The cursor enters the loaded program editing area	
4	Press 【Create】			
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 input, file name is Onc123 ● The cursor enters the editing area	
7	(Edit program)	---	● Edit program and complete	
8	Press 【Save file】		● The newly edited program is loaded as machining program immediately ● A prompt message File has been saved will be given	




Notices:

- 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	Automatic, single segment, manual, MDI
Basic requirements	Existing programs to be loaded in all disks		Display interface	3.2.2 "Select program" sub-interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		● 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】, and so on	 	● Select system disk/USB flash disk/network disk/user disk	
5	(Search loading program)	---	● Select programs to be loaded as "Current machining program" ● Search programs as per 6.1.1	
6	「Enter」		● Loading completed	





7.1.2 Program run

Operation name	Program run		Working mode	Automatic
Basic requirements	Machining program has been loaded		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		● Maintain the original interface	
2	Press 【Machining】		● Default interface, main menu	
3	(Loading program)	---	● Load machining program as per 7.1.1	
4	(Safety inspection)	---	● Complete deceleration and lock, and so on	
5	Press [Cycle start]		● Run a program automatically	





Notices:

- 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 verification

Operation name	Program verification		Working mode	Automatic, single segment
Basic requirements	Machining program has been loaded		Display interface	3.2.4 "Verify program" sub-interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		<ul style="list-style-type: none"> ● Maintain the original interface 	
2	Press 【Machining】		<ul style="list-style-type: none"> ● Default interface, main menu 	
3	(Loading program)	---	<ul style="list-style-type: none"> ● Load machining program as per 7.1.1 	
4	Press 【Verify program】		<ul style="list-style-type: none"> ● Working mode is displayed as "Verify" ● 【Verify program】 soft key is at highlight status 	
5	Press [Cycle start]		<ul style="list-style-type: none"> ● Exit from verification after automatic operation ● Press [Reset] to exit from verification 	

7.1.4 Program graphics simulation



Operation name	Program graphics simulation		Working mode	Automatic, single segment
Basic requirements	Machining program has been loaded		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		<ul style="list-style-type: none"> ● Maintain the original interface 	
2	Press 【Machining】		<ul style="list-style-type: none"> ● Default interface, main menu 	
3	(Loading program)	---	<ul style="list-style-type: none"> ● Load machining program as per 7.1.1 	
4	Press 【Display switch】		<ul style="list-style-type: none"> ● Press this key once to switch to an interface and repeatedly to switch among different interfaces cyclically ● Select "Graphics+ program" interface 	
5	Press [Cycle start]		<ul style="list-style-type: none"> ● Conduct automatic operation and realize graphics simulation 	

Notices

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

7.2 Automatic operation control







7.2.1 Single-segment operation

Operation name	Single-segment operation		Working mode	Single segment
Basic requirements	Complete loading of machining program		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【 Single segment 】		<ul style="list-style-type: none"> ● Maintain the original interface 	
3	Press 〔Machining〕		<ul style="list-style-type: none"> ● Default interface, main menu 	
4	(Loading program)	---	<ul style="list-style-type: none"> ● Load machining program as per 7.1.1 	
5	Press [Cycle start]		<ul style="list-style-type: none"> ● Press Start once to run a segment of program, and so on 	

Notices:

- Single segment mode can be verified and simulated like automatic mode.









7.2.2 Skip operation

Operation name	Skip operation		Working mode	Automatic, single segment
Basic requirements	Program segment number is prefixed with “/”, e.g.: /N1 X30 Z5		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【 Manual 】 or 【 Handwheel 】	 or 	<ul style="list-style-type: none"> ● Program skip operation function is set only under manual, handwheel or increment mode 	
2	Press [Program skip]		<ul style="list-style-type: none"> ● Program segments with skip symbol will be skipped 	
3	Press 【 Automatic 】		<ul style="list-style-type: none"> ● Maintain the original interface 	
4	Press 〔Machining〕		<ul style="list-style-type: none"> ● Default interface, main menu 	
5	(Loading program)	---	<ul style="list-style-type: none"> ● Load machining program as per 7.1.1 	
6	Press [Cycle start]		<ul style="list-style-type: none"> ● Skip the marked segment during automatic operation. 	

Notices:

- If program segments with skip symbol “/” are executed after [Program skip] is pressed, the system skips over this line of instructions and directly executes the next line of instructions.
- If [Program skip] is not pressed, the system still executes this line of instructions by sequence.

7.2.3 Run from any line

Operation name	Run from any line		Working mode	Automatic, single segment
Basic requirements	Do not start from the subprogram line		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		● Maintain the original interface	
2	Press [Machining]		<ul style="list-style-type: none"> ● Default interface, main menu ● Correctly load programs executing any line operation 	
3	Press [→]		● Enter the extension menu	
4	Press [Any line]		● Enter the "Any line" submenu	
5	Press [Designated line number] and [Designated N number]	 Or 	<ul style="list-style-type: none"> ● The indicator light lights off ● Suspend operation 	
6	(Input line number)	---	● Input a value, such as 8	
7	Press [Enter]		<ul style="list-style-type: none"> ● Confirm input ● The cursor moves to the line in front of the inputted line ● Move the cursor to the selected any line by [Cursor] 	
8	Press [Cycle start]		● Start to run from the designated line	

Notices:

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

0: Non-scanning mode. Namely modes before the target line are not inherited;

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

2: Full scanning mode. Namely modes before the target line are inherited.

