In this manual, we will try to describe various matters related to the operation of the system. Due to space limitations and the specific use of the product, it is not possible to describe in detail all the operations that must be done and/or cannot be done in the system. Therefore, anything not specifically specified in this manual is considered to be "impossible" or "impermissible".

preface

Dear Customer:

The company is deeply honored and grateful for the products you choose!

This user manual details the programming, operation and installation of CNC systems for machining centers. In order to ensure the safe, normal and effective operation of the product, please be sure to carefully read this manual before installing and using the product.

Security warnings

 Improper operation will cause accidents, and personnel with appropriate qualifications must be required to operate this system.

Special note: The system power supply installed on the chassis (inside) is a special power supply provided only for the numerical control system manufactured by our company. Users are prohibited from using this power supply for other purposes. Otherwise, there will be great danger!

Security warnings

■ Transportation and storage
1 . The stacking of product packaging boxes should not exceed six layers;
2 . Do not climb, stand or place heavy objects on the product box;
3 . Do not drag or handle the product using the cable connected to the product;
4 . It is strictly forbidden to bump and scratch the panel and display;
5 . The product packaging box should avoid moisture, exposure to the sun and rain.

■ Unpacking inspection
1 . After opening the package, please confirm whether it is the product you purchased;
2 . Check whether the product is damaged during transportation;
3 . Check whether each component is complete and whether there is any damage according to the list;
4. If there is any discrepancy in the product model, missing accessories or transportation damage, please contact the company in time.

■ Wiring
1 . The personnel participating in the wiring and inspection must be professionals with corresponding capabilities;
2. The product must be reliably grounded, the grounding resistance should be less than 0.1Ω, and the neutral wire (neutral wire) cannot be used instead of the ground wire;
3 . The wiring must be correct and firm to avoid product failure or unintended consequences;
4 . The surge absorbing diode connected to the product must be connected in the specified direction, otherwise the product will be damaged;
5 . Before plugging in or unplugging the product or opening the product chassis, the product must be powered off.

■ Overhaul
1 . The power must be cut off before servicing or replacing components;
2 . When a short circuit or overload occurs, the fault should be checked and restarted only after troubleshooting;
3. Do not power off the product frequently, if it is necessary to re-power after

the power off, the interval time is at least 1min.

statement

This manual describes as many different things as possible, but because there are too many possibilities involved to describe all the operations that can or cannot be performed, the contents not specifically stated in this manual can be considered unusable.

warn

Before installing, connecting, programming and operating this product, you must read this product manual and the instruction manual of the machine tool manufacturer in detail, and strictly follow the requirements of the manual and manual to carry out relevant operations, otherwise it may lead to damage to the product and machine tool, scrapping of the workpiece and even personal injury.

The product functions and technical indicators (such as accuracy, speed, etc.) described in this manual are only for this product, the CNC

machine tool installed in this product, the actual functional configuration and technical performance are determined by the design of the machine tool manufacturer, and the functional configuration and technical indicators of the CNC machine tool are subject to the instruction manual of the machine tool manufacturer; Although the system is equipped with a standard machine operator panel, the functions of the buttons of the standard machine panel are defined by the PLC program (ladder diagram). Please note that the functions of the buttons in the machine panel in this manual are described for standard PLC procedures!

*** The contents of this manual are subject to change without notice.**

Manufacturer's responsibility for safety

- The manufacturer shall be responsible for the hazards that have been eliminated and/ or controlled in the design and construction of the supplied CNC system and the accessories supplied with it.

——The manufacturer shall be responsible for the safety of the CNC system provided and the accessories supplied with it.

——The manufacturer shall be responsible for the use information and suggestions provided to the user.

User's responsibility for safety

——The user should learn and train the safe operation of the numerical

control system, and be familiar with and master the content of the safe operation.

——The user shall be responsible for the safety and risks caused by adding, changing or modifying the original numerical control system and accessories.

- The user shall be responsible for the hazards caused by failure to operate, adjust, maintain, install and store and transport the product in accordance with the provisions of the manual.

*** This manual is in the collection of end users. Sincerely thank you for your friendly support to our company in our products!**

Part II

Operating instructions

directory

Chapter 1 Operation mode and display interface

1.1 Panel division

The machining center system adopts a variety of operation panels, and the composition of the panels is basically the same, which can be roughly divided into 4 parts;

Part 1: Display menu, function menu keys (position, program, compensation and other keys), such buttons are processed by the system software, not controlled by PLC, used to switch different interfaces;

Part 2: MDI panel (editing keyboard), containing programming code word keys (XYZMST and other letter keys), number keys (0-9 keys), symbol buttons (+-etc.), Such keys are handled by the system software, not controlled by PLC, and are used to edit programs and modify parameters;

Part 3: Machine tool panel, including operation mode switching buttons (manual, handwheel, automatic, etc.), magnification adjustment buttons (feed magnification, spindle magnification, fast magnification, handwheel magnification adjustment button), manual control buttons, auxiliary function buttons (single-segment, skip-section, select-stop, spindle control, cooling, lubrication, pause, start, etc.), such buttons are all controlled by the system PLC control, for switching operating modes and manual control of auxiliary functions;

Part 4: Indicator lights, including zero return light for each axis and machine status indicator light (three-color lamp).

Panel division

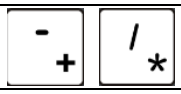
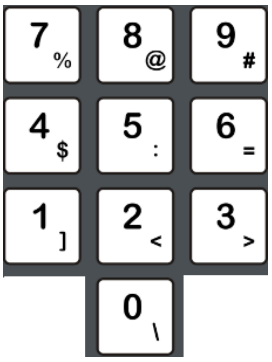
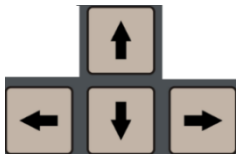
Note: Some machining center system models have different button arrangement positions.

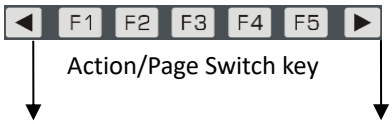
1.1.1 Status indication

	Shaft zero end indicator
	Tricolor lamp

1.1.2 Edit the keyboard

keystroke	name	Function description
	Reset key	CNC reset, feed, output stop, etc
	Address key	Address input

		Double address keys, press keys repeatedly, and switch between the two
	Symbol keys	Double address keys, repeatedly press the key, switch between the two
	Number keys	Digital input
	Enter the key	Determination of data input such as parameters, compensation quantities, etc
	Up key	Some model numbers and symbols are composite keys, and you can switch symbols after pressing the upper gear key
	Space bar	Switch input mode
	The first and last row keys	In the program interface, you can switch to the beginning and end of the program
	Cursor movement keys	Controls cursor movement
	Page turn key	Switch between pages on the same display screen

	Soft function keys	After using the function keys to switch the page set, you can use the corresponding software function keys to display the content of a sub-page in the current page, as shown below:  Return to the parent menu Move the menu key right The role of soft function keys: (1) Switch subpages in the current page set; (2) As an operation input for the currently displayed subpage, such as editing, modifying data, or displaying content, etc.
	Delete key	Remove programs, numbers, letters
	Left and right shift keys	The cursor moves quickly in the range

1.1.3 Display menu

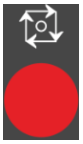
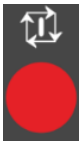
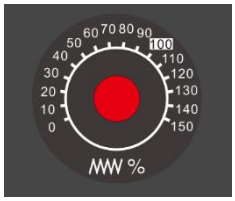
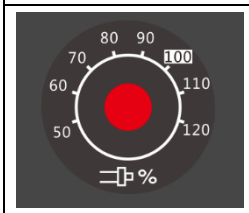
Menu key	remark
	Enter the location interface. The location interface has four surfaces: relative coordinates, absolute coordinates, comprehensive coordinates, and program supervision
	Enter the program interface. The program interface has four pages: program content, program status, program directory, and file directory
	Enter the compensation boundary and macro variable boundary, and repeatedly press the key to switch between the three interfaces. The compensation interface shows tool offset wear; The macro variables interface displays CNC macro variables
	Enter the program supervision interface. The program monitoring interface includes program editing, graphic settings, MDI, information/settings, workpiece origin, etc

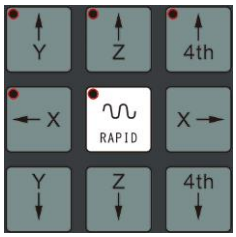
	Enter the CNC diagnosis interface, PLC status, PLC data, machine tool soft panel, version information interface. Repeated keystrokes can be used to switch between interfaces. CNC diagnostic interface, PLC status,PLC data display CNC internal signal status PLC all over the place Information on the status of address and data; The soft panel of the machine tool can be operated by the soft keyboard of the machine tool; The version information interface displays the version number of CNC software, hardware and PLC

1.1.4 Machine panel

The function of the buttons in the machine tool panel of the milling machine is defined by the PLC program (ladder diagram), and the specific function of each button can be found in the machine manufacturer's manual.

The functions of the buttons on the machine panel defined by the standard PLC program of the milling machine series are shown in the table below

keystroke	name	Function description	Mode of operation when the
	Feed hold key	Program and MDI code are suspended	Automatic method, input method, DNC method
	Cycle the start key	The program and MDI code run and start	Automatic method, input method, DNC method
	Feed magnification knob	Adjustment of feed rate	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Quick magnification key	Adjustment of fast movement speed	Automatic mode, input mode, machine tool zeroing, manual mode, DNC mode
	Spindle magnification knob	Spindle speed adjustment (spindle speed analog control mode is effective)	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode

	Manual tool change	Manual tool change	Machine tool zeroing, handwheel mode, single-step mode, manual mode
	Toggle the key	Spindle jogging status on/off	Machine tool zeroing, handwheel mode, single-step mode, manual mode
	Lubricate the key	Machine lubrication on/off	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	The coolant switch is on	Coolant on/off	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Blow on the key	Processing blow on/off	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Spindle control keys	The spindle rotates forward The spindle stops The spindle is reversed	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Quick switch	Fast speed/feed rate switching	Automatic mode, input method, manual method, DNC method
	X-axis feed key Y-axis feed key Z-axis feed key 4th axis feed key 5th axis feed key	Manual, single-step operation mode of forward/negative movement of each axis	Machine tool zeroing, single-step mode, manual mode
	Handwheel/step increment selection with quick magnification selection key	Handwheel movement per cell 1/10/100/1000 *Minimum equivalent single step movement per step 1/10/100/1000 *Minimum equivalent fast magnification Fo, 25%, F50%, F100 %	Automatic mode, input mode, machine tool zeroing method, handwheel mode, single step mode, manual mode, DNC mode

	Select Stop	When you select Stop is active, perform M01 Pause	Automatic method, input method, DNC method
	Single-stage switch	The program single-stage operation/continuous operation status is switched, and the single-stage operation indicator light is on when the single-stage is active	Automatic method, input method, DNC method
	Program segment selector switch	Whether the program segment marked with "/" at the beginning of the program segment skips the state switch, and the jump indicator light is on when the program segment selection switch is turned on	Automatic method, input method, DNC method
	The machine locks the switch	When the machine is locked, the machine lock indicator is on, and the feed shaft output is invalid	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Accessibility locks the switch	When the auxiliary function is locked, the auxiliary function lock indicator is on, and the output of the M, S, and T functions is invalid	Automatic method, input method, DNC method
	Spindle positioning switch	Turn spindle positioning on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Work light switch	Turn the work light on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode

	Chip conveyor switch	Turn the chip conveyor on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Custom output switches	Turn custom output on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Handwheel simulation development	Turn handwheel simulation on or off	Automatic method, input method, DNC method
	Automatic mode selection keys	Enter the automatic operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Select the key for the entry method	Enter the entry (MDI) operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Machine zero mode selection key	Enter the machine to return to zero operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Single-step/handwheel selection key	Enter single-step or handwheel operation mode (one of the two operation modes is selected by the	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Select keys manually	Enter the manual operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	DNC mode selects the key	Enter the DNC mode of operation	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Power-off memory key	Turn on the power-off memory function	Automatic method, input method, DNC method
	Tool changer back to zero key	Turn on the tool magazine zeroing function	The machine returns to zero
	Changer rotation key	Control the forward and reverse rotation of the tool magazine	Handwheel method, single-step method, manual mode

	Tool magazine advance and retreat key	Control the magazine forward and backward or the magazine inverted and returned	Handwheel method, single-step method, manual mode
	Tool magazine arm key	Controls the rotation of the disc magazine arm	Handwheel method, single-step method, manual mode

1.2 Overview of the mode of operation

The milling machine series has six operation modes, such as automatic, input, machine tool zeroing, handwheel, manual, and DNC mode.

- **Non-operational operation mode**

In the non-running operation mode, operations such as creating, deleting and modifying processing programs can be performed.

- **Automatic operation mode**

In automatic operation mode, run the program automatically.

- **Input operation mode**

In the input operation mode, you can enter parameters and enter and execute code snippets.

- **Machine tool return zero operation mode**

In the machine zero return operation mode, the feed axis return to the machine zero point operation can be performed separately.

- **Handwheel operation mode**

In handwheel feed mode, the CNC moves in selected increments.

- **Manual operation mode**

In the manual operation mode, manual feed, manual fast, feed rate adjustment, fast magnification adjustment, spindle start and stop, coolant switch, lubricating fluid switch, spindle jogging, manual loosening/ clamping and other operations can be performed.

- **DNC mode of operation**

In DNC operating mode, DNC runs programs.

1.3 Display interface

The machining center series has 5 interfaces, including position interface and program interface, and there are multiple display pages under each interface. Each interface (page) and operation mode are independent. The display menu, display interface and page hierarchy are shown in the figure below

Menu	Display the	Display the page

POS	Location interface	Coordinate switching--- relative zeroing--- workpiece origin --- tapping back
PROG FILE	Program interface	Perform machining--- find/replace--- file management--- area copy--- jump --- delete rows
OFFSET SETTING	Compensation interface	The coordinate system --- offset --- measured --- automatic tool set---- common variables --- system variables
	Macro variable interface	
DIAGNOSIS	CNC diagnosis	CNC diagnostic interface
	PLC diagnosis	PLC signal diagnostic interface
	IO diagnosis	Input/output diagnostic interface
	Version information	The system software version is displayed
	alarm	Switch to the alarm interface
	Spindle detection	Detect the spindle reaction time interface
	absolute value	Absolute values drive the monitoring interface
	Zero point detection	One turn signal back to zero monitoring interface
	CNC Help	Each interface is: operation table, alarm table, G code table, M code table, macro command
	Color setting	The program programming code displays the color setting

MON	Cheng Jian interface	Program editing--- graphic settings--- MDI --- information/settings--- workpiece origin--- tapping backback--- The cumulative number of cleared --- pieces is cleared --- man-hours are cleared

1.3.1 Location interface

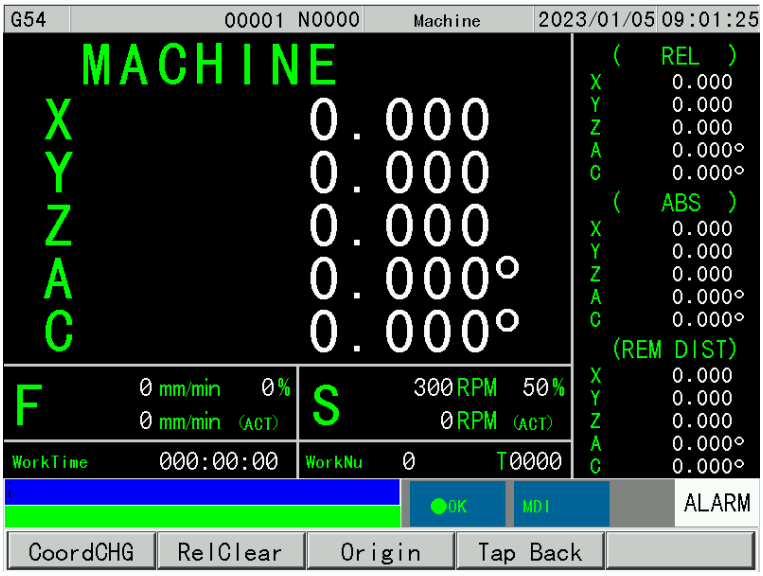
Press the POS key to enter the position interface, the position interface has coordinate switching (machine coordinates, relative coordinates, absolute coordinates, moving allowance), relative zeroing, workpiece origin, tapping back four pages, which can be viewed through the soft function button [F1-F4] key.

1) Mechanical coordinates display page

On the Mechanical Coordinates page, the F1 key toggles the display of machine coordinates, relative coordinates, absolute coordinates, and residual movement (the residual movement is only displayed in automatic and input mode).

The display value of the machine coordinate is the coordinate value of the current position in the machine tool coordinate system, which is established by returning to the zero point of the machine tool. The co-movement is the difference between the target position of the segment or MDI code and the current position.

The page is displayed as follows:



2) Display the page in relative coordinates

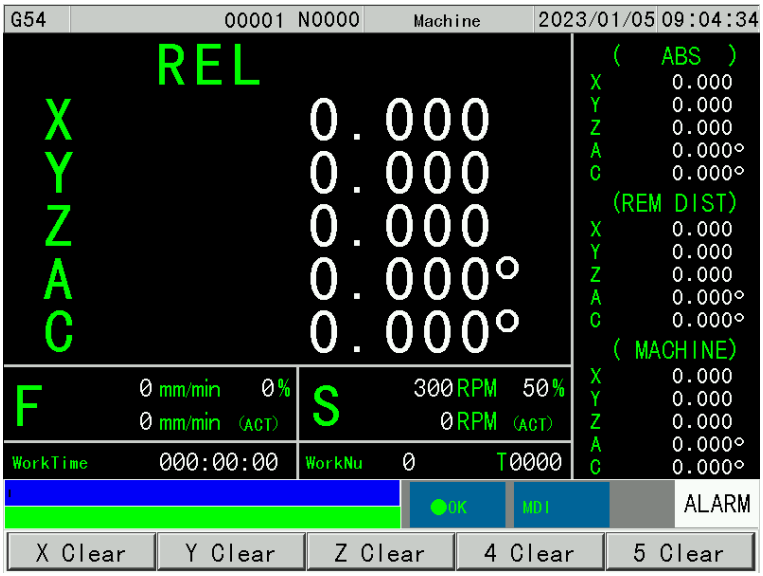
By switching the relative coordinates by the F1 key, the relative coordinate value displayed is the coordinate of the current position relative to the relative reference point, and the relative coordinates are maintained when the CNC is powered on. Relative coordinates can be zeroed out

at any time. After the relative coordinates are cleared, the current point is the relative reference point.

Under the relative coordinates interface, click the F2 key under the screen to enter the relative coordinates zero:

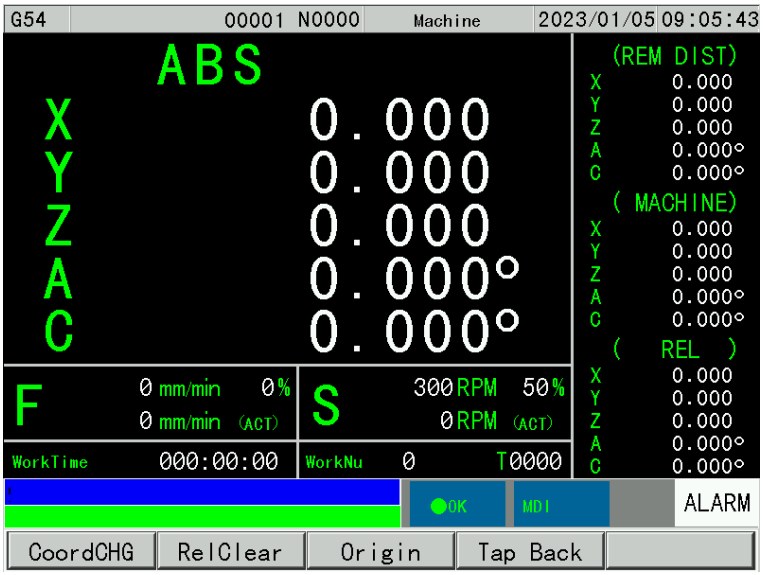
Under the relative coordinate display page, press the F1 key under the screen to clear the relative coordinate value of the X axis;
Under the relative coordinate display page, press theF2 key under the screen to clear the relative coordinate value of the Y axis;
Under the relative coordinate display page, press theF3 key under the screen to clear the relative coordinate value of the Z axis;
Under the relative coordinate display page, press theF4 key under the screen to clear the relative coordinate value of the 4-axis axis;
Under the relative coordinate display page, press the F5 key under the screen to clear the relative coordinate value of the 5-axis relative coordinate value;

Note: If 6th and 7th are valid, the zeroing method is the same as above.



3) Display the page in absolute coordinates

The displayed X, Y, Z coordinate values are the absolute position of the tool in the current workpiece coordinate system, and X, Y, Z when the CNC is powered on The coordinates are maintained, and the workpiece coordinate system is specified by G92, G54-G59.



Note: In editing, automatic, input, display "programming speed"; "Manual Speed" is displayed in machine zero, program zero, manual mode. Display "Handwheel Increment" in handwheel mode; Displays the Step Increment in single-step mode.

Actual speed: the actual processing speed after the feed rate calculation in the actual processing;
Number of processed pieces: when the program finishes M30 (or M99 in the main program), the number of processed pieces is increased by 1;
Cutting time: when the automatic operation starts the timing, the time unit is hours, minutes, seconds;
S0000: spindle speed feedback from spindle encoder, spindle encoder must be installed to display the actual speed of spindle;
Current Tool: The tool number specified by the T code in the program.
Tool offset: H00 Tool length compensation for the current machining program; D00 Radius compensation for the current machining program.
The number of machining pieces and cutting time power loss memory, the zero method is as follows:

The number of processed pieces is cleared: first press the 

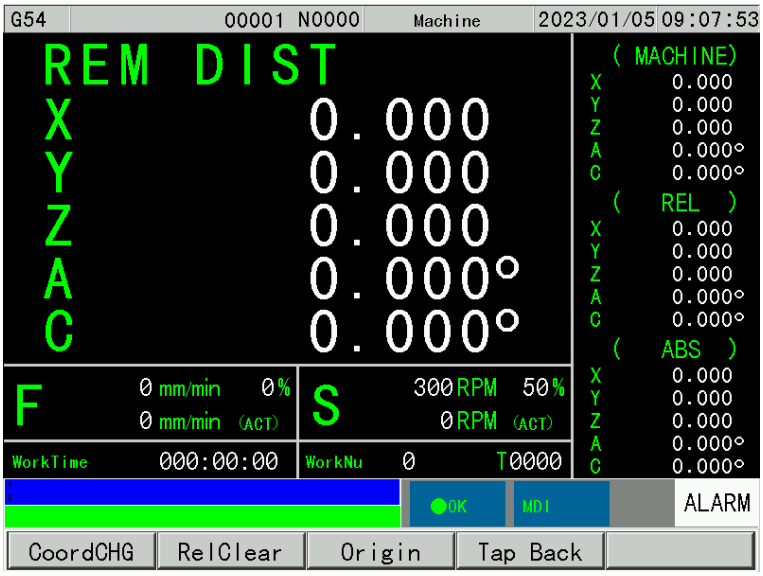
key, then press the right  button under the screen, and then press the F4 key under the screen [Zero Pieces] button to clear the number of pieces;

Cutting time zeroing: first press 

and hold the key, then press the right  button under the screen, and then press the F5 key [Working Hours Clear] button under the screen to clear the working hours;

4) Remaining Movement Display Page

On the co-movement display page, the displacement allowance for each axis of the current program segment is displayed at the same time.



1.3.2 Program Interface

Press the PROG FILE key to enter the program interface, the program interface has pages such as performing processing, find/replace, file management, area copying, jumping, deleting lines after page turning, absolute teaching, breakpoint start, tapping back, etc., and switching between pages through the F1-F5 buttons under the screen.

Program Content Edit Page:

On the program content editing page, the program content including the current program segment is displayed. Press PAGE ▼ keys, PAGE ▲ keys up, and down to view program content in non-run mode.

1) Program status page

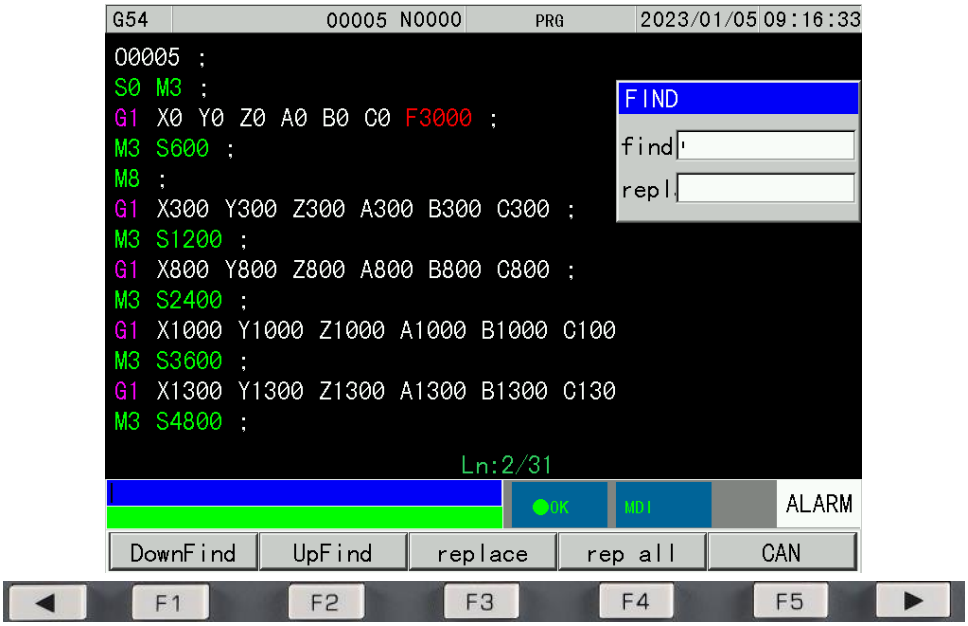
When the program content page is on, you can perform editing operations related to the program, such as processing, search/replace, area copying, jumping, deleting lines after page turning, and absolute teaching.

Program editing interface:

(Figure 1).

(Figure II).

Find/Replace interface:

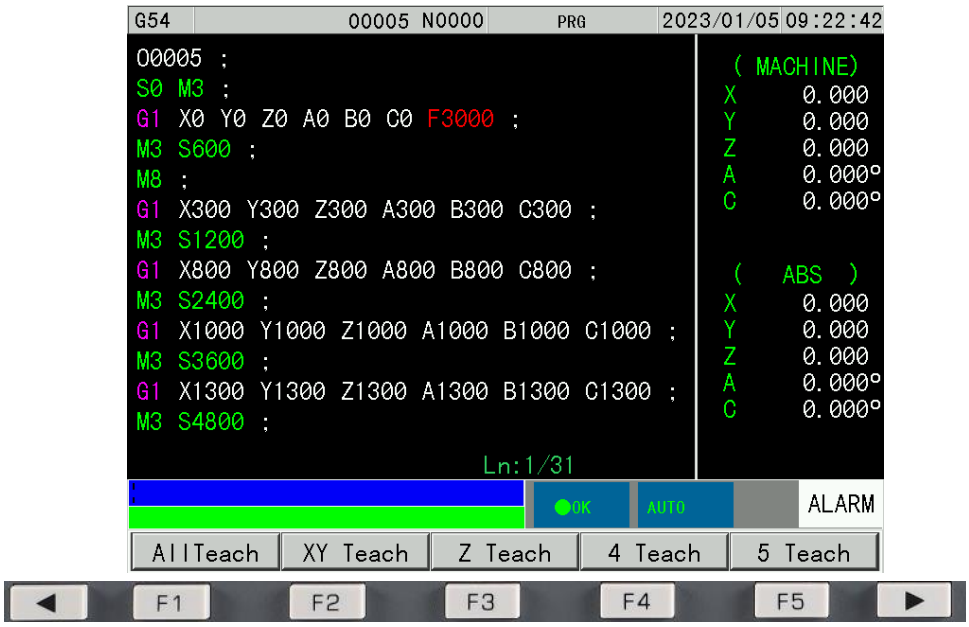


(Figure III).

Region Copy Interface:

(Figure 4)

Absolute teaching interface:



(Figure V).

2) File management page

Press the PROG FILE key to enter the program directory page, press the F3 key [File Management] under the screen to enter the file directory page, the page contains new, search, copy, U disk directory, perform processing, rename after page turning, delete, modify the number and other operation functions, you can enter the corresponding function through the F1-F5 keys under the screen 。

The file directory page lists all the processing programs, and in order to make it easier for

the user to find the program they want to select, the system displays the first 1 1 line of the current program on the right side of the page.

What the program catalog page displays:

(a) Number of existing programs:

Displays the number of programs (including subprograms) that have been deposited in the CNC

(b) Number of remaining procedures:

Displays the number of programs that can also be stored in the CNC

(c) Amount of storage used:

Displays the amount of storage (KB) occupied by the CNC's stored part program

(d) Remaining storage:

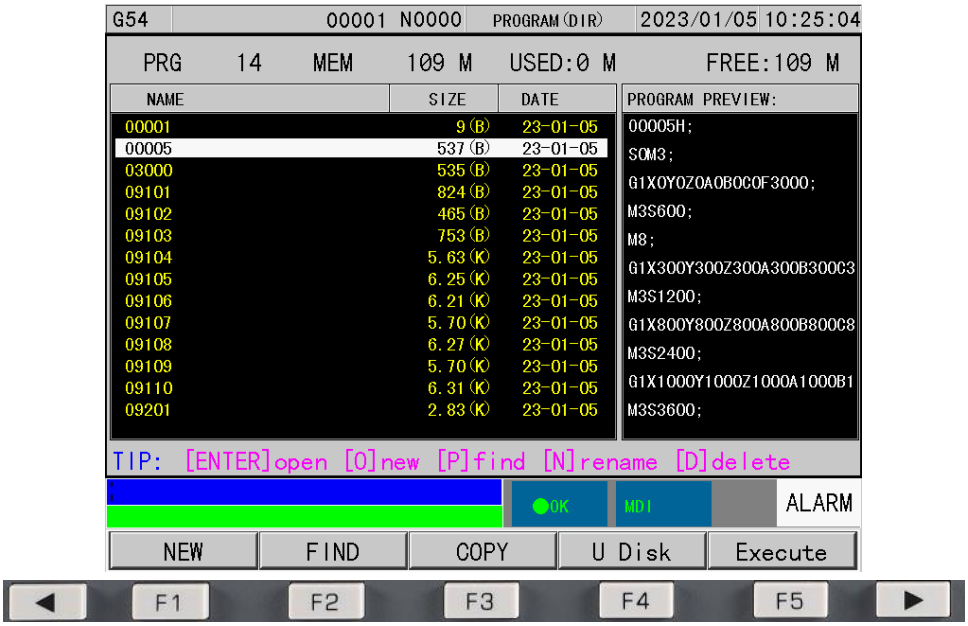
Displays the remaining capacity (KB) of the CNC storage part program

(e) Program Directory:

Displays the program number of the part program stored in order of the size of the part program name

(f) Program size:

Displays the amount of storage space occupied by the CNC program



3) USB stick directory page

Press the 

key to enter the program page, then press the F3 key [File Management] under the screen to enter the file directory page, and then press the F4 key [U disk directory] under the screen to enter the U disk directory interface.

Basic features:

F1 key: deposit U disk / deposit CNC, when switching between the system disk and the U disk, the corresponding display of "Save U disk" and "Save CNC";

F2 key: switch, switch between the system disk and the U disk, the purple border shows the

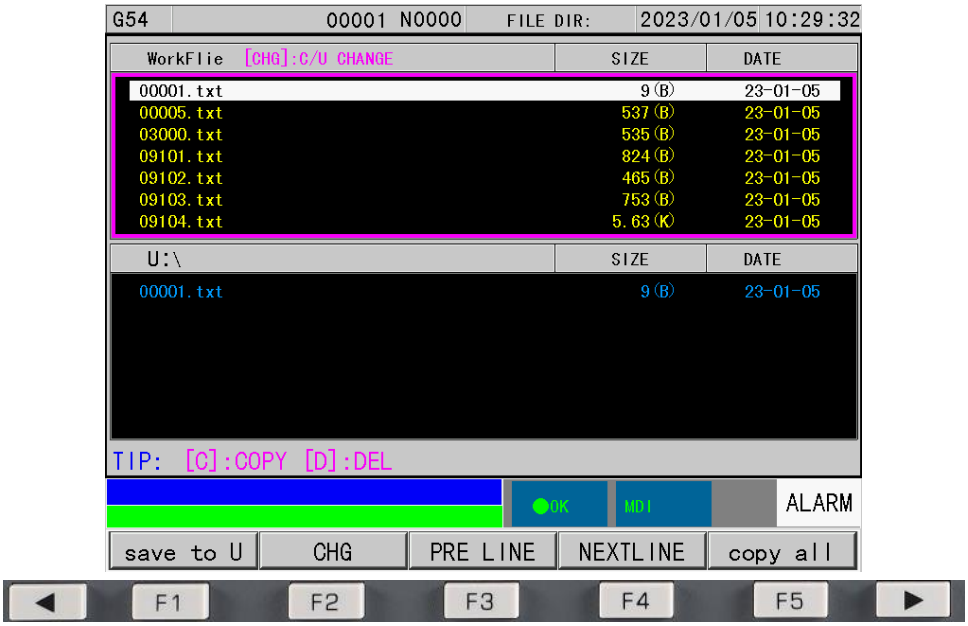
prompt;

F3 key: previous line, move up to select files in the selection disk;

F4 key: next line, move down the selection file in the selection disk;

F5 key: copy all, copy all the files in the selected disk to another selected disk;

The page is displayed as follows:



1.3.3 Compensation, macro variable interface

Under the key offset knife supplement interface, there are functions of coordinate system, bias, mid-point measurement, automatic tool alignment, common variables, and system variables after page turning, corresponding to pressing the F1-F5 keys under the screen to enter the relevant function interface.

1) Coordinate system interface

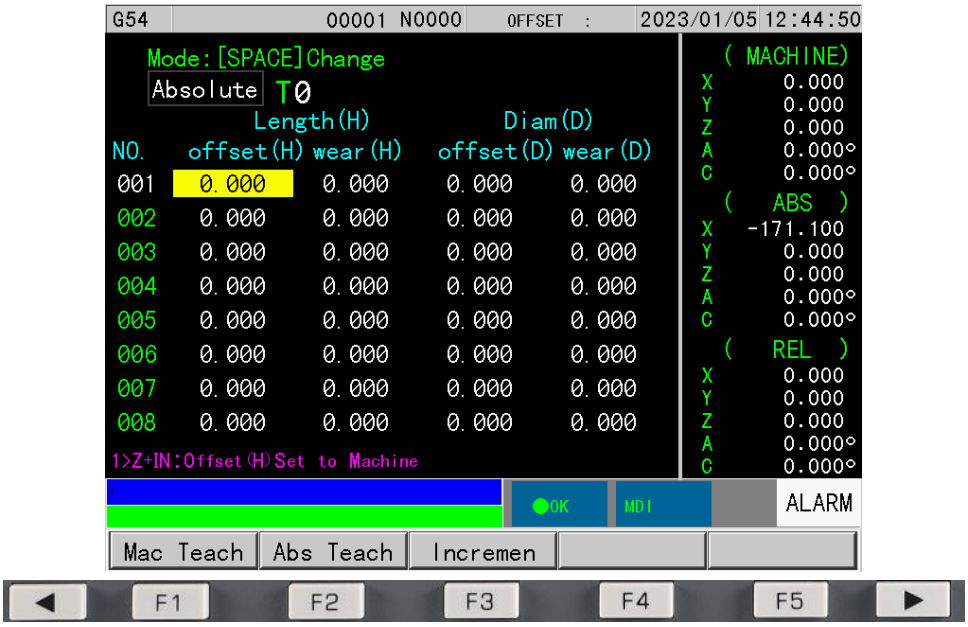
There are 6 main coordinate systems G54--- G5 9 and 48 sub-coordinate systems G54.01--- G54.48 in the coordinate system, F3 [one-key tool setting] and F4 [relative zeroing] can be divided into all axis settings and independent settings for each axis.

(Primary Coordinate System).

(sub-coordinate system)

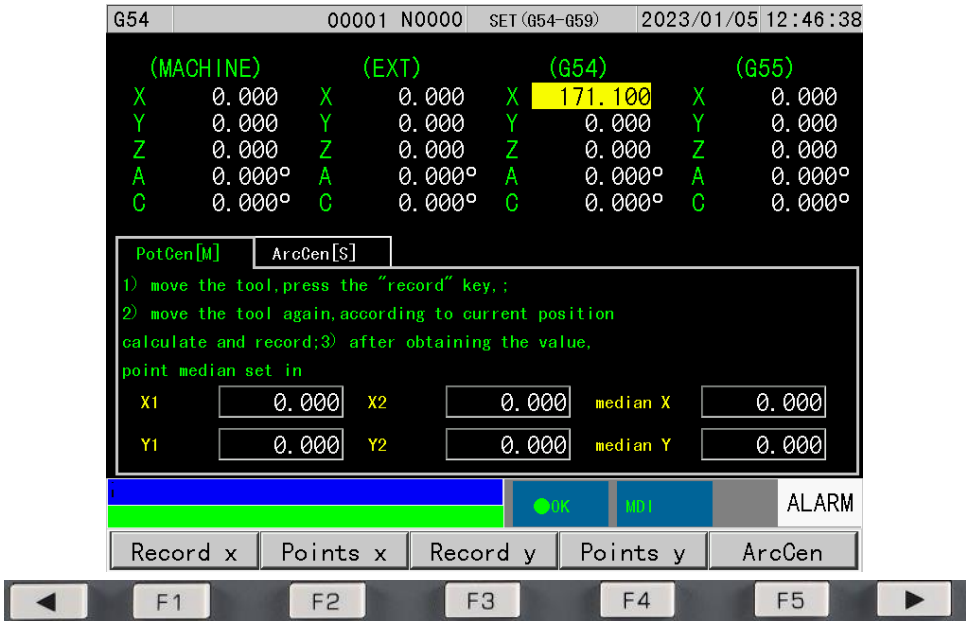
2) Tool offset wear interface

Tool offset wear interface has a total of 26 pages, a total of 256 offset and wear numbers (No.001 ~ No. 256) for user use, through keys PAGE▲, PAGE▼ keys to display each page, digital input through the F1 key [mechanical teaching], F2 key [absolute teaching], F3 key [incremental input], display the page as shown in the figure:



3) Split measurement interface

The minute measurement can be switched between the F5 key [Arc/Two Point Minute] or the letters [M] and [S] keys under the screen.

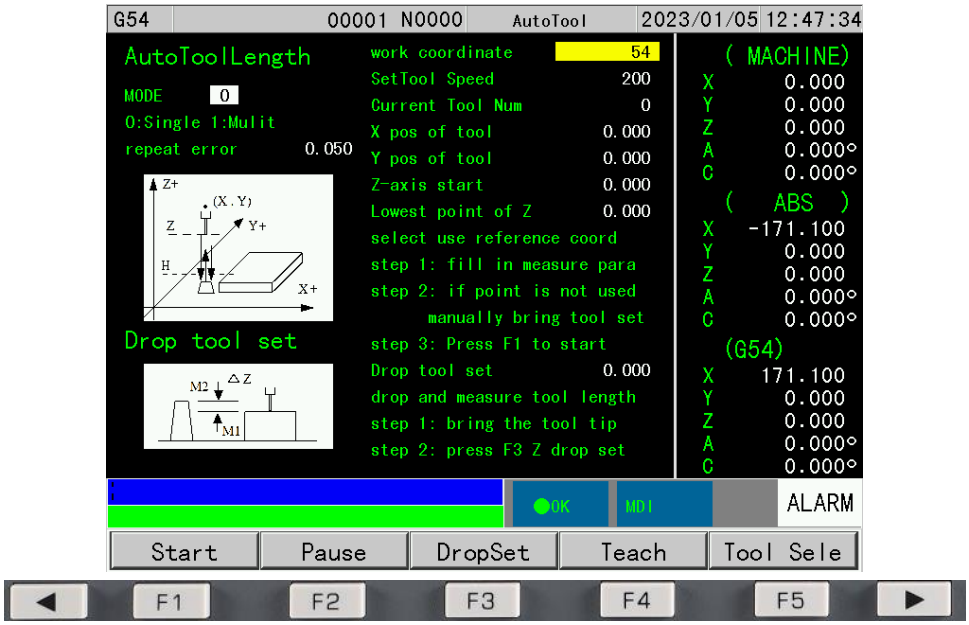


4) Automatic tool setting interface

Enter the automatic tool setting interface, and the optional parameter can realize automatic tool setting Z-axis tool setting.

Note: (1) The tool position of each axis is filled in with the machine coordinates of the current corresponding position;

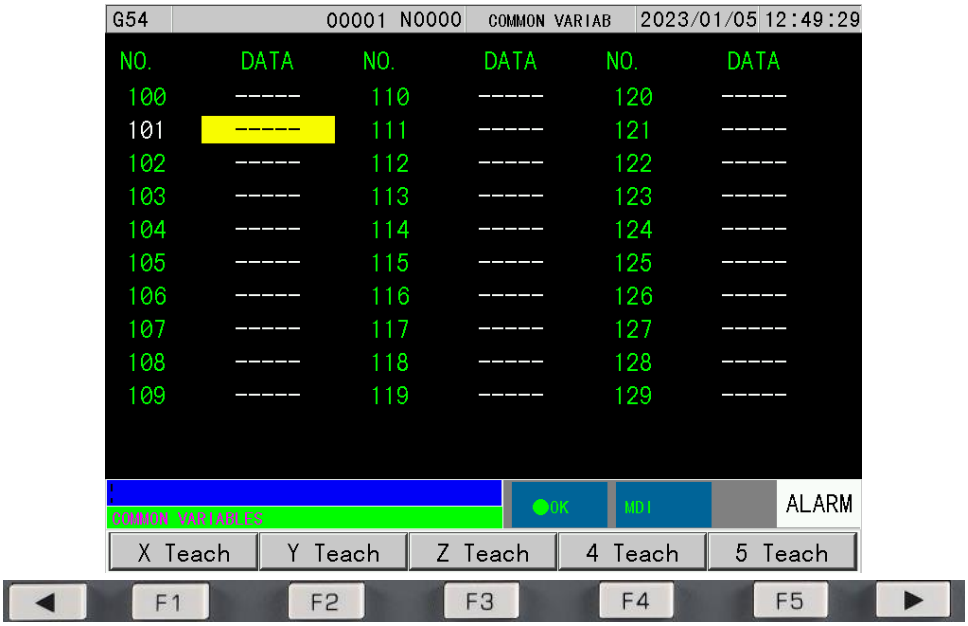
(2) To start automatic tool setting, it needs to be used in MDI mode;



5) Common variable interface

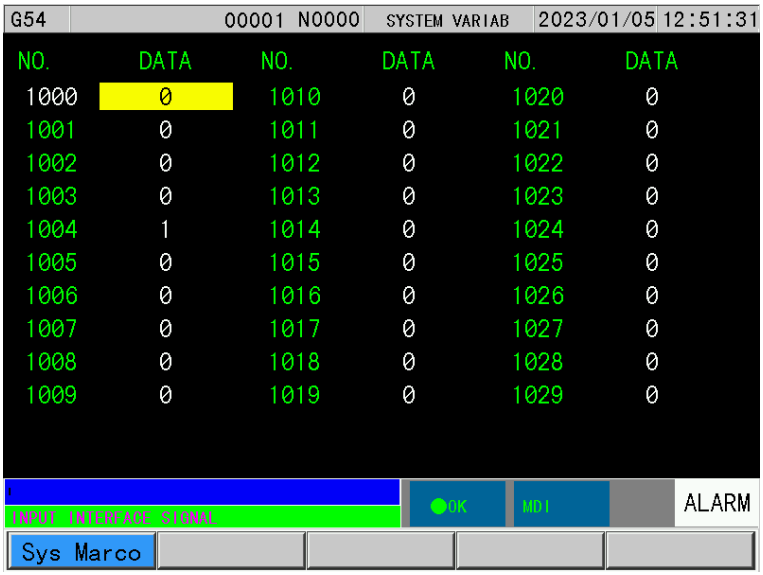
The macro variable interface has 20 pages, each page can be displayed by PAGE keys and

PAGE keys, and the macro variable page displays a total of 600 (No.100 ~ No.199 and No.500 ~ No.999) macro variables, macro variable values can be specified through macro code or directly set by the keyboard, and can be taught by the F1 key --- F5 key under the screen corresponding axis.