- Axis location sequence can be set by "Any line axis location sequence" parameter 040114, which is a value type parameter. The corresponding relationship between bit and 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 axes reach the right position. 0 means axes are not configured.

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






When the mill moves, 040114=101, it means that X/Z arrives at the right position simultaneously and Y does not move.

- While using "Designated N number" function, there should be instruction address N in the first program segment.





7.2.4 Stop operation

Operation name	Stop operation	Working mode	Automatic
Basic requirements	There is M00 "Stop operation" instruction in loading program	Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description
1	Press 【Automatic】		<ul style="list-style-type: none"> ● Maintain the original interface
2	Press 【Machining】		<ul style="list-style-type: none"> ● Default interface, main menu ● And load machining program correctly
3	Press [Cycle start]		<ul style="list-style-type: none"> ● The program is running
4	(Execute instruction) M00	----	<ul style="list-style-type: none"> ● The program suspends ● Manual tool changing and other operations can be executed
5	Press [Cycle start]		<ul style="list-style-type: none"> ● Continue running subsequent programs

7.2.5 Select stop

Operation name	Select stop	Working mode	Automatic
Basic requirements	There is M01 "Select stop" instruction in loading program	Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description
1	Press 【Automatic】		<ul style="list-style-type: none"> ● Maintain the original interface
2	Press [Select stop]		<ul style="list-style-type: none"> ● If this step is not executed, continuously run the program
3	Press 【Machining】		<ul style="list-style-type: none"> ● Default interface, main menu ● And load machining program correctly
4	Press [Cycle start]		<ul style="list-style-type: none"> ● The program is running
5	(Execute instruction) M01	----	<ul style="list-style-type: none"> ● The program suspends ● If step 2 is not executed, the program does not stop but runs continuously
6	Press [Cycle start]		<ul style="list-style-type: none"> ● Continue running subsequent programs





7.2.6 Suspend operation

Operation name	Suspend operation	Working mode	Automatic
Basic requirements	The program continuously runs	Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description
1	Press 【Automatic】		<ul style="list-style-type: none"> ● Maintain the original interface
2	Press 〔Machining〕		<ul style="list-style-type: none"> ● Default interface, main menu
3	(Running program)	---	<ul style="list-style-type: none"> ● The program is running
4	Press [Feed hold]		<ul style="list-style-type: none"> ● The indicator light lights off ● Suspend operation
5	Press [Cycle start]		<ul style="list-style-type: none"> ● Continuous operation

Notices:

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

7.2.7 Terminate operation












Operation name	Terminate operation		Working mode	Automatic
Basic requirements	The program continuously runs		Display interface	3.2 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		<ul style="list-style-type: none"> ● Maintain the original interface 	
2	Press [Machining]		<ul style="list-style-type: none"> ● Default interface, main menu 	
3	(Running program)	---	<ul style="list-style-type: none"> ● The program is running 	
4	Press [Feed hold]		<ul style="list-style-type: none"> ● The indicator light lights off ● Suspend operation 	
5	Press 【Manual】		<ul style="list-style-type: none"> ● To close MST manually 	
6	(Disable M and S functions)	---	<ul style="list-style-type: none"> ● Disable MST manually 	
7	Press [Emergency stop]		<ul style="list-style-type: none"> ● Terminate operation and reset 	





7.3 MDI operation

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

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

Where, HNC-808Di-TU system mostly has working mode **【MDI】** key. Operation and function of two types of "MDI" keys are basically identical.

Operation name	MDI operation		Working mode	MDI
Basic requirements	The system can run normally		Display interface	
SN	Operation steps	Key	Description	
1	Press 【MDI】	 or 	<ul style="list-style-type: none">● Enter the MDI interface and the main menu● The cursor is in the editing area	
2	(Edit MDI program)	---	<ul style="list-style-type: none">● Edit many lines at a time before operation	
3	Press 『Input』		<ul style="list-style-type: none">● Input all programs in the editing area	
4	Press [Cycle start]		<ul style="list-style-type: none">● The machine tool runs as per the input program● Programs in the editing area are reserved, even though the interface is switch	
5.1 To run MDI program by single segment				
5.1.1	Press 【MDI】	 or 	<ul style="list-style-type: none">● Enter the MDI interface and the main menu● The cursor is in the editing area	
5.1.2	Press 【Single segment】		<ul style="list-style-type: none">● The indicator light of single-segment mode lights up	
5.1.3	Press [Cycle start]		<ul style="list-style-type: none">● The machine tool runs as per the input program● Programs in the editing area are reserved, even though the interface is switch	
5.2 To rerun programs in the editing area				
5.2.1	Press 『Input』		<ul style="list-style-type: none">● Repeat steps 3 and 4	
5.2.2	Press [Cycle start]		---	
5.2 To suspend a program				
5.3.1	Press 『Suspend』		<ul style="list-style-type: none">● The machine tool stops operation and is at feed hold status● Press [Cycle start] to continuously run the subsequent programs	