6) System variables interface

The macro variable interface has 49 pages, each page can be displayed by PAGE key and PAGE key, and the system macro variable page is displayed (No.1000 ~ No. 7250) System macro variables, some system macro variables can only be read and not writable, some can be read and written, and they cannot be manually entered.



1.3.4 Program monitoring interface

Press the MON key to enter the program monitoring interface, which includes program editing, graphic settings, MDI, information/settings, workpiece origin, tapping rewind, cumulative zeroing, zero number of pieces, and zero working hours.

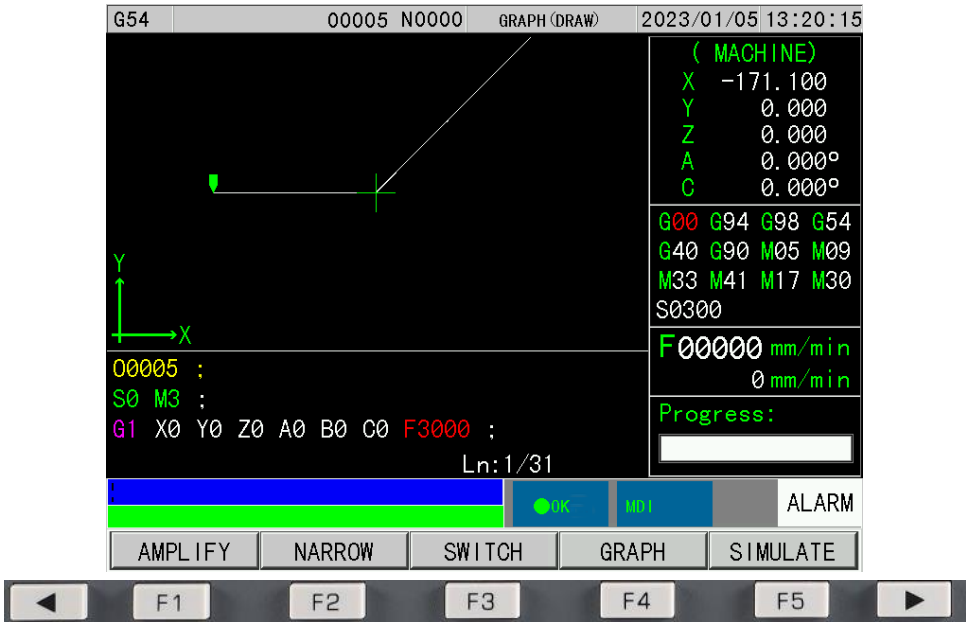
1) Program editing interface:

Press the MON key to enter the program monitoring interface, and then press the F1 key [Program Editing] under the screen to enter the program editing interface, and you can edit the program in a non-running state.

2) Graphical settings interface:

Press the MON key to enter the program supervision interface, and then press the F 2

key [Graphics Settings] under the screen to enter the graphics setting interface, the corresponding functions of the interface include zoom in, zoom out, switch, graphics generation, graphic simulation, graphic centering after page turning, and clearing.



3) MDI (Entry) Interface:

Press the 

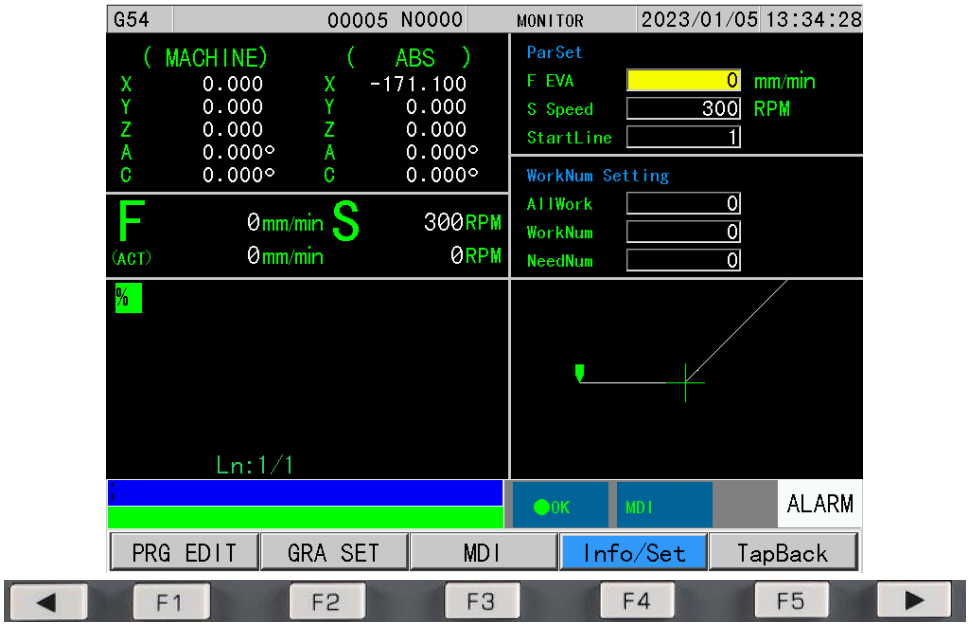
key on the panel to enter the input mode, then press the 

key to enter the program supervision interface, and then press the F 3 key [MDI] under the screen to enter the input editing interface.

4) Information/Settings Interface:

Press the 

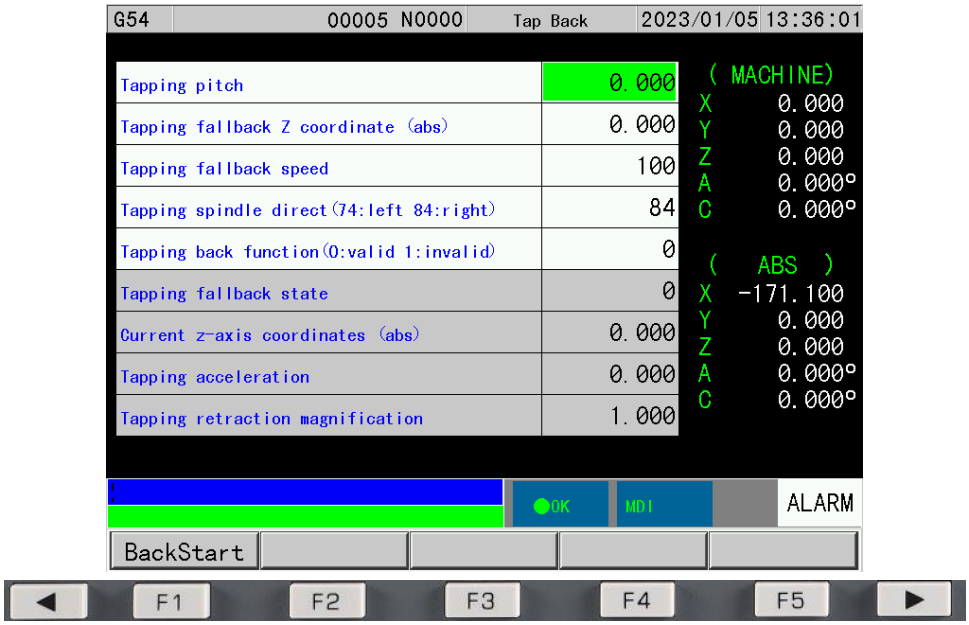
key to enter the program monitoring interface, and then press the F4 key [Information/Settings] under the screen to switch the screen processing modal display and the number of workpieces, feed rate, spindle speed setting interface.



5) Tapping fallback interface:

Press the 

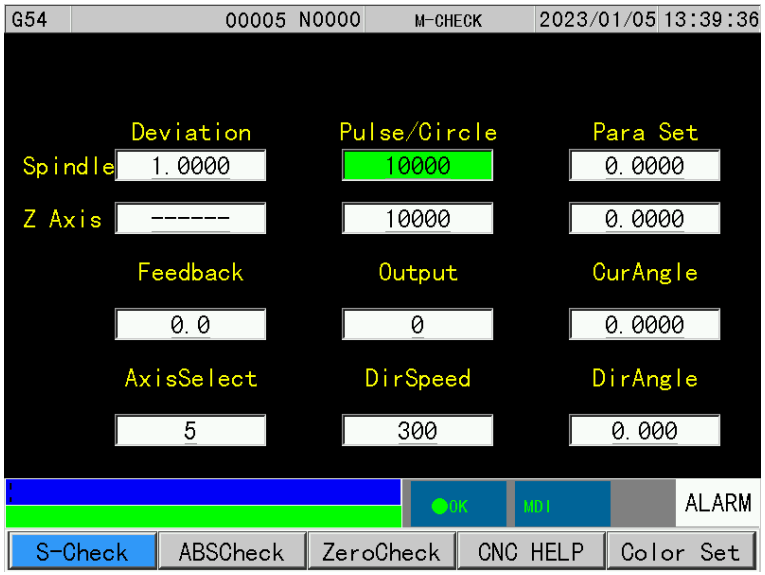
key to enter the program supervision interface, then press the right button 
on the screen to turn the page, and then press the F1 key [tapping back] under the screen to enter the tapping backoff setting interface, and set the relevant parameters of the rollback according to the current spindle tap pitch and position.

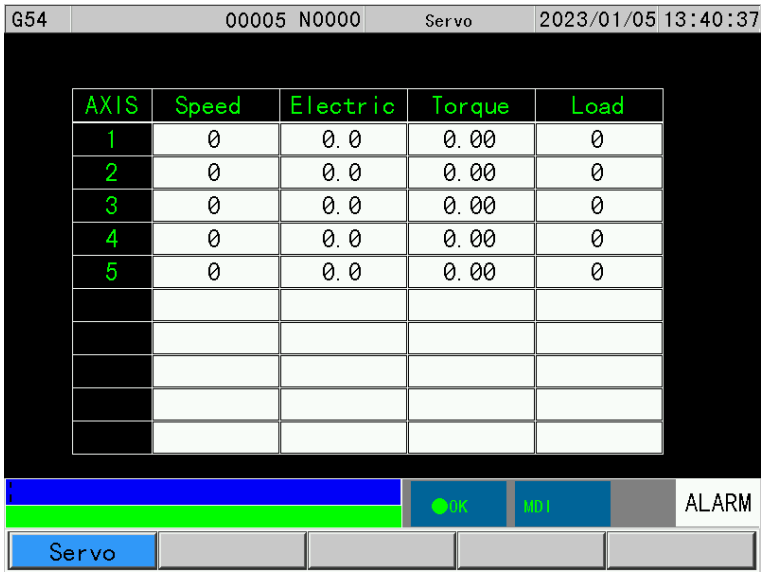


1.3.5 Diagnosis and alarm interface

Press the 

key to enter the diagnostic alarm interface, the diagnostic alarm interface corresponds to CNC diagnosis, PLC signal, IO diagnosis, version information, alarm, spindle detection after page turning, absolute value, zero point detection, CNC help, color setting, servo monitoring after page turning and other diagnostic alarm monitoring.





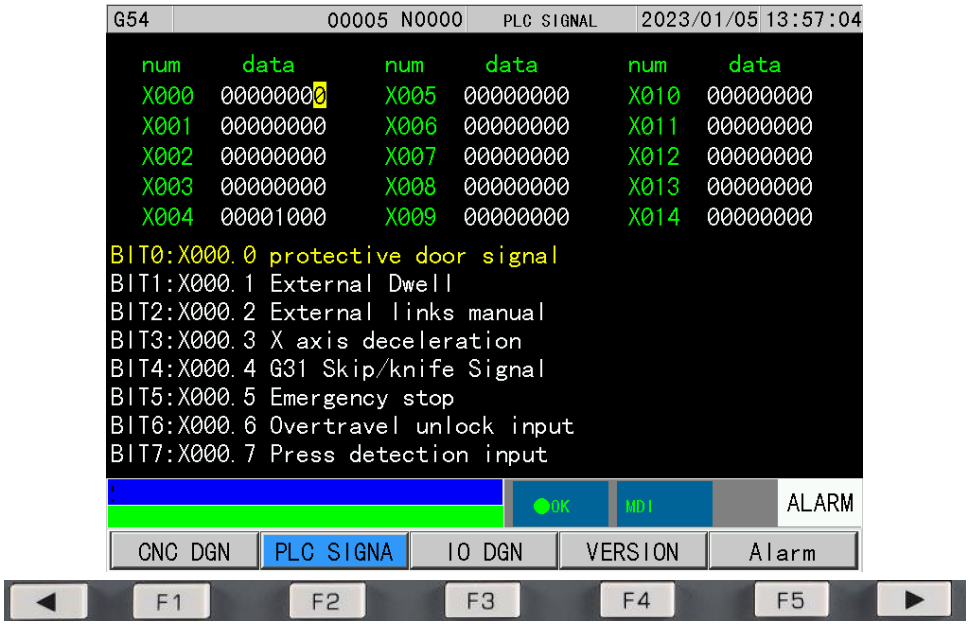
1) CNC diagnostic interface:

Press the 

key to enter the diagnostic alarm interface, and then press the F1 key [CNC Diagnosis] interface under the screen.

2) PLC signal interface:

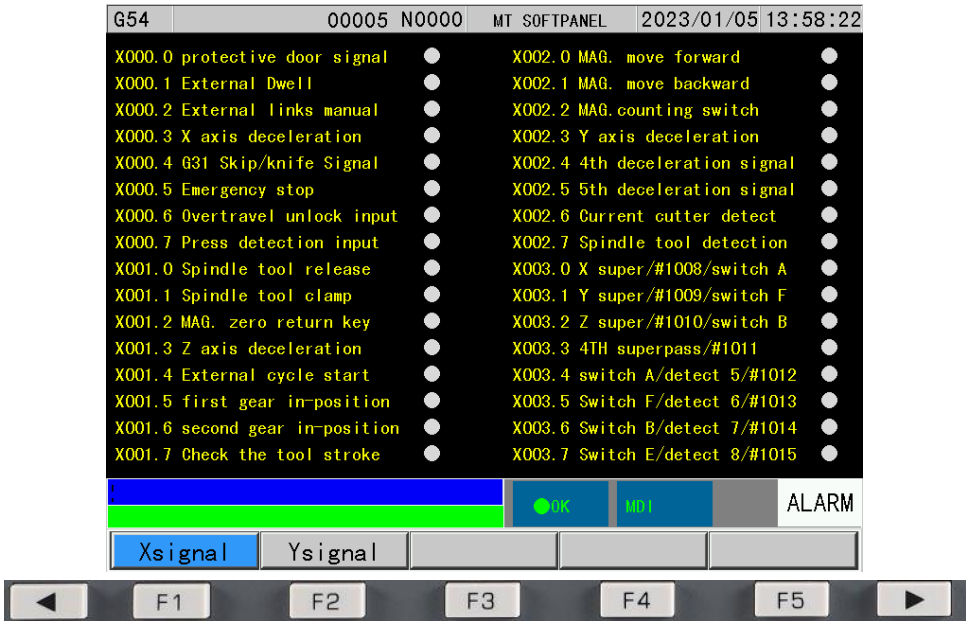
Press the DIAGNOSIS key to enter the diagnostic alarm interface, and then press the F2 key [PLC signal] interface under the screen, which diagnoses and displays the PLC signal and the system conversion signal change status.



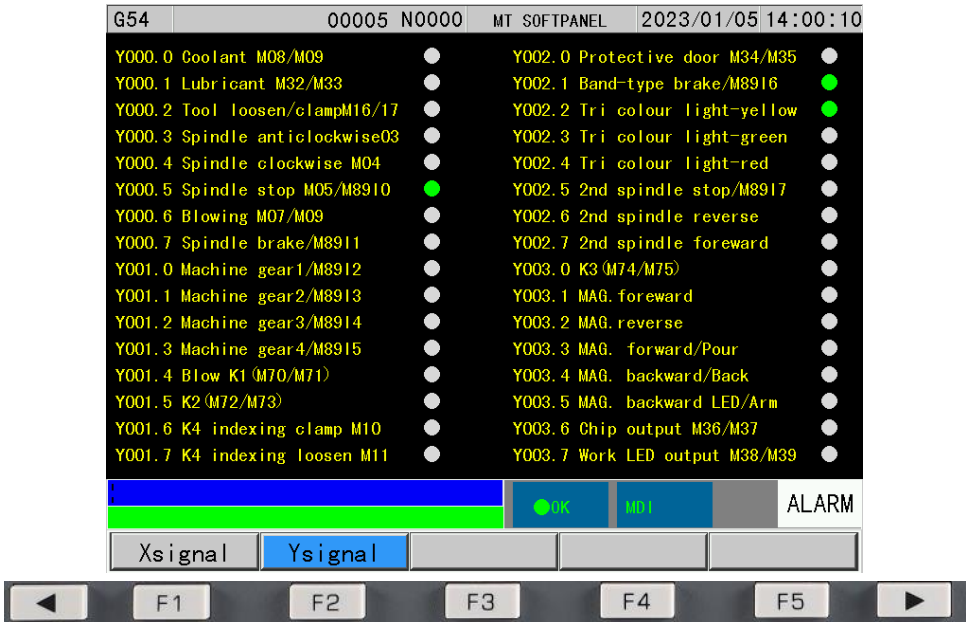
3) IO diagnostic interface:

Press the DIAGNOSIS key to enter the diagnostic alarm interface, and then press the F3

key [IO Diagnosis] interface under the screen, which displays the PLC internal input and output signals.



(input signal).

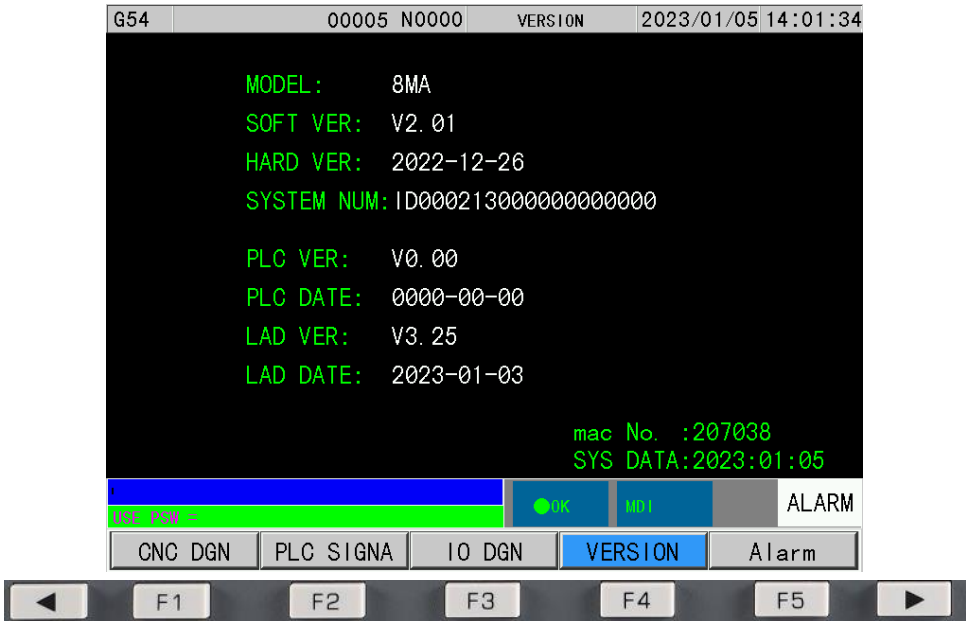


(output signal).

4) Version information interface:

Press the 

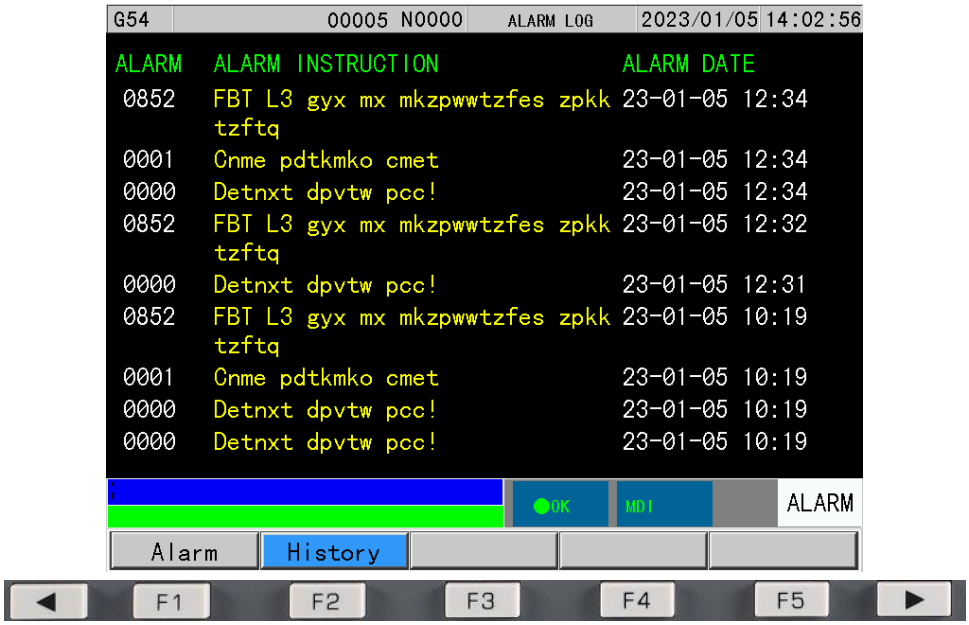
key to enter the diagnostic alarm interface, and then press the F4 key [Version Information] interface under the screen, which displays the basic information of the system and the machine number and installment password input.



5) Alarm interface:

Press the 

key to enter the diagnostic alarm interface, and then press the F5 key [Alarm] interface under the screen, the alarm interface contains the current alarm display and historical alarm log content.



Note: (1) Alarm clearance: When the alarm has been canceled, press the 

key to clear the alarm

content;

(2) Enter the alarm log interface. A total of 200 alarm log information can be viewed through

keys and

keys, and the order is arranged: the latest **alarm log information is listed at the top of the first page, and then pushed in order. When the alarm log exceeds 200, the last historical log information is cleared.**

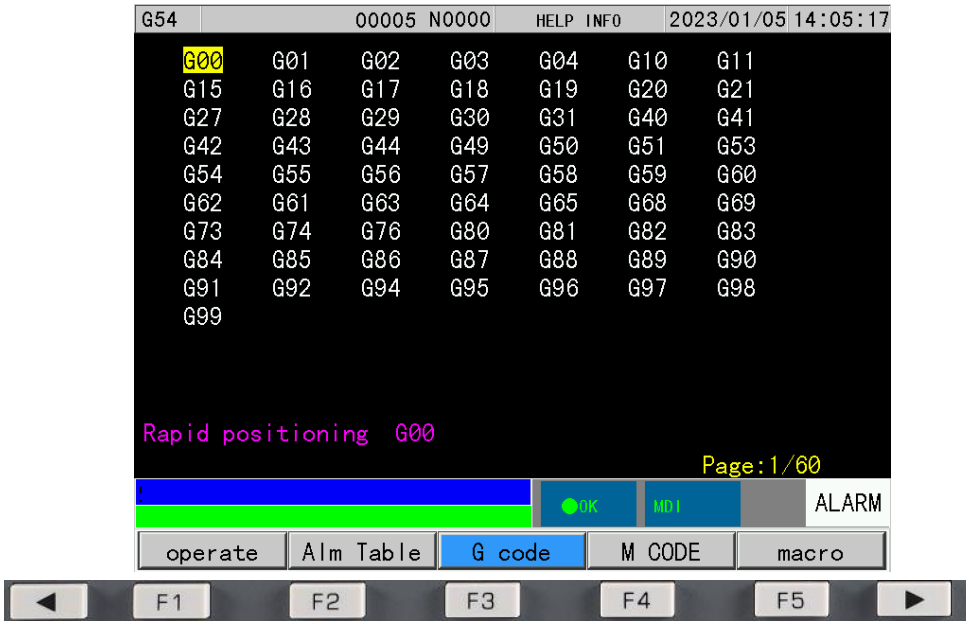
Note: Manual clearing of alarm logs: Press the function key [Clear History] under the **level 2 password to clear all log information.**

6) CNC Help Interface:

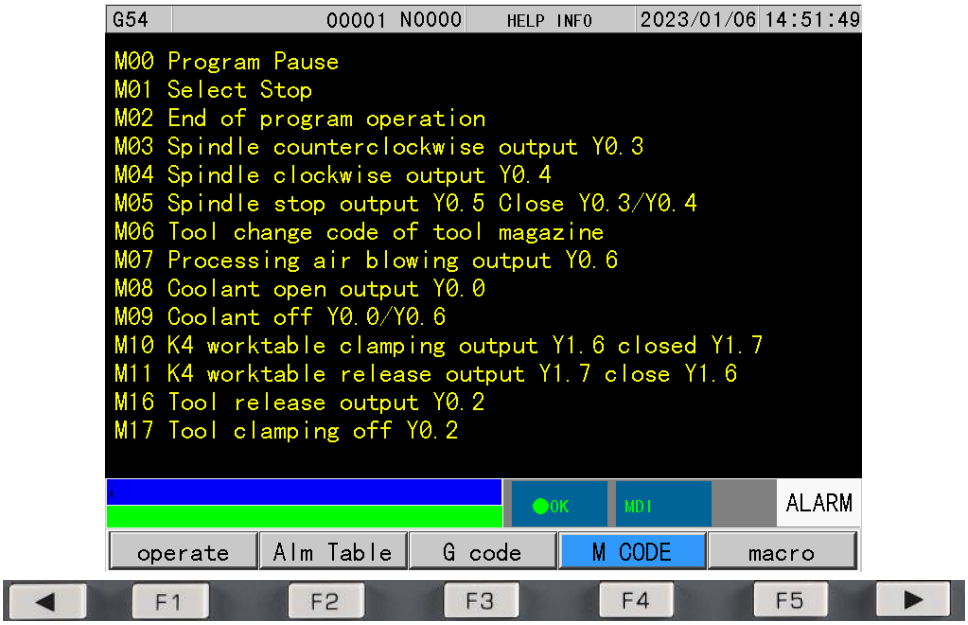
Press the

key to enter the diagnostic alarm interface, then press the right button

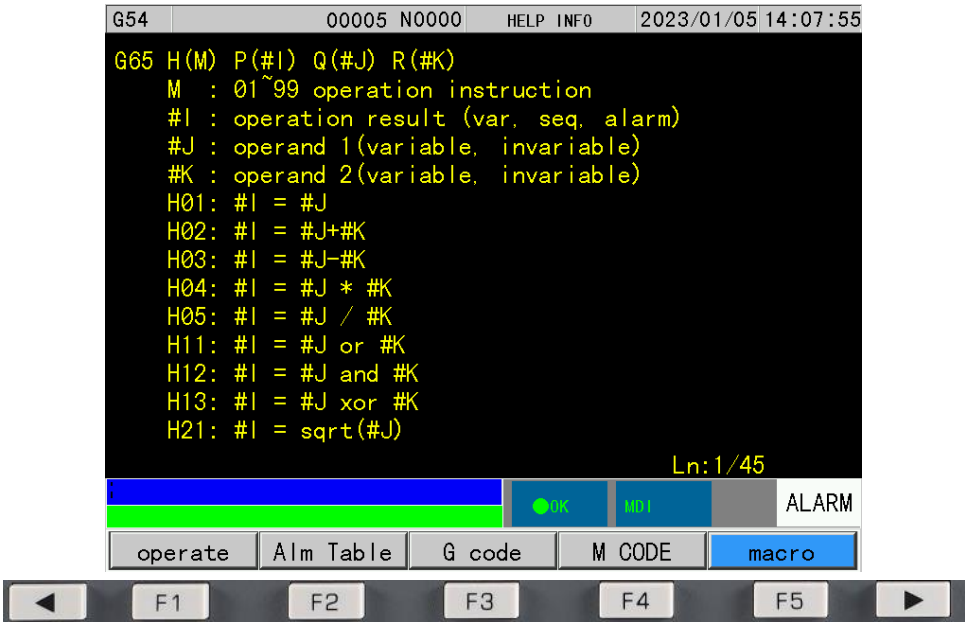
on the screen to turn the page, and then press the F4 key [CNC Help] under the screen to enter the CNC help interface, which has operation table, alarm table, G code table, M code table, macro instructions.



(G-Size Table)



(M Size Table)



(macros).

7） Settings interface:

Press the  screen or  key to switch functions, find the F1 key [Settings] function under the screen, click the F1 key under the screen to enter the setting function interface, and the corresponding password, backup, time, and switch interface under the setting interface.

Password interface:

Permission settings: Display and set the user operation level.

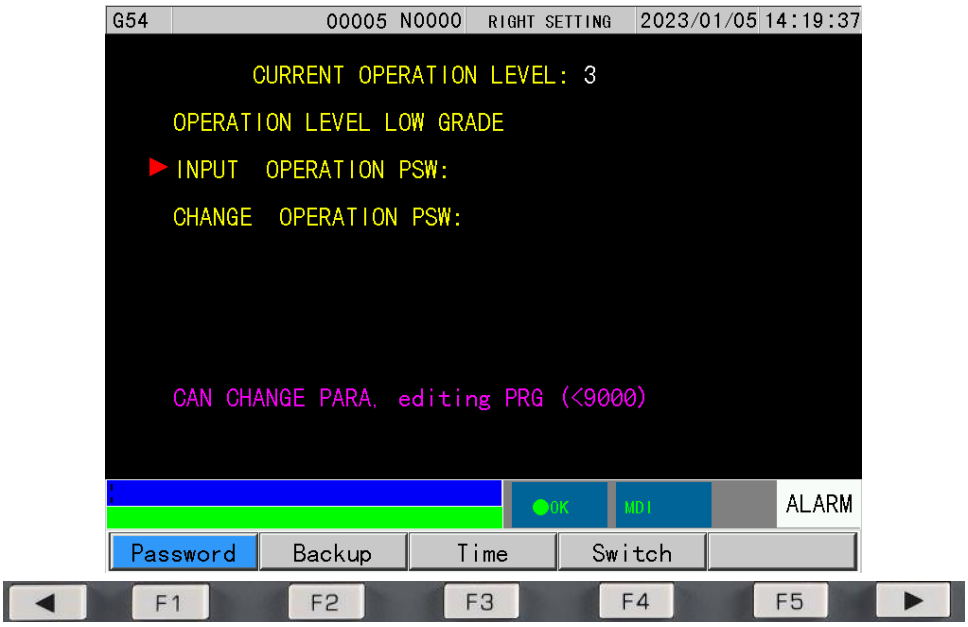
The system password level is divided into 4 levels, from high to low, they are machine tool manufacturer level (level 2), equipment management level (level 3), craftsman level (level 4), processing operation level (level 5).

Machine tool manufacturer level: allow to modify CNC status parameters, data parameters, screw parameters, tool compensation data, edit part programs (including macro programs), edit and modify PLC ladder diagram, download and upload ladder chart;

Device management level: initial password 12345, allowing to modify CNC status parameters, data parameters, knife supplement data, edit programs;

Technician level: initial password 1234, can modify the tool supplement data (for tool setting operations), macro variables, edit the part program, and cannot modify the status parameters, data parameters and screw parameters of the CNC.

Machining operation level: no password level, can be operated by the machine tool operation panel, can not modify the tool supplement data, can not select the part program, cannot edit the program, can not modify the status parameters, data parameters and screw parameters of the CNC.



Note: (1) The system level 1 password **is the most important password, and the original level 1 password of the system cannot be changed back to the factory password after being modified;**

(2) **The system cannot edit program or debugging parameters under the 5-level password;**

Backup interface:

This interface can perform backup and restore system parameters, and the interface can backup and restore the current ladder diagram and all ladder diagrams of the system under at least 2 levels of password.

Data manipulation: From this page, you can back up and restore CNC data (status parameters,

data parameters, screw parameters, tool offsets, etc.).

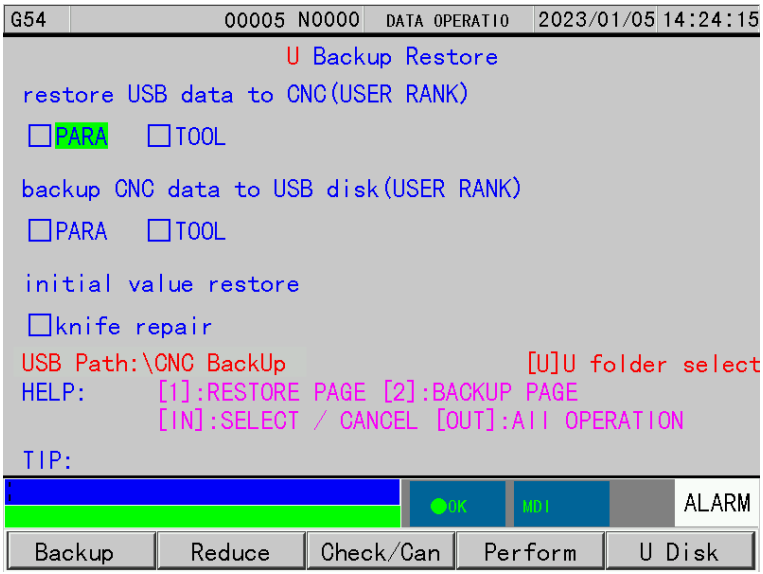
Initial value restore operation: You can restore parameters, knife compensation, and screw compensation to system defaults.

C drive data restore to CNC: restore the data files backed up on the system disk to the system.

Restore U disk data to CNC: Return the data files backed up on the U disk to the system.

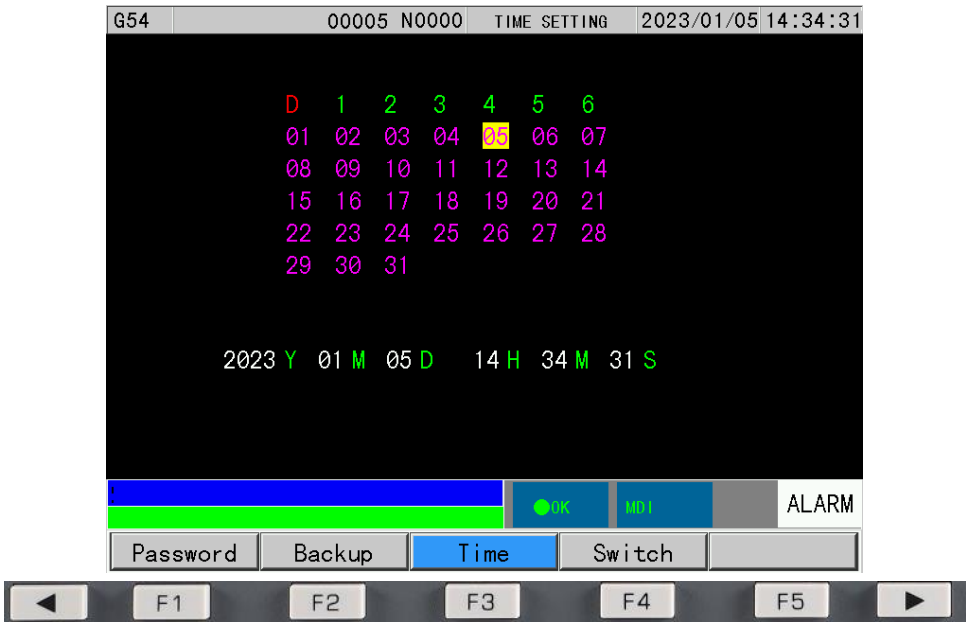
CNC data backup to C drive: backup the current parameters, knife repair, screw repair and ladder diagram of the system to the system disk.

CNC data backup to USB disk: backup the current parameters of the system, knife repair, screw repair and ladder diagram to the USB flash drive.



Time interface:

The current time and date are displayed under the interface, and the date and time of the interface can be modified and entered by pressing the panel SPACE key to switch and arrow key movement under the 1-level operation password.



Switch interface:

This interface can set parameter switches, program switches, automatic serial numbers and other settings.

Parameter switch: When the parameter switch is turned on, the parameters can be modified; When closed, parameter modification is prohibited.

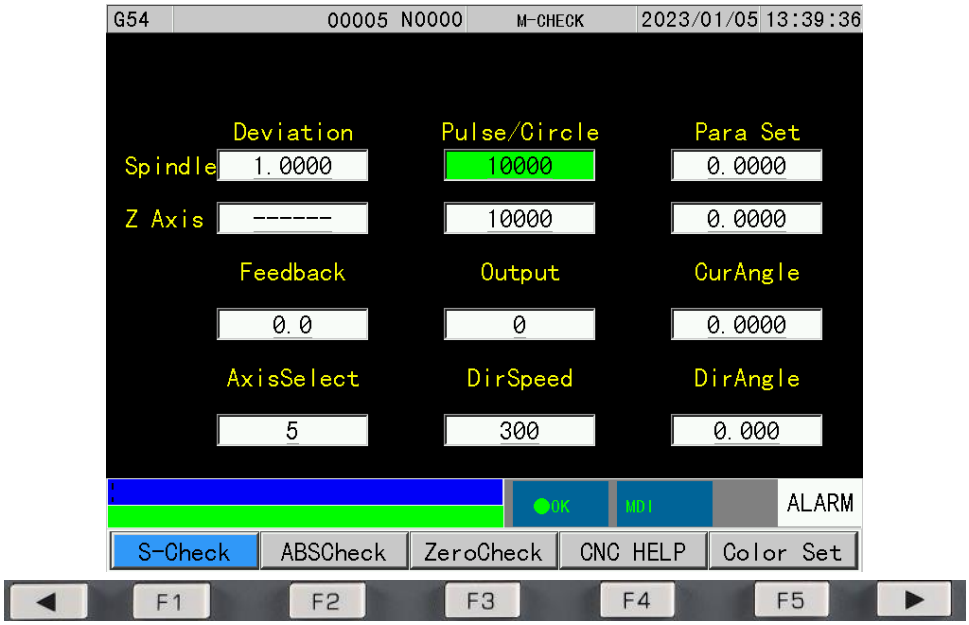
Program switch: When the program switch is turned on, you can edit the program; When closed, prohibits editing of the program.

Automatic serial number: when the automatic serial number switch is turned on, the program segment number is automatically generated when editing the program; When the automatic serial number switch is off, the program segment number is not automatically generated and must be entered manually if necessary.

Spindle detection interface:

Press the 

key to enter the diagnostic interface, press the screen 
down button and turn the page to find the F1 key [spindle detection], click to enter the spindle detection interface, in the spindle detection interface can display the spindle error, Z-axis error for debugging rigid tapping use, the display page is shown in the figure.



Absolute value interface:

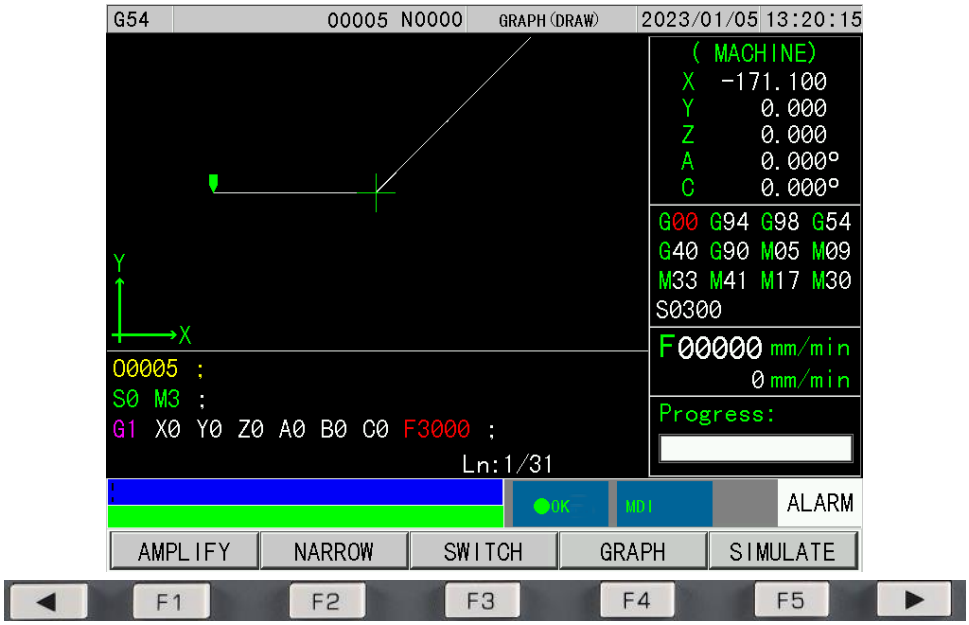
Press the 

key to enter the diagnostic interface, press the screen 
down button and turn the page to find the F2 key [absolute value], click into the absolute value interface, in the absolute value interface can display the machine coordinates of each axis, motor feedback coordinates, zero point position, etc For debugging absolute motors, the display page is shown in the figure.

1.3.6 Graphical Interface

Press the 

panel to enter the program monitoring interface, and then press the F2 key [Graphics Settings] under the screen to enter the graphical interface, you can enlarge, retract, clear and other operations of the graphics.



1.3.7 Status parameters, data parameters, screw complement parameter boundaries

Press the  screen or  key to switch the function, find the F2 key [parameter setting] function under the screen to click to enter the parameter interface, which has a parameter table, processing technology, user parameters, bit parameters, number parameters and screw supplementation Parameters and several other functional boundaries.

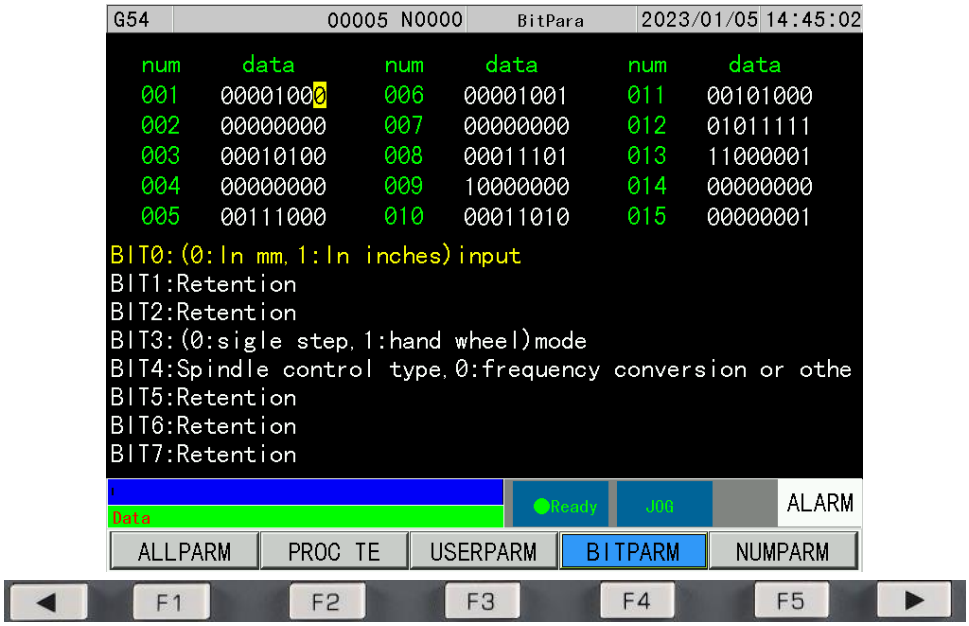
1) Status parameter interface

After entering the parameter interface, press the F4 key [bit parameter] under the screen to enter the status parameter page, the status parameters are displayed in four pages with a total of 60 components, and the relevant parameters can be viewed or modified by 

key and

key to enter each page, as follows:

As can be seen from the status parameter page, there are two lines of parameter content display lines in the lower part of the page, and the first line shows the English abbreviation of all bits of the parameter where the current cursor is located; The second line shows the Chinese meaning of a bit of the parameter where the current cursor is located, and you can press the panel  key or  keys to change the displayed parameter bit.