5.3 To cancel this operation			
5.4.1	Press 「Reset」		<ul style="list-style-type: none"> ● This operation is canceled and reenter to resume operation ● Not to reset during threading and drilling
5.4 To save MDI program in the system disk			
5.5.1	Press 『Save』		<ul style="list-style-type: none"> ● The input box gives a prompt message: Please enter a file name
5.5.2	(Enter a file name)	---	<ul style="list-style-type: none"> ● The program will be saved to PROG direction of the system disk
5.5.3	Press 「Enter」		<ul style="list-style-type: none"> ● The program has been saved and a prompt message will be given
5.5 To clear programs in the MDI editing area at a time			
5.6.1	Press 『Clear』		<ul style="list-style-type: none"> ● Power off to clear programs in the MDI editing area ● The edited MDI program cannot be cleared through switching the interface

Notices:







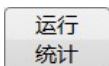
- "MDI" is a key on the upper panel or the lower panel and it should be set through parameter 000371. When it is set as 0, it is a key of the lower panel. When it is set as 1, it is a key of the upper panel.
- If 『MDI』 key is on the lower panel, “MDI” is working mode key. MDI working mode has an independent interface. Switch working mode and the interface changes accordingly.
- If 『MDI』 key is on the upper panel, “MDI” is function set key. This function is valid under automatic mode or single segment mode. The system will give a prompt message and an alarm after switching to other working modes.

7.4 Handwheel trial cut

This function controls the machine tool axis to run as per the program path through handwheel pulse generator. It is often used to inspect 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 segment mode.

Operation name	Handwheel trial cut		Working mode	Automatic
Basic requirements	Machining preparation is at normal automatic operating state of program		Display interface	See Chapter 3 "Machining" function set interface
SN	Operation steps	Key	Description	
1	Press 【Automatic】		<ul style="list-style-type: none"> This function can be executed under automatic operation 	
2	Press [Handwheel simulation]		<ul style="list-style-type: none"> If this function is valid, the indicator light lights up 	
3	Press [Cycle start]		<ul style="list-style-type: none"> The instruction runs normally before the machine tool axis moves, such as positive rotation of spindle, and so on Then, the feed axis of the machine tool does not move 	
4	(Rotate the handwheel)		<ul style="list-style-type: none"> If the handwheel rotates positively, the feed axis of the machine tool moves forward with the program If the handwheel rotates negatively, the feed axis of the machine tool moves backward with the program 	
5	(Inspect tool position)	---	<ul style="list-style-type: none"> Visually inspect correctness of tool position 	
6	Press [Handwheel simulation]		<ul style="list-style-type: none"> Function is released and the indicator light lights off The machine tool continuously runs the subsequent programs until shutdown 	


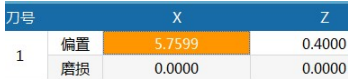

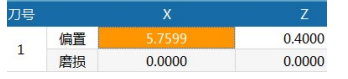
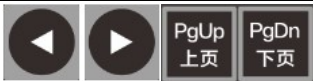


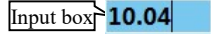

7.5 Machining information query

Operation name	Machining information query		Working mode	Automatic, manual, handwheel, homing, MDI
Basic requirements	---		Display interface	See Chapter 3 "Machining" interface
SN	Operation steps	Key	Description	
1	Press 『Machining』		<ul style="list-style-type: none"> ● Default interface, main menu 	
2	Press 『→』		<ul style="list-style-type: none"> ● Enter the extension menu 	
3	Press 『Machining information』	 	<ul style="list-style-type: none"> ● The system displays number of processed parts and information related to system operation time ● Enter the "Operation control" sub-interface and display operational statistical data 	
4	Press 『Presetting』		<ul style="list-style-type: none"> ● Set total number of processed parts, and so on 	
5	Press 『Reset』		<ul style="list-style-type: none"> ● Reset time and number of processed parts 	
6	Press 『Operation statistics』		<ul style="list-style-type: none"> ● Export or eliminate machining information file 	


8 Tool setting and machining setup








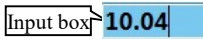

8.1 Directly input tool compensation value

8.1.1 Directly input tool compensation value under "Machining" function set

Operation name	Input tool compensation value for trial cut	Working mode	Manual, handwheel
Basic requirements	Manual or handwheel mode	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set
SN	Operation steps	Key	Description
1	Press [Machining]		<ul style="list-style-type: none"> Enter "Tool compensation table" 
2	Press [Tool compensation setup]		<ul style="list-style-type: none"> Enter "Tool compensation table" 
3	Press [Cursor] or [PgUp/PgDn]		<ul style="list-style-type: none"> Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set
4	Press [Enter]		<ul style="list-style-type: none"> Activate input box 
5	(Input offset value)	---	<ul style="list-style-type: none"> Input "10.04" 
6	Press [Enter]		<ul style="list-style-type: none"> Confirm input Skip to Z axis offset position of the tool

8.1.2 Directly input tool compensation value under "Setup" function set

Operation name	Directly input tool compensation value	Working mode	Manual, handwheel
Basic requirements	Manual or handwheel mode	Display interface	3.3 "Setup" function set interface
SN	Operation steps	Key	Description
1	Press [Setup]		<ul style="list-style-type: none"> "Setup" default interface, main menu



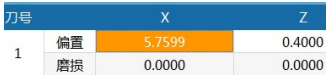

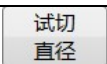

2	Press 『 Tool compensation setup 』		<ul style="list-style-type: none">Enter "Tool compensation table"	<table border="1"><thead><tr><th>刀号</th><th></th><th>X</th><th>Z</th></tr></thead><tbody><tr><td>1</td><td>偏置</td><td>5.7599</td><td>0.4000</td></tr><tr><td></td><td>磨损</td><td>0.0000</td><td>0.0000</td></tr></tbody></table>	刀号		X	Z	1	偏置	5.7599	0.4000		磨损	0.0000	0.0000
刀号		X	Z													
1	偏置	5.7599	0.4000													
	磨损	0.0000	0.0000													
3	Press 「Cursor」 or 「PgUp/PgDn」	   	<ul style="list-style-type: none">Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set													
4	Press 「Enter」		<ul style="list-style-type: none">Activate input box													
5	(Input offset value)	---	<ul style="list-style-type: none">Input “10.04”													
6	Press 「Enter」		<ul style="list-style-type: none">Confirm inputSkip to Z axis offset position of the tool													

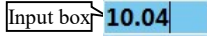




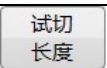
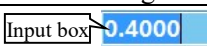


Notices:

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

8.2 Tool setting for trial cut

8.2.1 Tool setting for trial cut under "Machining" function set


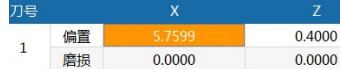




Operation name		Input tool compensation value for trial cut		Working mode	Manual, handwheel
Basic requirements		Manual or handwheel mode		Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set
SN	Operation steps	Key		Description	
1	Press 『Machining』			● "Tool compensation setup" sub-interface	
2	Press 『 Tool compensation setup』			● Enter "Tool compensation table"	
					
3	Press 「Cursor」 or 「PgUp/PgDn」			● Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set	
4	(Outer diameter of trial cut workpiece)	---		● After trail cut of workpiece, exit along A axis (for ease of measurement). In order to obtain X coordinate of the tool under the workpiece coordinate system	
5	(Measure outer diameter of workpiece)	---		● Obtain X coordinate value of the tool under the workpiece coordinate system through measurement. e.g.: 10.04	
6	Press 『 Trial cut			● Activate input box	
					

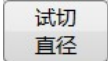

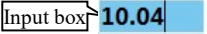




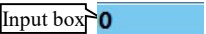

	diameter」		
7	(Input the measured value)	---	<ul style="list-style-type: none"> Input "10.04" 
8	Press 「Enter」		<ul style="list-style-type: none"> Confirm input and clear wear value under tool number. The input box exits from active state
9	Press 「Cursor」 or 「PgUp/PgDn」	  	<ul style="list-style-type: none"> The cursor moves to the next setup position. e.g.: Set Z axis offset of tool 1
10	(End face of trial cut workpiece)	---	<ul style="list-style-type: none"> After trial cut of workpiece, exit along X axis (for ease of measurement) in order to obtain Z coordinate of the tool under the workpiece coordinate system.
11	(Measure length of workpiece)	---	<ul style="list-style-type: none"> Obtain Z coordinate value of the tool under the workpiece coordinate system through measurement. If the zero point of the workpiece is on the front end and the end face is the front end, Z axis coordinate value of the tool in the workpiece coordinate system is 0 This value may be positive or negative
12	Press 「Trial cut length」		<ul style="list-style-type: none"> Activate input box 
13	(Input the measured value)		<ul style="list-style-type: none"> Input "0" 
14	Press 「Enter」		<ul style="list-style-type: none"> Confirm input and clear wear value under tool number The input box exits from active state

Notices:

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

8.2.2 Tool setting for trial cut under "Setup" function set

Operation name	Input tool compensation value for trial cut	Working mode	Manual, handwheel
Basic requirements	Manual or handwheel mode	Display interface	3.3 "Setup" function set interface
SN	Operation steps	Key	Description
1	Press 「Setup」		<ul style="list-style-type: none"> Enter "Tool compensation table" 
2	Press 「Cursor」	   	<ul style="list-style-type: none"> Move the cursor to the setup position. e.g.: X axis offset of tool 1 should be set
3	(Outer diameter of trial cut	---	<ul style="list-style-type: none"> After trial cut of workpiece, exit along A axis (for ease of measurement). In order to obtain X