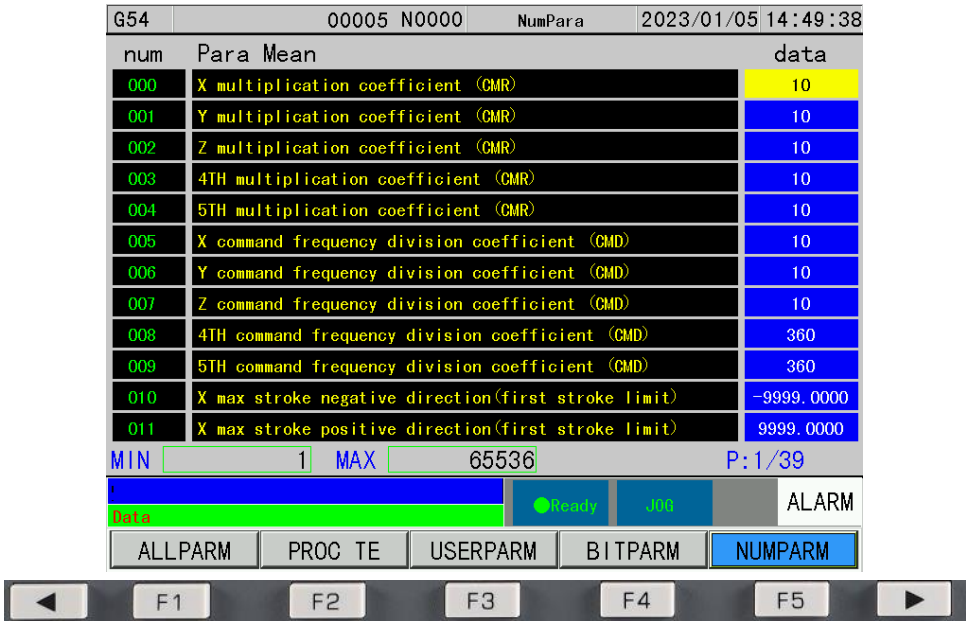
2) Data parameter interface

After entering the parameter interface, press the F5 key [Number Parameters] under the screen to enter the data parameter page, and you can enter each page through keys 

and 

keys to view or modify the relevant parameters, as follows:

As you can see from the data parameters page, there is a line of Chinese prompt lines on the left part of the page, showing the meaning of the parameter pointed to by the current cursor, and the lower part of the page showing the value range of the parameter setting.



Note: Under the first-level operation level, use panel keys 

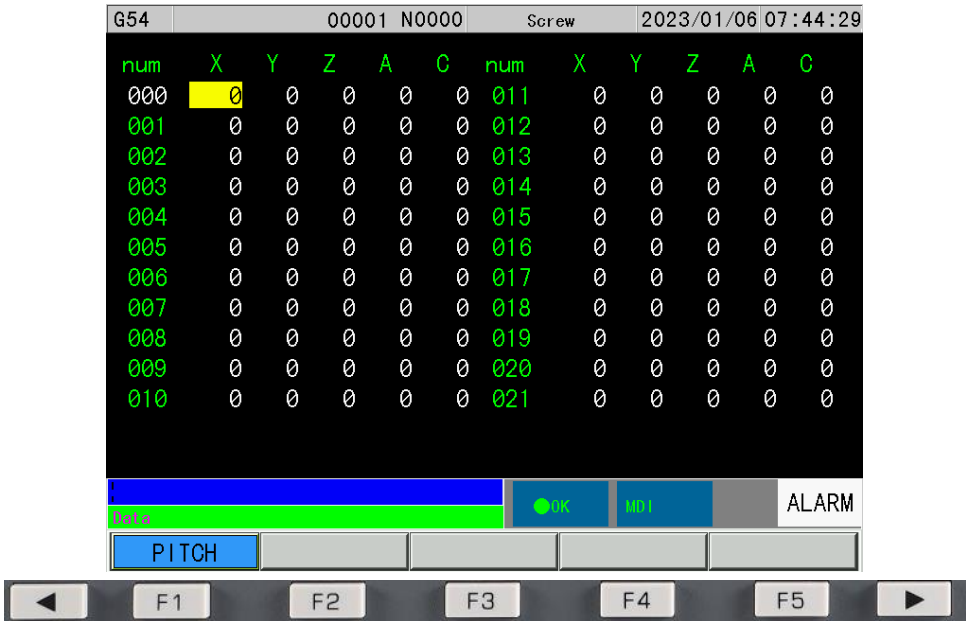
,  **keys,** and  **keys** to **switch to modify the maximum and minimum values of the parameter value range.**

3) Screw complement parameter interface

After entering the parameter interface, press the screen down  button to turn the page, find the F 1 key [screw supplement] to enter the pitch error compensation interface, the pitch error compensation is displayed in a total of 256 pages in 11 pages, and each page can be displayed by the 

key and 

key:



1.3.8 CNC diagnostics, PLC signals, machine soft panels, help information, version information interface

(See Section 1.3.5).

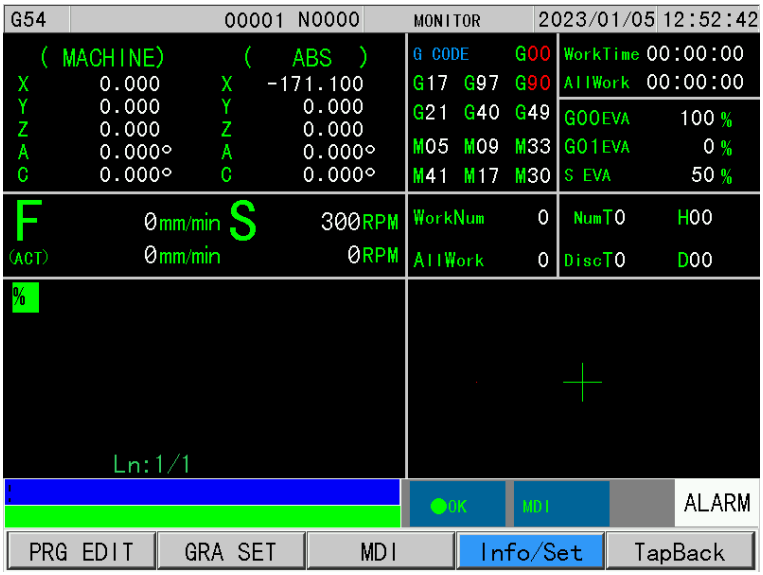
Chapter II: Power On, Shutdown and Safety Protection

2.1 Boot

Before the system is powered on, it should be confirmed:

1. The machine is in good condition.
2. The supply voltage meets the requirements.
3. Correct and secure wiring.

System self-test, initialization. After the self-test and initialization are completed, the current location (program supervision) page is displayed.



2.2 Shutdown

Before shutting down the system, you should confirm:

1. The feed shaft of the CNC is in a stopped state;
2. Auxiliary functions (such as spindle, water pump, etc.) are turned off;
3. First cut off the CNC power supply, and then cut off the machine tool power supply.

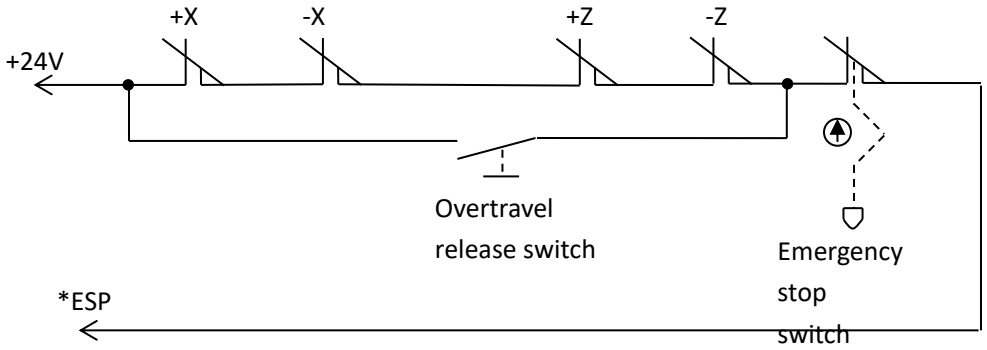
Note: For the operation of cutting off the power supply of the machine tool, please refer to the manual of the machine tool manufacturer.

2.3 Overtravel Protection

In order to avoid damage to the machine due to X-, Y-, and Z-axis overtravel, overtravel protection measures must be taken.

2.3.1 Hardware Overtravel Protection

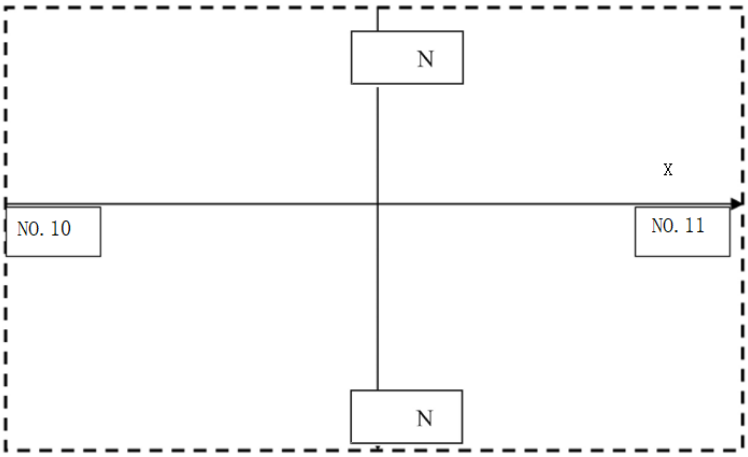
Install the stroke limit switches at the positive and negative maximum strokes of the X, Y, and Z axes of the machine tool, and wire them as shown below, at this time the Bit2 (EALM) of the state parameter No.60 and Bit3 (LALM) must be set to 0. When overtraveling, the stroke limit switch acts, the system stops moving and displays an emergency stop alarm.



When hardware overtravel occurs, the system will have an "emergency stop" alarm. The method to eliminate the "emergency stop" alarm is: press the overtravel release button without release, switch to the alarm information page, view the alarm information, reset and clear the alarm, and move the workbench in the opposite direction (if the positive overtravel, the negative direction is moved out; If the negative overtravel, the positive move out) is disengaged from the stroke switch.

2.3.2 Software Overtravel Protection

The software travel range is set by the data parameter NO.00 0 ~ NO.009, and the machine coordinate value is used as the reference value. The status parameter No.022 Bit1(LZR) can set whether the software limit function is active before the mechanical zero return.



If the machine position (machine coordinates) exceeds the dotted area in the figure above, an overtravel alarm appears. The method to cancel the overtravel alarm is: press the reset button, clear the alarm display, and move in the opposite direction (if the positive overtravel, the negative direction moves out; If the negative overtravel, the positive move out) is sufficient. 2.4 Emergency Operations

During machining, unexpected results may occur due to user programming, operation and product failure, and the system must be stopped immediately. This section describes the treatment that the system can carry out in an emergency, and the handling of CNC machines in an emergency can be found in the relevant instructions of the machine tool manufacturer.

2.4 Emergency Operations

During machining, unexpected results may occur due to user programming, operation and product failure, and the system must be stopped immediately. This section describes the treatment that the system can carry out in an emergency, and the handling of CNC machines in an emergency can be found in the relevant instructions of the machine tool manufacturer.

2.4.1 Reset

When the system is abnormal output and the coordinate axis is abnormal operation, press the 

key to make the system in a reset state:

1. All axis movements stop;
2. M, S function output is invalid (can be defined by whether the spindle turns off counterclockwise / clockwise turn, lubrication, cooling and other signals after the parameter setting button 

, PLC ladder diagram);
3. The automatic operation ends, and the modal function and state are maintained.

2.4.2 Emergency Stop

When the emergency stop button is pressed in danger or emergency during the operation of the machine tool (when the external emergency stop signal is valid), the CNC enters the emergency stop state, at which time the machine tool movement stops immediately, and the spindle rotation, coolant and other outputs are all turned off. Release the emergency stop button to release the emergency stop alarm, and the CNC enters the reset state. The circuit connection method is shown in section 2.3.1 of this chapter.

Note 1: Confirm that the fault has been eliminated before dismissing the emergency stop alarm;

Note 2: Pressing the emergency stop button before power-up and shutdown reduces the electrical shock of the device;

Note 3: After the emergency stop alarm is lifted, the operation back to the zero point of the machine tool should be performed again to ensure the correctness of the coordinate position (if the machine tool is not installed with the zero point of the machine tool, the zero point operation of the machine tool shall not be carried out);

NOTE 4: Only the status parameter NO Bit2 (EALM) of 60 is set to 0 for the external emergency stop to be valid.

2.4.3 Feed hold

The operation can be paused by  pressing a button during machine operation. It is important to note that this function does not stop the running action immediately when the thread is cut and the cyclic code is running.

2.4.4 Cut off the power supply

During the operation of the machine tool, the power supply of the machine can be cut off immediately in case of danger or emergency to prevent accidents. However, it must be noted that after cutting off the power, the CNC display coordinates may have a large deviation from the actual position, and operations such as re-aligning the tool must be carried out.

Chapter III: Manual Operation

The function of the buttons in the system machine panel is defined by the PLC program (ladder diagram), and the functional significance of each button can be found in the machine manufacturer's manual.

Please note that the following functions related to the operator panel buttons in this chapter are described for the standard PLC program of the system!

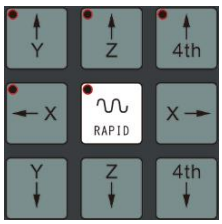
Press the 

key to enter the manual operation mode, under the manual operation mode, manual feeding, spindle control, rate adjustment, tool change and other operations can be carried out.

3.1 Axis movement

In the manual operation mode, the two-axis manual feed and manual fast movement can be made.

3.1.1 Manual feeding

Press 

and hold the feed axis and the direction selection key 

or 

the X axis arrow key to make the X axis negative or positive feed, and the axis movement stops when the button is released; Press 

and 

hold the Z arrow key to make the Z axis negative or positive feed, and the axis movement stops when the button is released; Press 

and 

hold the Y axis arrow key to make the Y axis negative or positive feed, and the axis movement stops when the button is released; Press 

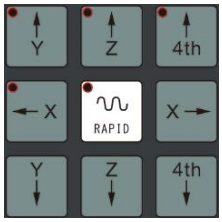
and 

hold or 4th axis arrow key to make the 4th axis negative or positive feed, and the axis movement stops when the key is released .

When manual feeding is performed, press the 

key to make the indicator light in the status indicator area light up to enter the manual fast move state.

3.1.2 Manual fast movement

Press 

the key until the 

fast movement indicator in the status indicator area is on, press the 

or 

key to make the X axis negative or positive rapid movement, release the button when the axis movement stops; Press the 

or 

key to move quickly in the negative or positive direction of the Z axis, and stop the axis movement when the button is released; Press 

and 

hold the Y axis arrow key to make the Y axis negative or positive feed, and the axis movement stops when the button is released; Press 

and 

hold or 4th axis arrow key to make the 4th axis negative or positive feed, and the axis movement stops when the key is released . Fast magnification real-time trimming is effective.

When performing manual fast movement, press the 

key so that the indicator light goes off, fast movement is invalid, and feed at manual speed.

Note 1: After the power is turned on, if the reference point is not returned, when the fast move switch is on (fast move indicator is on), whether the fast moving speed is manual feed speed or fast moving speed is determined by the Bit0 bit (ISOT) of the system status parameter NO.012) select;

Note 2: In Edit / Handwheel mode, the 

key is invalid.

3.1.3 Speed Trimming

When manual feeding, the manual feed magnification can be modified using the feed rate rotary switch on the panel, a total of 16 steps. When the parameter NO.110 is set to 1260, the relationship between the feed rate and the feed rate is shown in the following table

Feed rate (%)	Feed speed (mm/min).
0	0
10	126
20	252
30	378
40	504
50	630
60	756
70	882
80	1008
90	1134
100	1260
110	1386
120	1512
130	1638
140	1764
150	1890

Note: This table has an error of approximately 2%.

When moving quickly manually, you can press or modify the magnification of manual fast movement, and the fast magnification has Fo, 25%, 50%, and 100% four gears. (Fo speed is set by data parameter No.085).

3.2 Other manual operations

3.2.1 Turn counterclockwise, turn clockwise, stop control

: In manual operation mode, press this key, the spindle turns counterclockwise;

: In manual operation mode, press this key to stop the spindle;

: In manual operation mode, press this button to rotate the spindle clockwise.

3.2.2 Spindle jogging

: The spindle is now jogged.

Function description: Press the 

key on the panel to enter the jogging state, and the spindle jogging function needs to be in the stop state for opening and closing the spindle jogging function.

Spindle jogging state, press the 

key, counterclockwise to jog; Press the 

key to turn the jog clockwise. The jogging speed is set by data parameter No.208.

When the spindle jog rotates, press the 

key to stop the spindle jogging rotation, and the spindle braking signal will not be output when the jogging rotation stops.

K10.4 is set to 1, and spindle jogging is valid in any way. In automatic or input mode, the spindle is in the jog rotation state, at which time the running program will turn off the jog rotation of the spindle and turn off the jogging function.

Parameter settings:

PLC parameter K10.4 1/0: Spindle jogging is valid in any mode / manual, handwheel, zero return mode.

Data parameter No.208: rotation speed when the spindle jogging

3.2.3 Coolant Control

: In any mode of operation, press this button to switch the coolant between switches.

Parameter setting: PLC parameter K10.1 1/0: Spindle lubrication and cooling output hold/off during reset

3.2.4 Lubrication control

Function description:

1) Non-automatic lubrication:

DT13 = 0: Non-automatic lubrication.

When DT13=0, in order to lubricate the flip output, press the machine tool operation panel 

key to lubricate the output, and press repeatedly to cancel the lubrication output.

When executing M32, lubricate the output, then execute M33, lubricate the output cancel.

When DT13>1, for the lubrication timing output, press the machine tool operation panel 

button, lubricate the output, after the time set by the data parameter 112, the lubrication output is canceled; M32, lubrication output, after the time set by data parameter DT13, lubrication output canceled. If the time set by DT13 is not reached, M33 is executed at this time, and the lubrication output is canceled.

2) Automatic lubrication:

DT13>0: Automatic lubrication with configuration time DT13 and lubrication interval DT53

After the system is powered on, start lubricating the time set by DT13, then stop the output, and after the time set by DT53, repeat the output lubrication, and cycle in turn. M32, M33 codes, machine operator panel keys are also valid for automatic lubrication.

Parameter settings:

PLC parameters: K10.1 1/0: spindle lubrication cooling output hold/off during reset

PLC parameters: K16.2 1/0: When automatic lubrication is effective, turn on yes / no output lubrication

PLC data: DT53: Automatic lubrication interval time (ms).

PLC data: DT13: Automatic lubrication output time (ms).

DT05: M code execution duration (ms).

DT13: lubrication opening time (0-60000ms) (0: lubrication unlimited time).

3.2.5 Tool change

: In manual operation mode, press this button to loosen / clamp the spindle tool.

3.2.6 Tuning of spindle magnification

In manual operation mode, the spindle speed can be adjusted when the analog voltage output is selected to control the spindle speed.

Press the spindle magnification rotation rate switch on the system panel to adjust the spindle magnification to change the spindle speed, which can realize a total of 8 levels of real-time adjustment of spindle magnification 50%~120%.

Chapter 4 Handwheel/Single Step Operation

In handwheel/ step operation, the machine moves according to the selected increment value.

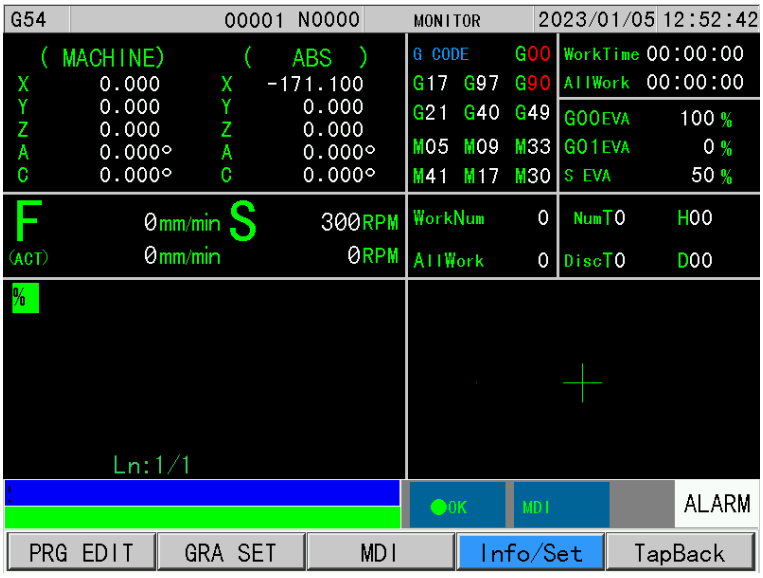
The function of the buttons in the system machine panel is defined by the PLC program (ladder diagram), and the functional significance of each button can be found in the machine manufacturer's manual.