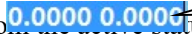


	workpiece)		coordinate of the tool under the workpiece coordinate system
4	(Measure outer diameter of workpiece)	---	<ul style="list-style-type: none"> Obtain X coordinate value of the tool under the workpiece coordinate system through measurement. e.g.: 10.04
5	Press 『Trial cut diameter』		<ul style="list-style-type: none"> Activate input box 
6	(Input the measured value)	---	<ul style="list-style-type: none"> Input "10.04" 
7	Press 「Enter」		<ul style="list-style-type: none"> Confirm input and clear wear value under tool number. The input box exits from active state
8	Press 「Cursor」		<ul style="list-style-type: none"> The cursor moves to the next setup position. e.g.: Set Z axis offset of tool 1
9	(End face of trial cut workpiece)	---	<ul style="list-style-type: none"> After trial cut of workpiece, exit along X axis (for ease of measurement) in order to obtain Z coordinate of the tool under the workpiece coordinate system.
10	(Measure length of workpiece)	---	<ul style="list-style-type: none"> Obtain Z coordinate value of the tool under the workpiece coordinate system through measurement. If the zero point of the workpiece is on the front end and the end face is the front end, Z axis coordinate value of the tool in the workpiece coordinate system is 0 This value may be positive or negative
11	Press 『Trial cut length』		<ul style="list-style-type: none"> Activate input box 
12	(Input the measured value)		<ul style="list-style-type: none"> Input "0" 
13	Press 「Enter」		<ul style="list-style-type: none"> Confirm input and clear wear value under tool number The input box exits from active state

Notices:

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

8.3 Overall translation of tool rest




8.3.1 Overall translation of tool rest under "Machining" function set



Operation name	Overall translation of tool rest	Working mode	Manual, handwheel
Basic requirements	The machine tool is at stop state	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set
SN	Operation steps	Key	Description
1	Press 【Machining】		<ul style="list-style-type: none"> "Tool compensation setup" sub-interface
2	Press 【Tool compensation setup】		<ul style="list-style-type: none"> Enter "Tool compensation table"
3	Press 【Translation of tool rest】		<ul style="list-style-type: none"> Activate the dialog box Press 「Esc」 to exit from 
4	(Input translation amount)		<ul style="list-style-type: none"> Separate the value between two coordinates by Space e.g.: X/Z offset is 0.02/0.01, fill in 0.02 space 0.01
5	Press 「Enter」		<ul style="list-style-type: none"> Confirm input and all tool compensation amounts translate for the set value

Notices:

- Confirm "Translation of tool rest" and the original wear value will not be cleared;
- Tool test translation function cannot be used during program run.

8.3.2 Overall translation of tool rest under "Setup" function set

Operation name	Overall translation of tool rest	Working mode	Manual, handwheel
Basic requirements	The machine tool is at stop state	Display interface	3.3 "Setup" function set interface
SN	Operation steps	Key	Description
1	Press 【Setup】		<ul style="list-style-type: none"> "Tool compensation setup" sub-interface
2	Press 【Translation of tool rest】		<ul style="list-style-type: none"> Activate the dialog box  Press 「Esc」 to exit from the active state

3	(Input translation amount)		<ul style="list-style-type: none"> ● Separate the value between two coordinates by Space ● e.g.: X/Z offset is 0.02/0.01, fill in 0.02 space 0.01
4	Press 「Enter」		<ul style="list-style-type: none"> ● Confirm input and all tool compensation amounts translate for a certain distance



Notices:

- Confirm "Translation of tool rest" and the original wear value will not be cleared.

8.4 F/S machining setup

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

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












Operation name	Machining configuration	Working mode	Manual, automatic
Basic requirements	When there is no F/S instruction in machining program	Display interface	See Chapter 3 "Machining" interface
SN	Operation steps	Key	Description
1	Press 『Machining』		<ul style="list-style-type: none"> ● Switch to the machining interface
2	Press 『Machining configuration』		<ul style="list-style-type: none"> ● F and S setup menus pop up
3	(Set machining F and S values)		<ul style="list-style-type: none"> ● If there are no F and S in machining program, this value prevails ● Set default S, namely rotate speed under manual mode


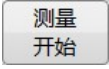




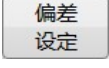
Notices:

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

8.5 Tool measurement

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

Operation name	Tool measurement	Working mode	(Handwheel manual) and (automatic)
Basic requirements	The machine tool is configured with tool setting gage	Display interface	See 3.3.4 3 "Tool measurement" sub-interface
SN	Operation steps	Key	Description
1. Preparation for tool measurement			
1	Press 【Handwheel】		<ul style="list-style-type: none"> Under handwheel mode: Touching tool setting gage, tool setting and other operations are completed
2	Press 【Setup】		<ul style="list-style-type: none"> Set tool measurement in the second page of the interface
3	Press 【→】		<ul style="list-style-type: none"> Enter the extension menu
4	Press 【Tool measurement】		<ul style="list-style-type: none"> Enter the tool measurement sub-interface
5	Press 【Measurement parameter】		<ul style="list-style-type: none"> Enter the measurement parameter sub-interface Fill length and width of measuring instrument, calibration of tool nose direction, measurement times, measurement speed and trigger speed (which must be filled. See 3.3.4 for specific definition)
6	Press 【Cursor】		<ul style="list-style-type: none"> Press 【Cursor】 to items to be filled Press 【Enter】 to activate input
7	Press 【Enter】		
8	(Fill in measurement parameter)	---	---
9	Press 【Enter】		<ul style="list-style-type: none"> Press 【Enter】 to confirm input Press 【Reset】 to abandon input
2. Tool calibration			
10	Press 【Lathe tool calibration】		<ul style="list-style-type: none"> Enter the lathe tool calibration sub-interface Calibration is the precondition of measurement
11	Press 【Cursor】		<ul style="list-style-type: none"> Move the cursor to the position of measuring instrument <ul style="list-style-type: none"> ➢ Z axis moves to P1 ➢ X axis moves to P2 The tool moves to P point <ul style="list-style-type: none"> P1绝对位置 (Z向) P2绝对位置 (X向) P3绝对位置 (Z向) P4绝对位置 (X向) Press 【Enter】 to confirm position and generate the initial value of tool
12	(Move the tool manually)	---	
13	Press 【Enter】		












			setting
14	Press 【Automatic】		<ul style="list-style-type: none"> ● Switch working mode to "Automatic"
15	Press 『 Start measurement』		<ul style="list-style-type: none"> ● Press 『 Start measurement』 and a prompt message Inspect the cursor position will be given ● When tool offset is greater than 50mm, collision should be avoided during measurement.
16	【Cycle start】		<ul style="list-style-type: none"> ● The tool starts tool setting automatically as per the set value of "Measurement parameter" ● Calibrate the standard tool automatically and determine the positional relationship between the standard tool and the measuring instrument.
3. Automatic measurement, determine standard tool and the position among all tools			
17	Press 『 Lathe tool measurement』		<ul style="list-style-type: none"> ● Enter the lathe tool measurement sub-interface
18	(Set the tool for automatic measurement)	---	<ul style="list-style-type: none"> ● Select the tool for automatic measurement by 「Cursor」 ● Activate “Tool nose orientation” and “ON/OFF” by 「Enter」 and input
19	Press 『 Start measurement』		<ul style="list-style-type: none"> ● Press 『 Start measurement』 and a prompt message Inspect the cursor position will be given ● When tool offset is greater than 50mm, collision should be avoided during measurement.
20	Press [Cycle start]		<ul style="list-style-type: none"> ● The tool starts tool setting automatically as per the set value of "Measurement parameter" ● Generate relative position value among measurement tools automatically
4. Set the positional relationship between the standard tool and the workpiece			
21	Press 『 Offset setup』		<ul style="list-style-type: none"> ● Enter the offset setup interface ● Press 「Cursor」 to select set parameters and 「Enter」 to activate and input parameter value
22	(Set "Standard tool number")	---	<ul style="list-style-type: none"> ● Select a measurement tool as the standard tool and fill in tool number
23	(Set "Whether to use the original tool offset")	---	<ul style="list-style-type: none"> ● If the standard tool has been set and the offset value should be adopted, fill in "1" ● If the standard tool is not set or the value is not adopted, fill in "0"
24	(If the original tool offset is not used, tool setting is needed)	---	<ul style="list-style-type: none"> ● If "Whether to use the original tool offset value" is set as "0", tool setting should be completed under this interface. ● Working mode should be switched to handwheel during tool setting.

Notices:

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

8.6 Thread repair function

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

Operation name	Thread repair	Working mode	(Manual, handwheel) and (automatic)
Basic requirements	Tool setting is needed for the newly clamped thread parts	Display interface	See Chapter 3 "Setup" interface
SN	Operation steps	Key	Description
1	Press 【Setup】		● Enter the "Setup" function set, default interface
2	Press 【→】		● Enter the extension menu
3	Press 【Thread repair】		● Enter the thread repair sub-interface
4	Press 【Invalid re-cutting】		● The spindle cannot rotate then
5	Swing 【Handwheel】	 or 	● Swing Z/X axis of the machine tool and align the tool nose at the peak of the thread groove
6	Press 【Set all axes】		● Input coordinate values of "Guide axis/spindle" of the tool under the machine tool coordinate system automatically. 
	Or press 【Input】		● Input coordinate values of "Guide axis/spindle" of the tool under the machine tool coordinate system manually.
7	Press 【Invalid re-cutting】		● For thread repair and machining
8	Press 【Automatic】		<ul style="list-style-type: none"> ● Run thread programs and repair thread parts ● X coordinate value of thread programs should be consistent with bottom diameter of repaired thread
	(Thread repair and machining)	---	● A certain allowance is reserved when repaired for the first time and reach repair requirements after repeated override


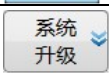
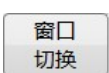


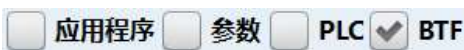
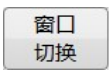


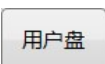


Notices:

- Before thread repair, detect following error when the guide axis moves at 1000mm/min and fill in the value of axis parameter 102049.
- While setup thread repair parameter, "Invalid re-cutting" in the submenu must be pressed to set tool setting or clear data;
- After thread repair parameter is set, "Valid re-cutting" must be pressed to realize thread repair function;
- Thread repair function can be used to repair single-start thread only.

9 Machine tool commissioning

9.1 System upgrade



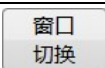

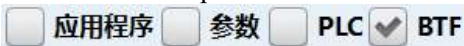









9.1.1 System upgrade

Operation name	System upgrade	Working mode	Emergency stop
Basic requirements	<ul style="list-style-type: none"> The machine tool must be at "Emergency stop" state System upgrade patch file name is "***.BTF" 	Display interface	3.3 "Maintain" function set interface
SN	Operation steps	Key	Description
1	Press [[Maintain]]		<ul style="list-style-type: none"> Enter the "Maintain" function set interface
2	Press [[System upgrade]]		<ul style="list-style-type: none"> Enter the "System upgrade" sub-interface
3	Press [[Window switch]]		<ul style="list-style-type: none"> Switch to "Upgrade selection" window above the interface
4	Press [[Cursor]]		<ul style="list-style-type: none"> Select upgrade item Select BTF item for comprehensive upgrade
5	Press [[Enter]]		<ul style="list-style-type: none"> Confirm the selected item 
6	Press [[Window switch]]		<ul style="list-style-type: none"> Switch to the upgrade patch file source selection window below the interface 
7	Press [[USB flash disk]] / [[User disk]]	 	<ul style="list-style-type: none"> Select upgrade patch file from USB flash disk by default Upgrade patch file in the user disk also can be selected
8	Press [[Cursor]]		<ul style="list-style-type: none"> Select upgrade file package Upgrade patch file name must be BTF file
9	Press [[Enter]]		<ul style="list-style-type: none"> Start system upgrade Do not power off before upgrade is completed

Notices

- Upgrade should be conducted with system administrator permission and system upgrade is often conducted by HNC CNC technical personnel;
- Do not power off during system upgrade

9.1.2 System backup

Operation name	System backup	Working mode	Emergency stop
Basic requirements	The machine tool must be at "Emergency stop" state	Display interface	3.3 "Maintain" function set interface
SN	Operation steps	Key	Description
1	Press 『Maintain』		● Enter the "Maintain" function set interface
2	Press 『System upgrade』		● Enter the "System upgrade" sub-interface
3	Press 『Window switch』		● Switch to upgrade patch window above the interface
4	Press 『Cursor』		● Select backup 
5	Press 『Enter』		● Confirm to select
6	Press 『Cursor』		● Select backup 
7	Press 『Enter』		● Confirm backup 
8	Press 『Window switch』		● Switch the cursor to the backup target disk window
7	Press 『USB flash disk』 / 『User disk』		● Default back target disk is 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

Notices:













- While backing up a file to the system disk, size of system disk should be noted and backup package of V2.40.00 version software is about 200MB

9.2 Batch commissioning








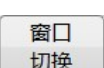




This function is limited to permission and it can be used for permissions for batch commissioning. This function can load/back up PLC, fixed cycle, parameter, G code, parameter configuration and other files necessary for commissioning separately/by batches.

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

9.2.1 Batch load commissioning

Operation name	Batch load commissioning		Working mode	Emergency stop
Basic requirements	The machine tool must be at "Emergency stop" state Files loaded by batches must be .tar file		Display interface	3.3 "Maintain" function set interface
SN	Operation steps	Key	Description	
1	Press 『Maintain』		● Enter the "Maintain" function set interface	
2	Press 『Batch commissioning』		● Enter the "Batch commissioning" sub-interface	
3	Press 『Window switch』		● Switch to system disk ● Enter file type selection window	
4	Press 「Cursor」		● Move the cursor to type of the file to be loaded	
5	Press 「Enter」		● Confirm selection type	
6	Press 『USB flash disk』 / 『User disk』	 	● Select the source disk of the loaded file	
7	Press 『Window switch』		● Switch the cursor to the file source disk window	
8	Press 「Cursor」		● Move the cursor to type of the file to be loaded	
9	Press 『Load』		● A prompt message "Whether to load file XXXX.tar?(Y/N)"	
10	Press 「Y」 or 「Enter」	 or 	● A prompt message "Loading a file succeeds, please power off and restart!"	