Please note that the following functions related to the operator panel buttons in this chapter are described for the standard PLC program of the system!

4.1 Single stepper feed

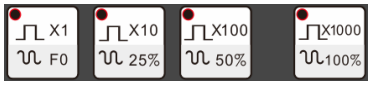
Set the Bit3 bit of system parameter No.001 to 0, press the 

key to enter the single-step operation mode, and the page is displayed as follows:



4.1.1 Selection of increments

Press the 

key to select Move Increment, and the Move Increment is displayed on the page. When the BIT7 of PLC status parameter K016 is 0, it 

is valid.

4.1.2 Movement direction selection

Press the 

or 

key once to make the X axis negative or positive feed in single-step increments; Press the 

or 

key once to make the Z axis feed negatively or positively in single-step increments. Press the 

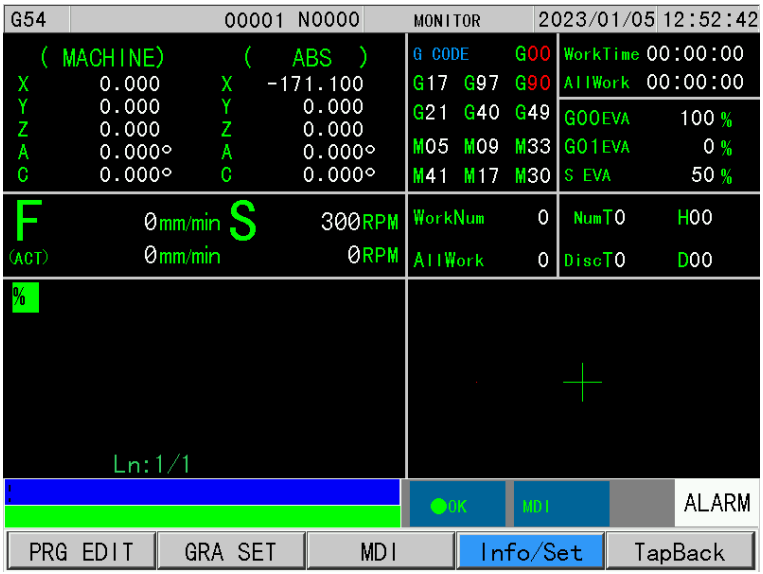
or 

key once to make the Y axis negative or positive feed in single-step increments.

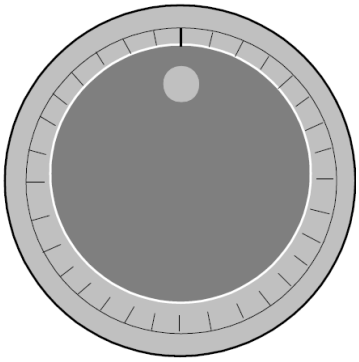
4.2 Handwheel (Hand Crank Pulse Generator) feed

Set the Bit3 bit of system parameter No.001 to 1, press the 

key to enter the handwheel operation mode, and the page is displayed as follows:



The shape of the handwheel is shown in the figure below:



4.2.1 Selection of increments

(See 4.1. Section 1).

4.2.2 Selection of movement axis and direction

In the handwheel operation mode, press the 

, 

, 

, 

keys to select the corresponding axis.

The handwheel feed direction is determined by the direction of rotation of the handwheel. In general, the handwheel is clockwise for positive feed and counterclockwise for negative feed. If sometimes the handwheel is clockwise for negative feed and counterclockwise for positive feed, the handwheel end A and B signals can be exchanged. The feed direction when the handwheel rotates can also be selected by the BIT0 position of parameter No 013.

4.2.3 Other Operations

1. Turn counterclockwise, turn clockwise, stop control

: Handwheel/ In single-step mode, press this button, the spindle rotates counterclockwise;

: Handwheel/ single step mode, press this button, the spindle stops;

: Handwheel/ In single-step mode, press this button and the spindle turns clockwise.

2. Spindle jogging

: The spindle is now jogged.

In the spindle jogging state, press the 

key to turn jogging counterclockwise; Press the 

key to turn the jog clockwise. The jogging speed of the spindle is set by data parameter No.208. See section 3.2.2 of this article for details.

3. Coolant control

See paragraph 3.2.3 of this article for details

4. Lubrication control

See paragraph 3.2.4 of this article for details

5. Manual tool change

: Handwheel/In single-step mode, press this button to release / clamp the tool .

6. Trimming of spindle magnification

Handwheel/ In single-step mode, the spindle speed can be adjusted when the analog voltage output is selected to control the spindle speed.

Use the panel spindle magnification knob switch to adjust the spindle magnification to change the spindle speed, which can realize a total of 8 levels of real-time adjustment of spindle magnification 50%~120%.

4.2.4 Descriptions

1. The relationship between the handwheel scale and the machine tool movement is shown in the following table

	The amount of movement per tick on the			
Handwheel	0.001	0.01	0.1	1
The coordinates	0.001mm	0.01mm	0.1mm	1mm

(Minimum input increment of 0.001mm as an example).

Note 1: The handwheel increment is related to the current metric imperial input state of the system and the minimum input increment of the system;

Note 2: The speed of handwheel rotation should not be higher than 5r/s, if it exceeds **5r/s, it may cause the scale value and the amount of movement to discrepancies.**

Chapter V: Input Operations

In the input operation mode, you can set parameters, enter code words, and execute code words.

The function of the buttons in the system machine panel is defined by the PLC program (ladder diagram), and the functional significance of each button can be found in the machine manufacturer's manual.

Please note that the following functions related to the operator panel buttons in this chapter are described for the standard PLC program of the system!

5.1 Code Word Input

Select the input operation mode, enter the program status page, enter a program segment G00 X50 Z100, the operation steps are as follows:

1. Press the 

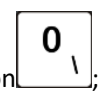
key to enter the input operation mode;

2. Press the 

key to enter the program monitoring interface, and then press the F3 key [MDI] under the screen to enter the program editing status page under Input:

3. Type the address key 

, number key 

, and so on 

;

4. Type the address key 

, the number key 

, and the number key in turn 

5. Type the address key 

, number key 

, 

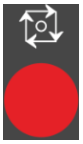
, 

, in turn;

After performing the above operations, the page is displayed as follows (4 programs can be entered, 6 programs are displayed):

5.2 Code Word Execution

After entering the code word, press the F1 key [Confirm] under the screen, and the page is displayed as follows:

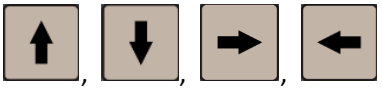
Press the  key to execute the entered program segment. During operation, you can press  keys,  keys, and emergency stop buttons to stop the program segment.

Note: Subroutine call code (M98P; etc) Execution under MDI is invalid.

5.3 Setting of Parameters

In the input mode, you can modify the parameter values by entering the parameter interface, as detailed in Chapter 10 of this article.

5.4 Modification of Data

Under the program status page, if there is an error in the field input process before executing the entered program segment, you can use the  and

arrow keys to move the cursor to the error, and press the 

key to move forward to delete the program modification.

5.5 Other Operations

1. In the adjustable spindle magnification handwheel/single-step mode, the spindle speed can be adjusted when the analog voltage output is selected to control the spindle speed. Use the spindle magnification knob switch to adjust the spindle magnification to change the spindle speed, which can realize a total of 8 levels of real-time adjustment of spindle magnification 50%~120%.
2. The fast magnification button can be adjusted, and the fast moving feed speed can be adjusted to achieve 4-level real-time adjustment of fast moving speed.
3. Can be repaired to adjust the feed rate
   Under the input operation mode, use the panel feed rate knob switch to adjust the feed rate to change the feed speed , which can realize the real-time adjustment of 16 levels of the actual speed of 0 ~ 150% of the feed rate specified by the F code.

Chapter VI: Program Editing and Management

In the editing operation mode, the program can be established, selected, modified, copied and deleted, and two-way communication between CNC and CNC and CNC and PC can also be realized.

In order to prevent the program from being accidentally modified and deleted, the system sets the program switch. Before editing the program, you must turn on the program switch, which is set in section 10.1.1 of this article.

For ease of management, the system provides 3 levels of user permission settings. You must have an operation level of more than 4 (level 4, level 3, etc.) to turn on the program switch and edit the program. The actions allowed for each operation level are described in Section 10.3.

6.1 Establishment of Programs

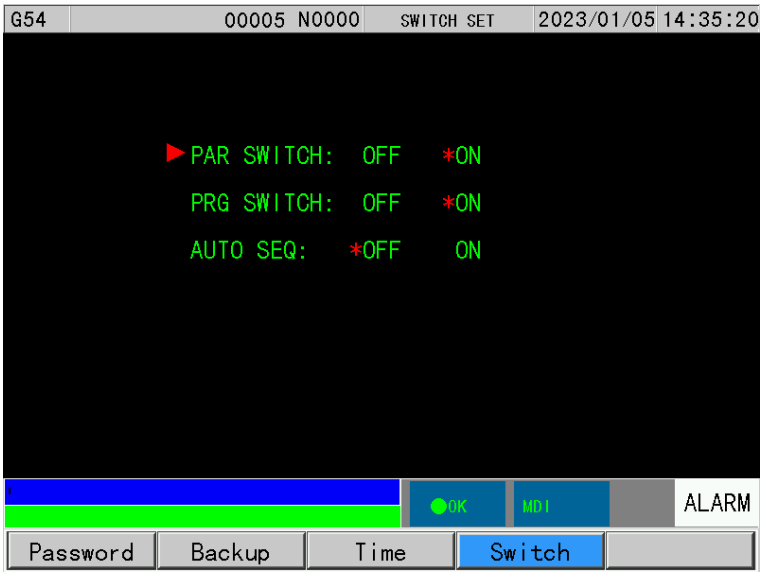
6.1.1 Generation of program segment numbers

In a program, the program segment number can be programmed and not numbered, and

the program is executed in the order in which the program segment is programmed (except when called).

When the "Auto Serial Number" switch on the switch setting page is off, the CNC does not automatically generate the program segment number, but the program segment number can be manually programmed during programming.

When the switch setting page "Auto serial number" switch is on, CNC automatically generates the program segment number, when editing, press the Enter key to automatically generate the program segment number of the next program segment, and the incremental value of the program segment number is set by the CNC data parameter No 316 (The settings for automatic sequence numbers are described in Section 10.1.1 of this article).



6.1.2 Input of program content

1. Press the PROG FILE key to enter the program interface.

2. Type the address key and then the number key, use the Enter key to wrap the line to edit, use the ↑, ↓, →, ← arrow keys to move the cursor to the error, and press the BACK SPACE key to move forward to delete to modify the program

3. Enter one by one according to the programmed part program, each input character, immediately give the input character displayed on the screen (the processing of the composite key is to press this composite key repeatedly to achieve alternate input), and after completion, you need to switch to other working modes to store the program.

6.1.3 Cursor movement

1) Press the PROG FILE key to enter the program content display page;

2) Press the ↑ key, ↓ key to move the program cursor up and down between the lines of the program segment;

3) Press the → key and ← key to move the program cursor left and right in the current program segment;

4) Press the  key to quickly move the program cursor left and right in a column;

5) Press the  key or  keys to turn the program page up and down;

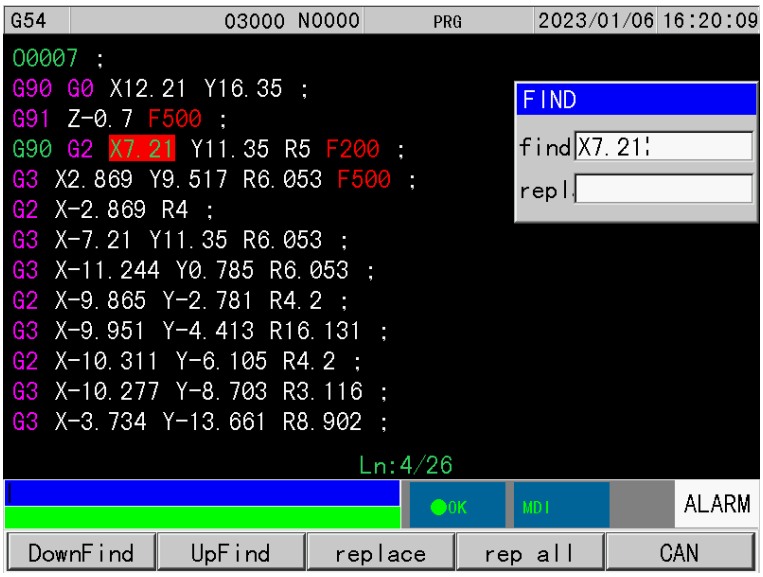
6) In the program display page, press the  key and the cursor returns to the beginning of the program.

6.1.4 Search for words and line numbers

Word retrieval: Looks up the specified character up or down from the current position of the cursor

The steps for the search method are as follows:

1) Press  the key to enter the program content page;
2) Press the F2 key [Find/Replace] under the screen to pop up the search dialog box, enter the word you want to retrieve;
3) Press the F1 key [Find down] or F2 key [Find Up] under the screen to display the page as follows:



4) You can also enter the replacement content in the replacement column in the dialog box, and then press the F3 key [Replace] or F4 key [Replace All] under the screen to replace and modify;

Note 1: If the word entered is not found, the words "Search to the end of the program, whether to start from the beginning" will pop up.

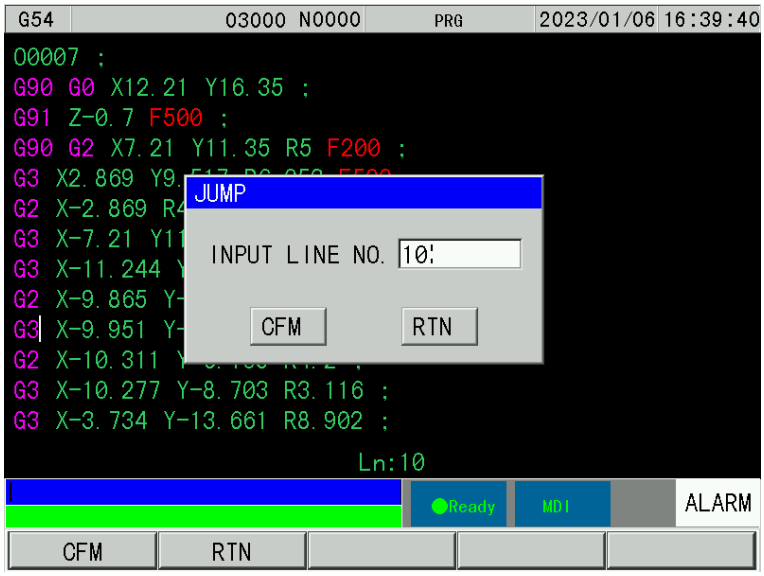
Note 2: Character retrieval and scanning is not possible in macro editing mode.

Note 3: In character retrieval, characters in the called subroutine are not retrieved, and characters in the subprogram are retrieved in the subprogram.

Line retrieval: Quickly position the cursor on a specified line in the program.

The steps for the search method are as follows:

1) The program content display page;
2) Press the F5 key [Jump] under the screen to appear the jump line number input dialog box;
3) Enter the program line number;
4) Press the F1 key [Confirm] on the screen and the cursor will jump to the entered program line number, as shown in the figure below.



6.1.5 Insertion of words

The how-to steps are as follows:

1) Press the PROG FILE key to enter the program content page;
2) Press the ↑ key, ↓ key to move the cursor up and down between the lines of the program segment;
3) Press the → key, ← key to move the cursor left and right in the current program segment;
4) Enter the words you want to insert, and the system will edit the insertions one by one to the left of the cursor.

6.1.6 Deletion of Words

The how-to steps are as follows:

1) Press the 

key to enter the program content page;

2) Press the  key,  key to move the cursor up and down between the lines of the program segment;

3) Press the  key,  key to move the cursor left and right in the current program segment;

4) Position the cursor to the position that needs to be deleted, press the 

key, and the system will delete one by one with the position of the cursor to the left;

6.1.7 Modification of Words

The how-to steps are as follows:

1) Press the 

key to enter the program content page;

2) Press the  key,  key to move the program cursor up and down between the lines of the program segment;

3) Press the  key and  key to move the program cursor left and right in the current program segment;

4) Move the cursor to the right of the word you need to modify, and then press the 

key, the system will delete the cursor to the left one by one;

5) Then enter the desired content in order at the position where the cursor stays.

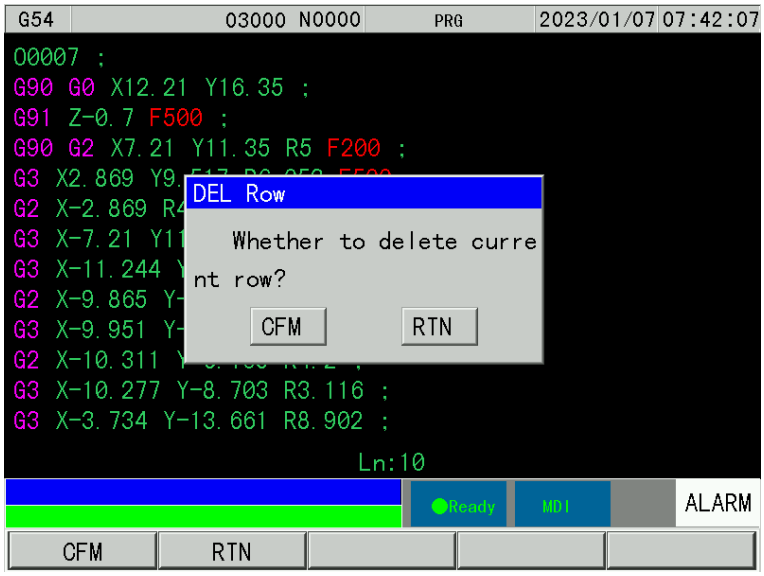
6.1.8 Deletion of Single Program Segments

The how-to steps are as follows:

1) Press the PROG FILE key to enter the program content page;

2) Press the ↑ key, ↓ key to move the program cursor up and down between the lines of the program segment;

3) Move the cursor to the delete program segment, then press the ▶ screen down key to turn the page back, and then click the F1 key [Delete Line] at the bottom of the screen, at this time the Delete Current Line dialog box will pop up, click the F1 key [Confirm] to delete the segment where the cursor is located, as shown in the figure below.



6.1.9 Deletion of Multiple Program Segments

The how-to steps are as follows:

1) Press the PROG FILE key to enter the program content page;

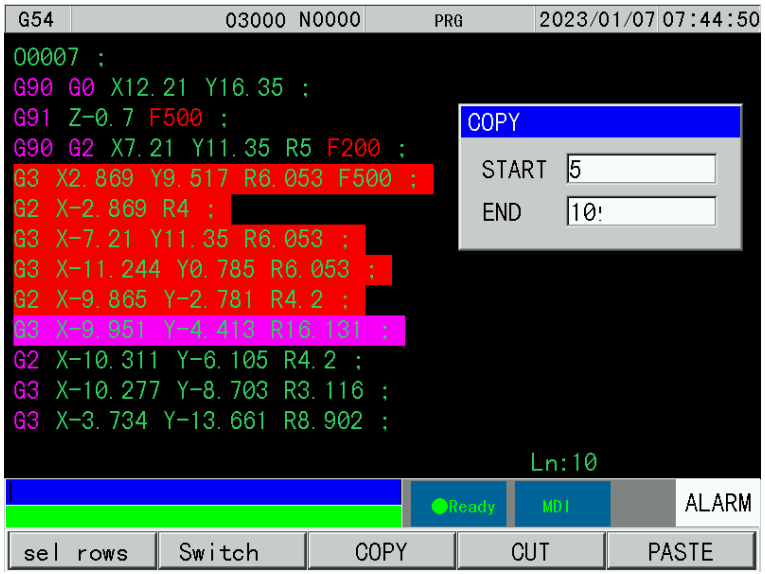
2) Press the ↑ key, ↓ key to move the program cursor up and down between the lines of the program segment;

3) Move the cursor to the first or last line of the deleted program segment;

4) Click the F4 key [Region Copy] under the screen to enter the region selection state, and then click the F1 key [Line Selection] under the screen, at which time the row selection dialog box will pop up;

5) Press the  key or  keys up or down to select End Row OK;

6) Press the F4 key [Cut] under the screen, and the system will select the program cut between the start line and the end line, as shown in the figure below.



6.1.10 Block Deletion

(Reference 6.1. 9 knots operation)

6.1.11 Replication of Single Program Segments

The how-to steps are as follows:

1) Press the 

key to enter the program content page;

2) Press the  key,  key to move the program cursor up and down between the lines of the program segment;

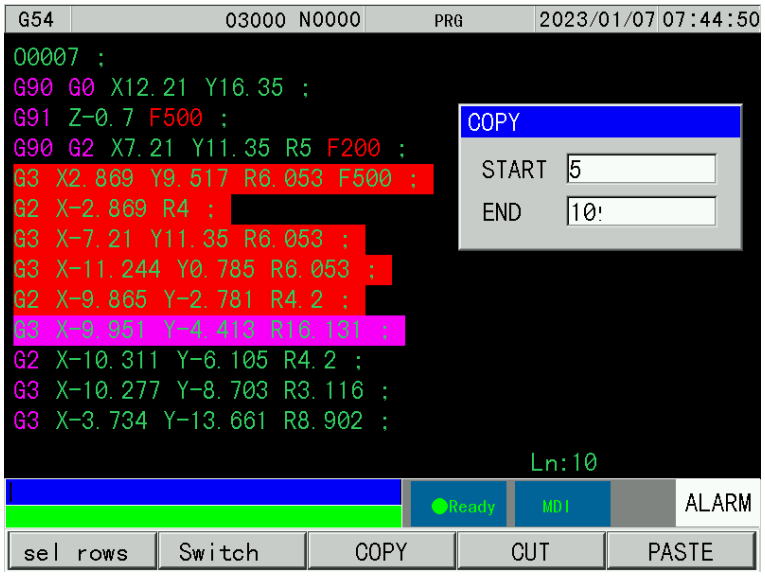
3) Move the cursor to the first or last line of the copied program segment;

4) Click the F4 key [Region Copy] under the screen to enter the region selection state, and then click the F1 key [Line Selection] under the screen, at which time the row selection dialog box will pop up;

5) Press the  key or  keys up or down to select End Row OK;

6) Press the F3 key [Copy] under the screen, and the system will copy the program between the start line and the end line of the region selection;

7) Move the cursor to the line where you need to add the copied segment, and then press the F5 key [Paste] under the screen, and the system will paste the previously copied program segment, as shown in the figure below.