9.2.2 Batch backup commissioning

Operation name	Batch backup commissioning		Working mode	Emergency stop
Basic requirements	The machine tool must be at "Emergency stop" state		Display interface	3.3 "Maintain" function set interface
SN	Operation steps	Key	Description	
1	Press 『Maintain』		<ul style="list-style-type: none"> Enter the "Maintain" function set interface 	
2	Press 『Batch commissioning』		<ul style="list-style-type: none"> Enter the "Batch commissioning" sub-interface 	
3	Press 『Window switch』		<ul style="list-style-type: none"> Switch to system disk Enter file type selection window 	
4	Press 『Cursor』		<ul style="list-style-type: none"> Move the cursor to type of the file to be backed up 	
5	Press 『Enter』		<ul style="list-style-type: none"> Confirm selection type 	
6	Press 『USB flash disk』 / 『User disk』	 	<ul style="list-style-type: none"> Select a target disk to back up files 	
7	Press 『Window switch』		<ul style="list-style-type: none"> Switch the cursor to the file source disk window 	
8	Press 『Cursor』		<ul style="list-style-type: none"> Move the cursor to the file directory to be loaded 	
9	Press 『Backup』		<ul style="list-style-type: none"> A prompt message "Please enter a backup package name" will be given 	
10	(Enter a backup package name)	---	<ul style="list-style-type: none"> File name must be suffixed with .tar 	
11	Press 『Y』 or 『Enter』	 or 	<ul style="list-style-type: none"> Complete backup and give a prompt message Backup succeeds 	

Attention: 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 instruction position of machine tool axis. This function can decrease error of actual position and instruction position through increasing or decreasing actual displacement of machine tool.

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

9.3.1 Generation of pitch error compensation data file

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


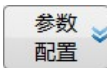
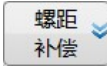
误差补偿表格	
图表类型	各自补偿
补偿类型	绝对值
补偿分辨率	1
正负符号转换(+/-)	补偿值
参考点位置	0.000
补偿起点	0.000

- "Chart type" **must** select "Separate compensation";
- "Compensation type" **must** select "Absolute value";
- "Compensation resolution" must be "1"um;
- "Conversion of positive and negative symbols (+/-)" **must** select "Compensation value";
- "Reference point position" **must be "0"**;
- Machine tool coordinate position with "Compensation start point" as the compensation start point, **it must be 0**;
- Machine tool coordinate position with "Compensation end point" as the compensation end point;
- "Compensation interval" is compensation interval and it must be a positive value.

9.3.2 Operation of thread repair error compensation sub-interface

HNC-818D-TU system can enter this function sub-interface under "Machining" and "Maintain" function sets.


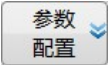
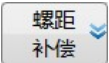

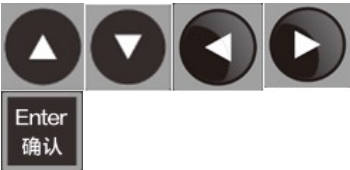






Enter the pitch error compensation sub-interface under the "Maintain" function set

Operation name	Operation of thread repair sub-interface	Working mode	Automatic, single segment, manual, increment
Basic requirements	The machine tool is at stop state	Display interface	3.2 "Tool compensation setup" sub-interface under "Machining" function set
SN	Operation steps	Key	Description
1	Press 【Maintain】		● "Tool compensation setup" sub-interface
2	Press 【Parameter setup】		● Enter the "Parameter setup" 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 thread repair file generated by Renishaw software).

Operation name	Import of thread repair file		Working mode	Automatic, single segment, manual, increment
Basic requirements	Pitch error compensation data file has been generated correctly		Display interface	3.2 "Parameter setup" sub-interface under "Maintain" set
SN	Operation steps	Key	Description	
1	Press [Maintain]		● "Maintain" main menu	
2	Press [Parameter setup]		● "Parameter setup" sub-interface and menu	
3	Press [Pitch compensation]		● "Pitch compensation" sub-interface and menu (as shown above)	
4	Press [Cursor]		● Select compensation axis	
5	Press [Cursor] or [Enter]		<ul style="list-style-type: none"> ● Setup option of direction key (options in the above area 1, see Remarks for specific options) ● Press [Enter] to confirm option or setup 	
6	Press [One-key clear]		<ul style="list-style-type: none"> ● If thread compensation data is not imported for the first time, please press One-key clear to clear thread compensation data ● If thread compensation data is imported for the first time, this step can be omitted 	
7	Press [Load rtl]		<ul style="list-style-type: none"> ● Enter "Error compensation data file (*.rtl)" for search ● Select thread compensation data file of corresponding axis 	
8	Press [USB flash disk] / [User disk]		● Select compensation data file disk	
9	Press [Cursor]		● Select compensation data file	
10	Press [Enter]		● Press [Enter] to import thread compensation data	
11	Press [Reset]		● After thread compensation data is imported successfully, press Reset to validate them	
12	(Inspect thread compensation data)		● Inspect thread compensation type, start point position, compensation point number, compensation interval and initial number of	

			data sheet ● During unidirectional compensation type, inspect whether backlash compensation is enabled and backlash value ● thread compensation data is imported incorrectly, execute from step 4 again
--	--	--	---

Remarks: Select options of area 1

- Selection of compensation axis: Axis 0, axis 1 and axis 2.....;
- Pitch error compensation type: Forbidden, unidirectional compensation, bidirectional compensation;
- Start point position : The same as "Renishaw error compensation table";
- Compensation interval : The same as "Renishaw error compensation table";
- Compensation point number : The same as "Renishaw error compensation table";
- Initial number of data sheet : Initial number of system division 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 for HNC-808Di-TU CNC system 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-condensable
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 normal 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 screen of fan

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