6.1.12 Replication of multiple program segments

(Refer to Section 6.1.11).

6.1.13 Copying of Blocks

(Refer to Section 6.1.11).

6.1.14 Pasting of program blocks

(Refer to Section 6.1.11).

6.2 Deletion of Programs

6.2.1 Deletion of Individual Programs

The procedure is as follows:

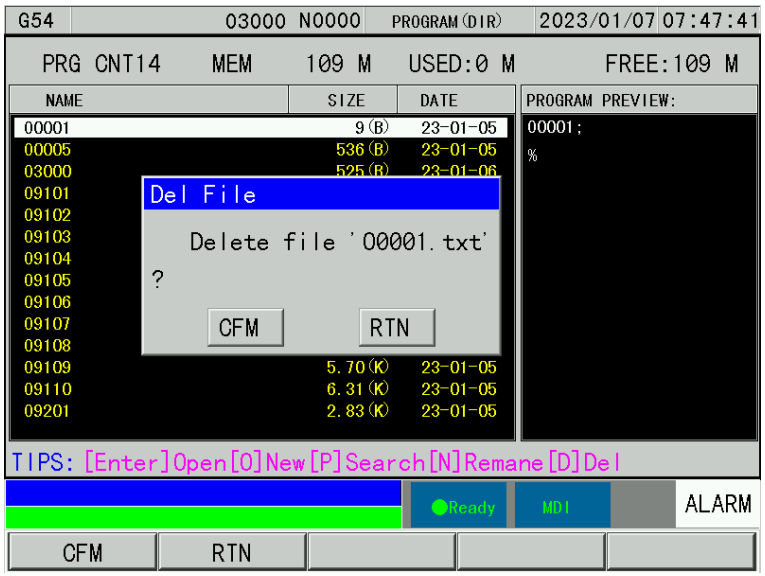
1) Press the 

key to enter the program content page, and then press the F3 key [File Management] under the screen to enter the local directory page;

2) Press the  key and  key to move the cursor bar to the name of the program to be deleted;

3) Press the  screen down arrow to turn the page back, and then press the F2 key [Delete] under the screen, at which time the file deletion dialog box will pop up;

4) Press the F1 key [Confirm] under the screen to delete the file, as shown in the following figure.



6.2.2 Deletion of All Programs

(Currently, all programs are not supported, and individual programs are deleted one by one).

6.3 Selection of Programs

When there are already multiple programs in the CNC, you can select the program in the following ways.

6.3.1 Search Method

(The operation mode has been canceled).

6.3.2 Scanning method

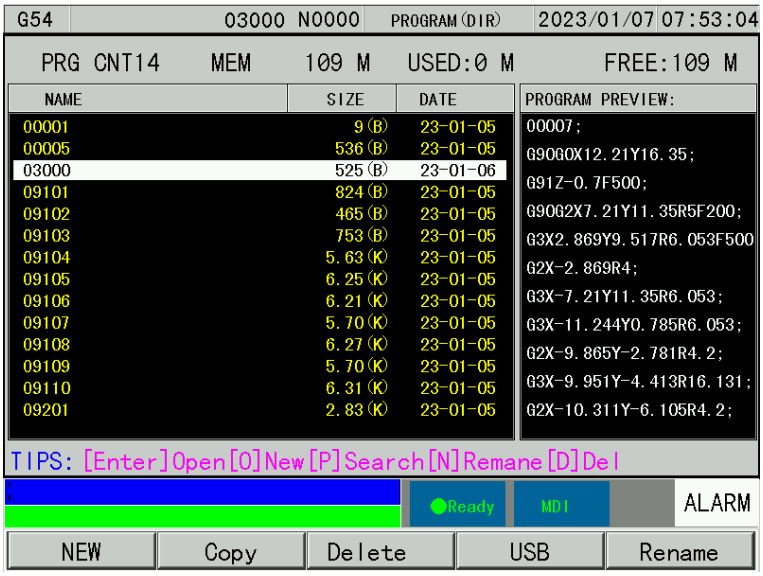
(The operation mode has been canceled).

6.3.3 Cursor confirmation method

1) Must be in a non-operational state;

2) Press the PROG FILE key to enter the program content page, and then press the F3 key [File Management] under the screen to enter the local directory page;

3) Press the PAGE ▲ key, PAGE ▼ key to page up and down, and then press the ↑ key and ↓ key to move the cursor to the name of the program to be selected (when the cursor moves, the program content also changes), as shown in the following figure;



4) Press the Enter key to open the program.

6.4 Renaming of the Program

1) Must be in a non-operational state;

2) Press the PROG FILE key to enter the program content page, and then press the F3 key [File Management] under the screen to enter the local directory page;

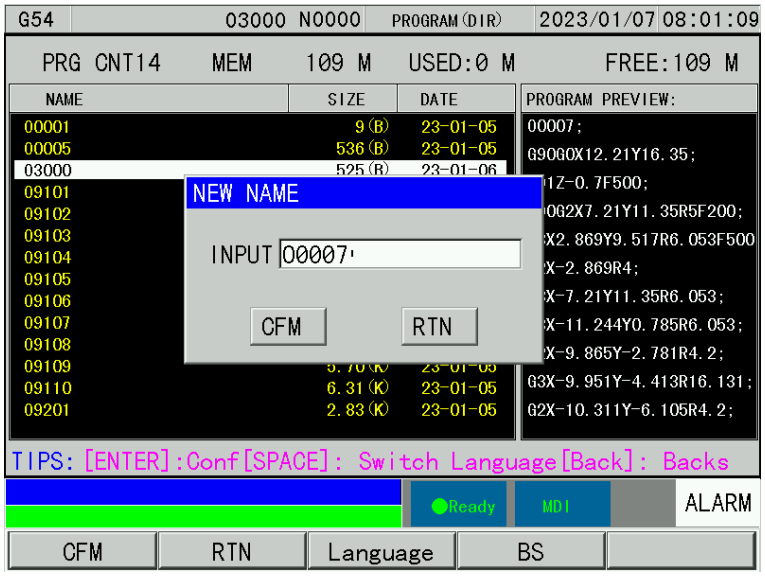
3) Press the PAGE ▲ key, PAGE ▼ key to page up and down, and then press the ↑ key

and  key to move the cursor to the name of the program to be selected (when the cursor moves, the program content also changes);

4) Press the  screen down key to turn the page back, and then press the F1 key [Rename] under the screen, at which time the rename dialog box will pop up;

5) Press the F4 key [Backspace] under the screen to delete the original program name;

6) Use the letter and number keys to re-enter the new file name, press the F1 key [Confirm] on the screen to complete the program name modification, as shown in the figure below.



Note: When modifying the program name, you can press the F3 key [Chinese-English switch] under the screen to switch to Chinese input mode.

6.5 Reproduction of Programs

To save the current program:

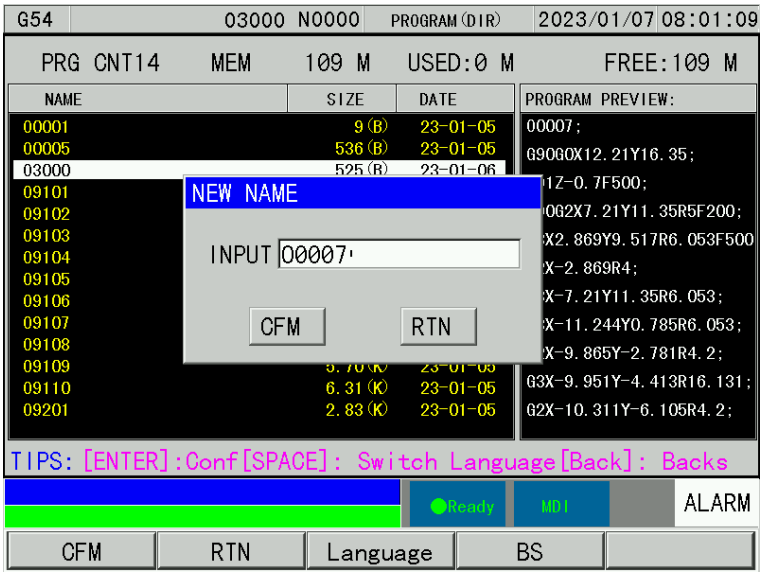
1) Press the  key to enter the program content page, and then press the F3 key [File Management] under the screen to enter the local directory page;

2) Press the  key,  key to page up and down, and then press the  key and  key to move the cursor to the name of the program to be selected (when the cursor moves, the program content also changes);

3) Press the F3 key [Copy] on the screen, and the copy new program name input dialog box

will pop up;

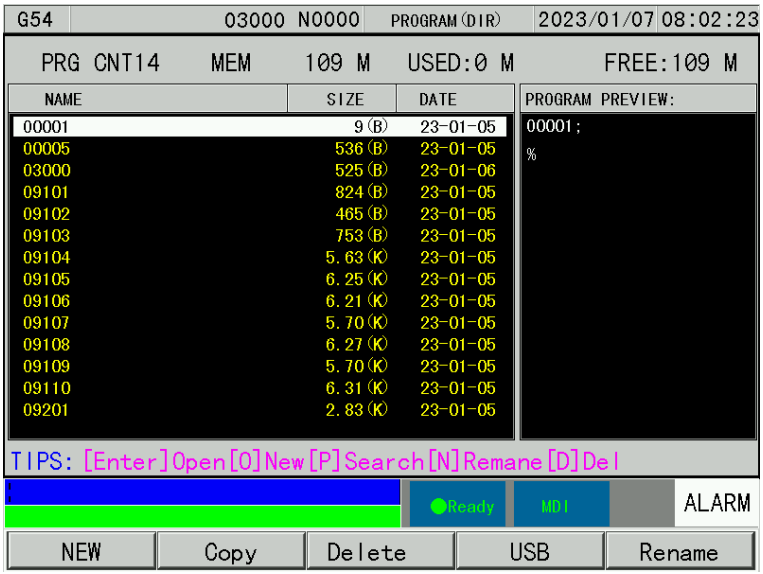
4) Type the new file name in turn, and then press the F1 key [Confirm] under the screen to complete the copy of the program, as shown in the figure below.



6.6 Program Management

6.6.1 Program Directory

Press the PROG FILE key to enter the program content page, and then press the F3 key [File Management] under the screen to enter the local directory page. In this page, the CNC's saved program names are displayed in the form of a table of contents, up to 14 program names can be displayed on one page, and when the number of saved programs exceeds 14, you can press the up/down key to display another page of the program catalog.



1) Open the program

Open the specified program:

Use the key  or  keys to move up or down to find the program you want to open, and tap the screen key  to open the program.

2) Removal of the program:

Use the  key or  keys to move up and down to find the program you want to remove, and tap the screen key  to delete the program.

6.6.2 Number of stored programs

The system can store up to 400 programs, which shows the number of parts programs that CNC can currently store and remain.

6.6.3 Storage Capacity

The program storage capacity of this system (total capacity is 109M), this page shows the current number of stored programs, as well as the occupied storage capacity and remaining storage capacity.

Chapter VII Automatic Operation

The function of the buttons in the system machine panel is defined by the PLC program (ladder diagram), and the functional significance of each button can be found in the machine manufacturer's manual.

Please note that the following functions related to the operator panel buttons in this chapter are described for the standard PLC program of the system!

7.1 Autorun

7.1.1 Selection of Runners

Press the system panel 

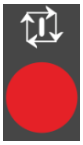
key to enter the program content page, then press the F3 key [File Management] under the screen to enter the local directory page, use the  key or  keys to move up and down to find the program you need to open, and click the screen key 

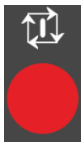
to open the program.

7.1.2 Startup of Autorun

1. Press the 

key to select the automatic operation mode;

2. Press the  button to start the program and the program runs automatically.

Note: The program starts on the line where the cursor is located, so you should check if the cursor is on the section of the program that needs to be run before pressing the  **key to run.**

7.1.3 Stop of Auto-Run

* Code stop (M00).

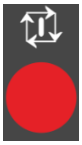
1. M00

   After the program segment containing M00 is executed, it stops running automatically, and all modal functions and states are saved. After pressing the panel button  or the external

run button, the program continues.

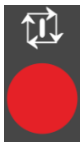
2. M01

Press the 

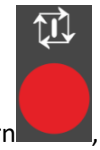
key, the select stop indicator is on, and the stop function is active. After executing the program segment containing M01, the automatic operation is stopped, and the modal functions and states are all saved. After pressing the panel button  or the external run button, the program continues.

* Press the relevant key to stop

1.  When the button or the external pause button is pressed in automatic operation, the machine is in the following state:

1) Machine tool feed deceleration stop;
2) modal functions, states are saved;
3) After pressing the  button, the program continues.

2. Press the reset key 

1) All axis movements stop;
2) M, S function output is invalid (can be set by the parameter whether to automatically turn off the spindle counterclockwise / clockwise turn  , lubrication, cooling and other signals after pressing the button);
3) The automatic operation ends, and the modal function and state are maintained.

3. Press the emergency stop button during the operation of the machine tool in danger or emergency pressing the emergency stop button (when the external emergency stop signal is valid), the CNC enters the emergency stop state, at this time the machine tool movement immediately stops, and all outputs (such as spindle rotation, coolant, etc.) are turned off. Release the emergency stop button to release the emergency stop alarm, and the CNC enters the reset state.

4. When the conversion operation mode is converted to the machine tool zeroing, handwheel/

single-stepping, manual, and program zeroing mode during automatic operation, the current program segment is immediately "paused"; When switching to edit and enter mode during automatic run, "Pause" is displayed only after the current program segment has finished running.

Note 1: Confirm that the fault has been eliminated before dismissing the emergency stop alarm;

Note 2: Pressing the emergency stop button before power-up and shutdown reduces the electrical shock of the device;

Note 3: After the emergency stop alarm is lifted, the operation back to the zero point of the machine tool should be performed again to ensure the correctness of the coordinate position (if the machine tool is not installed with the zero point of the machine tool, the zero point operation of the machine tool shall not be carried out);

Note 4: External emergency stop **is valid only if** the **Bit2 bit (EALM) of status** parameter **No.60 is set to 0.**

7.1.4 Run automatically from any segment

In the non-running state, press the key to enter the program interface, press the 

key 

or 

key to select the program content page;

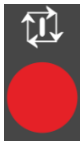
1、  key or  keys to move the cursor to the section of the program that is ready to start running (e.g. from the second line, move the cursor to any position on the second line, as shown in the figure below);

2、 If the mode (G, M, T, F code) of the program segment where the current cursor is located is

default and inconsistent with the mode of the program segment in which it is running, the corresponding modal function must be executed before proceeding to the next step;

3、 Click the F1 key [Perform Processing] under the screen, and the system page returns to the program supervision interface;

4、 Press the 

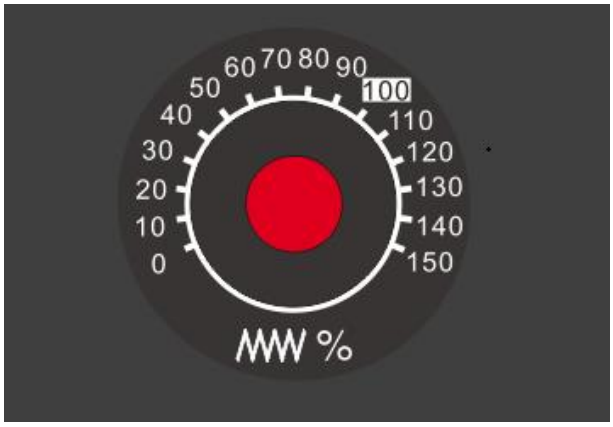
key to enter the automatic operation mode, press  the button to start the program to run.

7.1.5 Feed, fast speed adjustment

When running automatically, the running speed can be changed by adjusting the feed and moving magnification without changing the speed value set in the program and parameters.

* Adjustment of feed rate

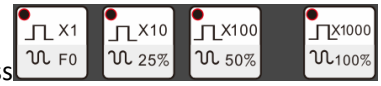
Press the feed rate knob on the secondary panel to select, which can realize 16 levels of real-time adjustment of the feed rate.



Note 1: The value specified **by F in the feed rate adjustment program;**

Note 2: Actual feed rate = value specified in F × feed rate.

* Quick magnification adjustment

Press 

the button to achieve fast rate FO, 25%, 50%, 100% real-time adjustment.

Note 1: CNC parameter No.090 **~ No.094 sets the fast moving rate of each axis respectively;**

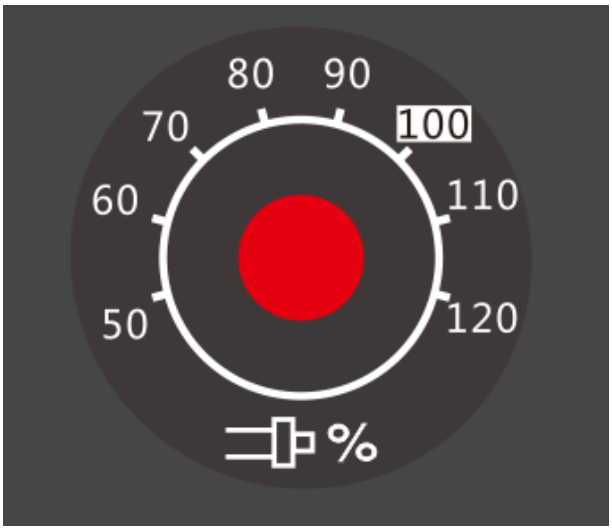
Actual fast movement rate = value set by the parameter × fast magnification

Note 2: When the fast magnification is **F0, the minimum rate of fast movement is set by CNC parameter No.085.**

7.1.6 Spindle speed adjustment

In automatic operation, the spindle speed can be adjusted when the analog voltage output is selected to control the spindle speed.

Use the spindle magnification knob on the secondary panel to select, adjust the spindle magnification to change the spindle speed, and realize 8 levels of real-time adjustment of spindle magnification 50% ~ 120%.



Note: The actual output analog voltage value = the **analog voltage value calculated by parameters × the spindle magnification**

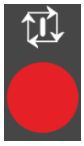
7.2 DNC runs

CNC has DNC function, which can be connected to CNC through DNC communication software or U disk to realize the operation of large-capacity programs.

The sequence of steps is: press the machine panel key 

, enter the DNC mode, press the 

key to enter the program management on the premise that the PC end is ready, then press the F3 key [File Management] under the screen to enter the file directory, and then press the F4 key [U disk directory] under the screen Enter the U disk program directory, click the F2 key [Switch] under the screen to switch to the U disk, use the  key or  keys to find the desired program, click the key to open the program, click the F1 key [Perform Machining] under the screen, press the machine panel cycle start button 

, and start the program 

DNC processing.

Note: DNC processing is used as much as possible for large-capacity programs, which reduces the time consumed by the program during copy transfer and the read time when the local program is opened.

7.3 Status at runtime

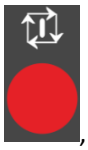
7.3.1 Single-segment operation

When executing the program for the first time, to prevent unexpected programming errors, you can choose to run in a single stage.

In the automatic operation mode, the single-stage program switch is turned on as follows:

Press the 

key to make the single-stage operation indicator in the status indication area turn on, indicating that the single-stage operation function is selected;

When running in a single segment, the CNC stops running after the current program segment is executed; When proceeding to the next program segment, press the button again  , and so on until the program finishes running.

Note 1: In the **G28 code, at the midpoint position, a single segment stops;**

Note 2: The execution call subroutine (M98) **and subroutine call return code (M99) single program segment are invalid. However**, in **the M98 and M99 program segments, except** for addresses other than **N**, **O**, **P,** the **single segment stops working.**

7.3.2 Handwheel simulation

Before running the program automatically, in order to prevent unexpected programming errors, you can select the handwheel simulation state to verify the operation of the program. In the automatic operation mode, the handwheel analog switch is turned on as follows:

In automatic mode, press the 

key to make the handwheel simulation indicator in the status indicator area light up, indicating that the handwheel test cutting state has been entered.

In the handwheel test cutting state, the machine tool feed and auxiliary functions are effective (if the machine tool locks, the auxiliary lock switch is in the off state), that is to say, the state of the handwheel test cut switch has no effect on the machine tool feed and auxiliary function execution, and the actual running maximum speed is determined by the data parameter No 0 149 settings.

7.3.3 The machine locks the operation

In the automatic operation mode, the machine tool locks the switch and opens as follows:

Press the 

key to make the machine tool lock running indicator in the status indication area on, indicating that the machine tool lock running state has entered;

Machine locking operation is often used in conjunction with the auxiliary function locking function for program calibration. Machine locked operation:

1) The machine tool drag plate does not move, the "machine tool coordinates" in the comprehensive coordinate page under the position interface does not change, and the relative coordinates, absolute coordinates and residual movement display are constantly refreshed, the same as when the machine tool lock switch is closed;
2) M, S, and T code can be executed normally.

7.3.4 Accessibility Lockup Operation

In the automatic operation mode, the machine tool locks the switch and opens as follows:

Press the 

key to make the auxiliary function lock running indicator in the status indication area 

turn on, indicating that the auxiliary function lock running state has been entered;

At this time, the M, S, T code is not executed, and the machine drag plate moves. It is usually used in conjunction with the machine locking function for program calibration.

Note: When the accessibility lock is active, it does not affect the execution of M00, M30, M98, M99.

7.3.5 Program segment selection

When you do not want to execute a certain segment of the program in the program and do not want to delete it, you can select the program segment selection function. When a program segment has a "/" sign at the beginning and the program segment selector switch is turned on (the machine panel button or program selector external input is valid), the program segment skip does not run when running automatically.

In the automatic operation mode, the program segment selection switch is turned on as follows:

Press the 

key to make the program segment selection indicator in the status indication area light on;

Note: (1) When the program segment **selection switch is not turned on**, the program segment with the "**/" sign at the beginning of the program segment** will **not be skipped when running automatically, and it will be executed as usual.**

(2) When there are two or more "/" signs before the program is completed, it means that the program

segment is not executed, and the paragraph will not be executed if the segment jump function is turned on or off.

7.4 Other Operations

1、 Automatic operation mode, button, coolant on 

/ off switch;

2、 Press any key in the 

、 

、 

、 

、 

or 

key to realize the conversion of the operation mode;

3、 Press the 

key to reset the CNC .

4、 Automatic lubrication function (see Chapter 3 of this article for details).

Chapter 8 Zeroing Operations

The function of the buttons in the system machine panel is defined by the PLC program (ladder diagram), and the functional significance of each button can be found in the machine manufacturer's manual.

Please note that the following functions related to the operator panel buttons in this chapter are described for the standard PLC program of the system!

8.1 Machine tool returns to zero

8.1.1 Machine tool zero

Machine tool coordinate system is the reference coordinate system of CNC for coordinate calculation, is the inherent coordinate system of the machine tool, the origin of the machine tool coordinate system is called the machine tool zero point (or machine tool reference point), the machine tool zero point is determined by the zero point switch or zero return switch on the installation of the machine tool, usually the zero point switch or zero return switch is installed at the maximum stroke in the positive direction of the X axis and Z axis.

8.1.2 Operation steps of machine tool zeroing

1、 Press the 

button to enter the machine tool zero return operation mode, and the bottom line of the display page displays the words "mechanical zero return", which is displayed as follows:

2、 Press the 

, 

, 

or 

keys to select back to the X, Y, Z, or 4th axis machine zero;

3、 The machine tool moves in the direction of the zero point of the machine tool, and returns to the zero point of the machine tool after the deceleration signal and zero signal detection, at which time the shaft stops moving, and the end indicator light of zero return is on.

Indicator indicating the end of return to zero

Note 1: If the CNC machine tool is not equipped with the machine zero point, the machine tool return zero operation shall not be used;

Note 2: The end of zero indicator turns off in the following cases:

1) Move out from the zero point;

2) CNC power off;

Note 3: After returning to the zero point of the machine tool, the **CNC cancels the tool length compensation;**

Note 4: The parameters related to the zero return of the machine tool are detailed in the fourth part of "Installation and Connection";

Note 5: After performing the machine zero return operation, the original workpiece coordinate system is reset.

8.2 Other button functions of the panel

1、 Buttons 

, spindle turns counterclockwise;

2、 Press the 

button, the spindle stops;

3、 Buttons 

, spindle clockwise;

4、 Button 

, coolant on/ off switch;

5、 lubrication control (see Chapter 3 of this article for details);

6、 Buttons 

, manual loosening / clamping knife;

7、 Press the 

key to blow on/off toggle;

8、 Press the 

key to switch chip conveyor on/off;

9、 Press the 

key, the work light on/off switch;

10、Press the 

, 

key, K1/K2 corresponds to the output on/off switch;

11、Trimming of spindle magnification (see Section 7.1.6 for details);

12、Fast magnification tuning (see 7.1 5 chapters);

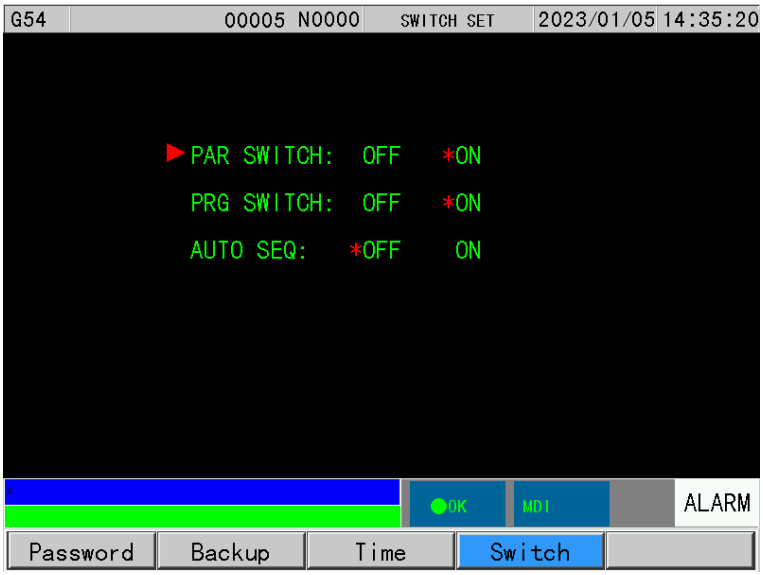
13、Adjustment of the feed rate (see 7.1 5 chapters).

Chapter 9 Data Setting, Backup, and Recovery

9.1 Setting of Data

9.1.1 Switch Settings

On the switch setting page, you can display and set the on and off status of parameters, programs, and automatic serial numbers, and the page is displayed as follows:



1、 Press the screen  and  key to switch the function display, find the F1 key [Settings] under the screen, and click the F1 key under the screen to enter the setting page;

2、 Press the  or  key to move the cursor over the item you want to set;

3、 Press  and  key to toggle switch status, button  , "*" to move left, turn off switch, press button  , "*" to move right, open switch;

Parameters can only be modified when the parameter switch is on; Programs can only be edited when the program switch is on; The sequence number of the program segment is automatically added only when the automatic sequence number switch is turned on.

Note: (1) When the parameter switch **is switched from "off" to "on"**, the **CNC** will **appear an** alarm, the 

button can eliminate the alarm, if you switch the switch state of the parameter again, there will be no alarm. For safety reasons, after the parameter modification is completed, be sure to set the parameter switch to "Off".

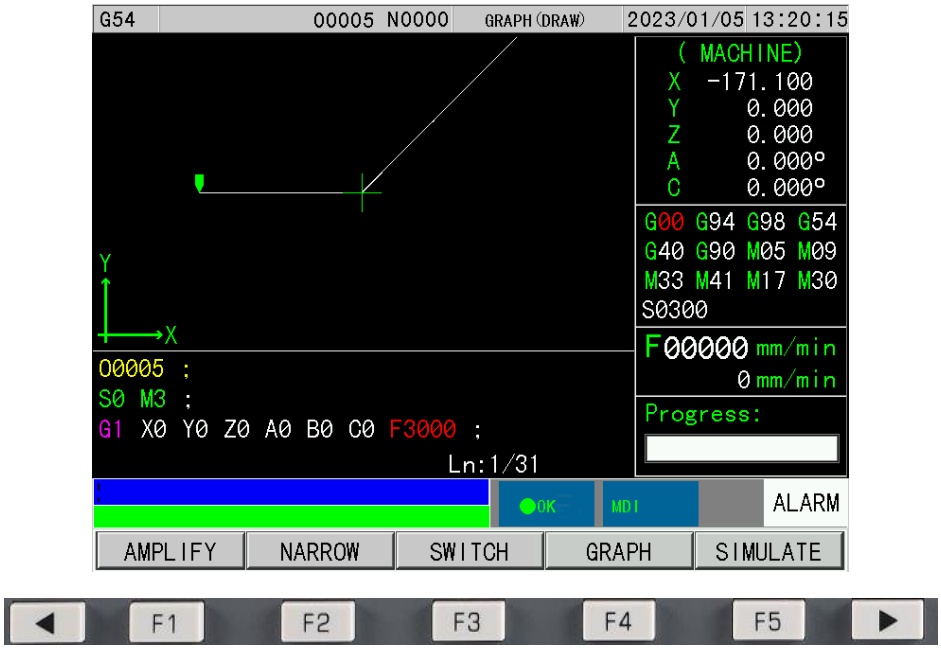
(2) BIT 0 of status parameter No. 10 can set whether the parameter switch is turned on when booting.

9.1.2 Graphics Settings

Press the panel 

key to enter the program monitoring interface, and then press the

F2 key [Graphics Settings] under the screen to enter the graphical interface, you can zoom in, zoom out, switch, generate graphics, simulate graphics, clear and other operations.



The meaning of the graphical parameters

Coordinate system settings: multiple graphic directions can be switched according to different coordinate displays;

1) Zoom in and out of the graph trajectory

On the graphic display page, you can zoom in and out of the graphic trajectory in real time by pressing the F1 key [Zoom in] and the F2 key [Zoom Out] on the screen.

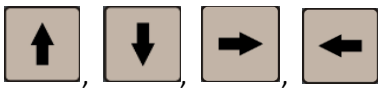
2) Graph track display start, clear

On the graphic track display page, press the F4 key [Graph Generation] once to start automatic one-click drawing;

On the graphic trajectory display page, press the F5 key [Graphic Simulation] once to start to quickly display the path pattern according to the program running trajectory;

On the page where the graph track is displayed, press F2 [Clear] once after turning the page back to clear the current graphic track.

3) The movement of the graphical trajectory display

On the graph track display page, you can press the  arrow keys to move the graph track.

9.1.3 Setting of Parameters

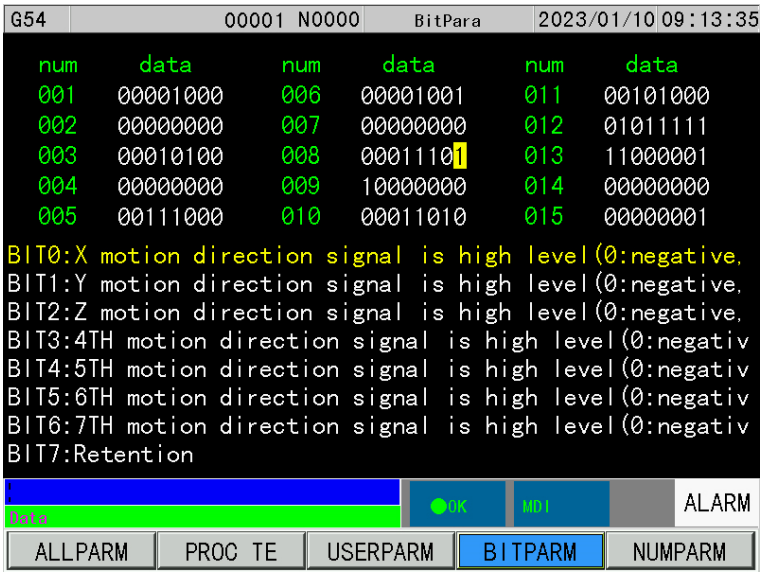
Find the F2 key [Parameter Settings] by pressing the screen  and 

toggle keys, and the parameter settings can adjust the relevant characteristics of the drive unit, machine tool, etc. The significance of each parameter is detailed in Appendix I

Press the F2 key [Parameter Settings] key to enter the parameter interface, then press the F4 key [Bit Parameter] under the screen to enter the status parameter page, press the 

or 

key to switch between the parameter pages, as shown in the following figure:



A. Modify the settings of bit parameters

1、 Word position modification:

1) Turn on the parameter switch;

2) Move the cursor over the parameter number you want to set:

Method 1: Press 

or 

key to the page where the parameter to be set is located,

and press the  key or  key to move the cursor to the parameter number to be set;

Method 2: Press the address key 

, parameter number and 

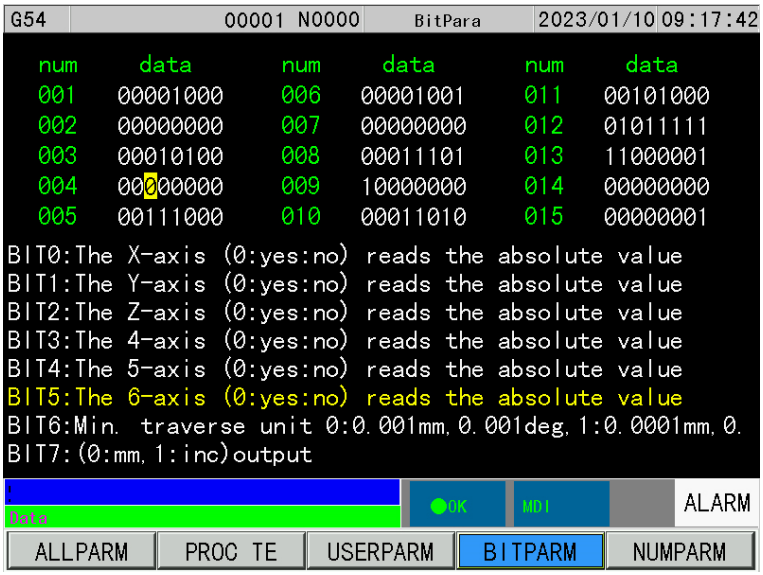
key.

3) Enter new parameter values;

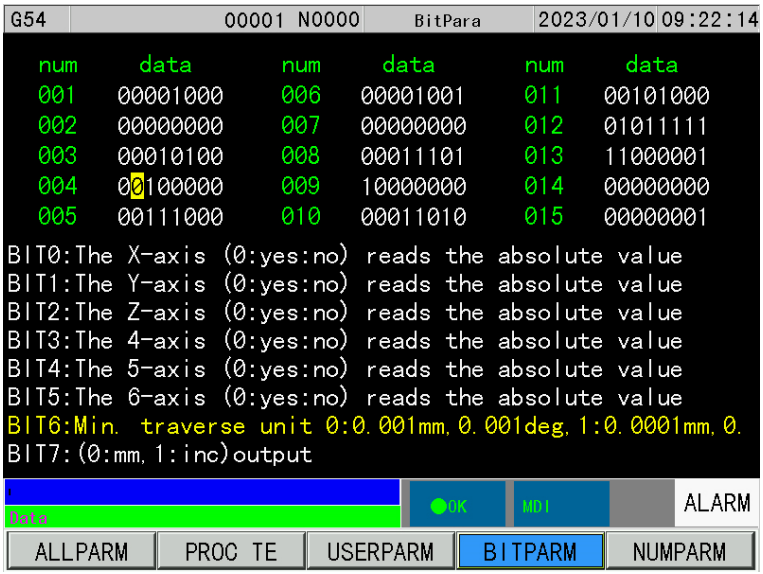
4) Press the number 0 \ or 1] key, the parameter value is entered and displayed;

5) For safety reasons, after all parameters are set, the parameter switch needs to be turned off.

Example: Set Bit5 of the status parameter No 004 to 1, leaving the rest unchanged. Follow the steps above to move the cursor over No 004, as shown in the following image:



Press the number 1] keys to modify the parameters. The display page is as follows:



2、 Modify by section:

1) Turn on the parameter switch;

2) Move the cursor over the parameter number you want to set;

Method 1: Press PAGE or PAGE key to the page where the parameter to be set is located, and press the ↑ key or ↓ key to move the cursor to the parameter number to be set;

Method 2: Press the address key P, parameter number and Enter key.

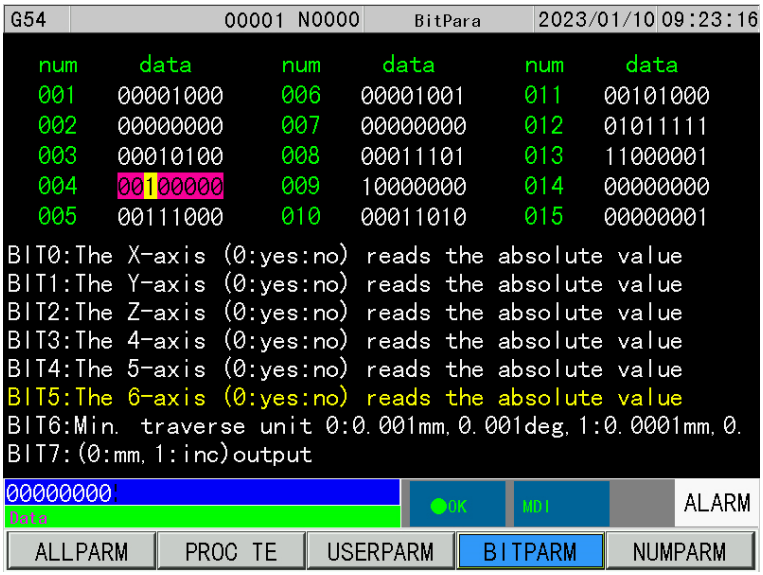
3) Press the SPACE jump into the parameter to enter the modification status for the entire line of the parameter, and the serial number is inverted in purple.

4) Enter the serial number in the status bar, press the Enter key to confirm the modification, and exit the entire line modification status after confirmation.

5) For safety reasons, after all parameters are set, the parameter switch needs to be turned off;

Example: Set the Bit5 bit of the status parameter No 004 to 1, and leave the rest unchanged.

Follow the steps above to move the cursor over No 004 and modify the status by SPACE jumping into the parameter serial number. As shown in the following figure:



B. Modification settings of data parameters and screw supplementary data

1) Turn on the parameter switch;

2) Move the cursor over the parameter number you want to set;

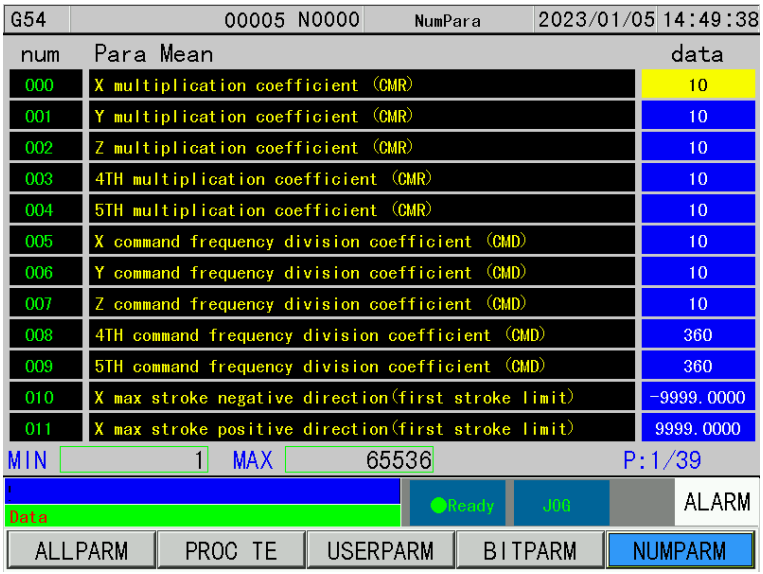
3) Enter new parameter values;

4) Press the Enter key, the parameter value is entered and displayed;

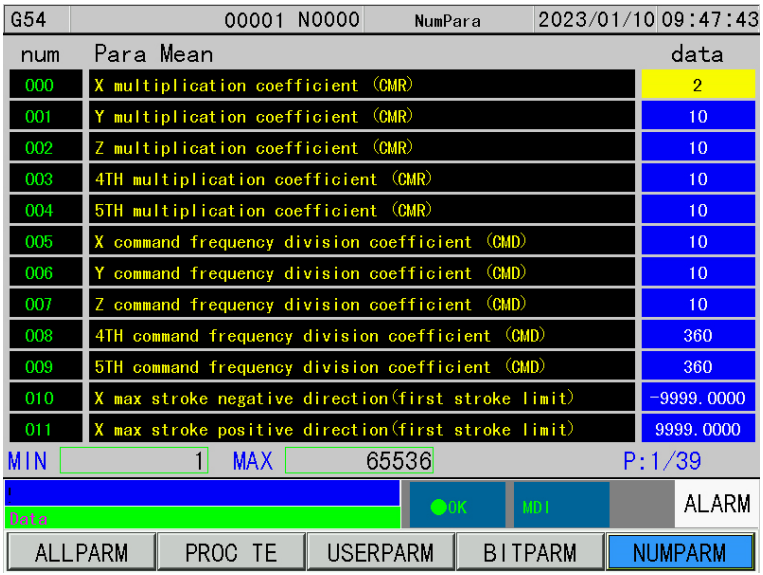
5) For safety reasons, it is recommended to turn off the parameter switch after all parameters have been set.

Note: The screw supplement data can only be modified under the second-level operation authority.

Example 1: Set the data parameter No 000 to 2. Move the cursor over No 000 as described above, type 2 in the prompt line, as shown in the following figure



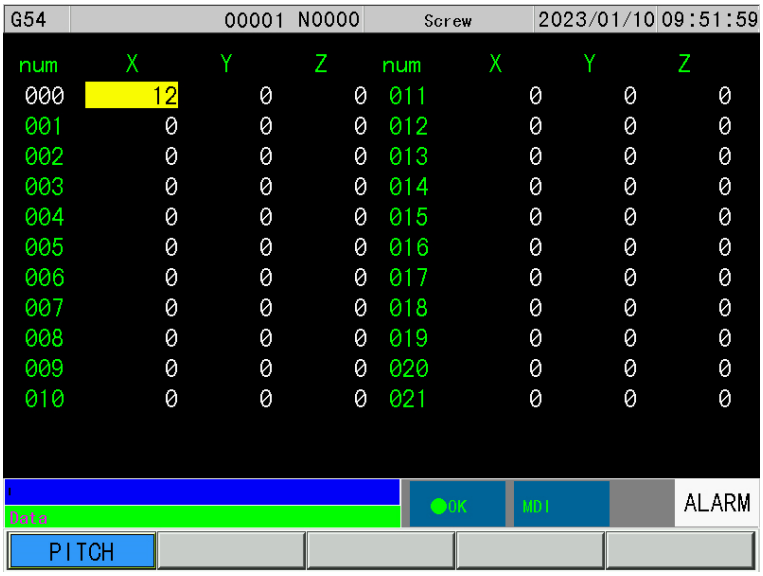
Press the Enter key to modify the parameters. The display page is as follows:



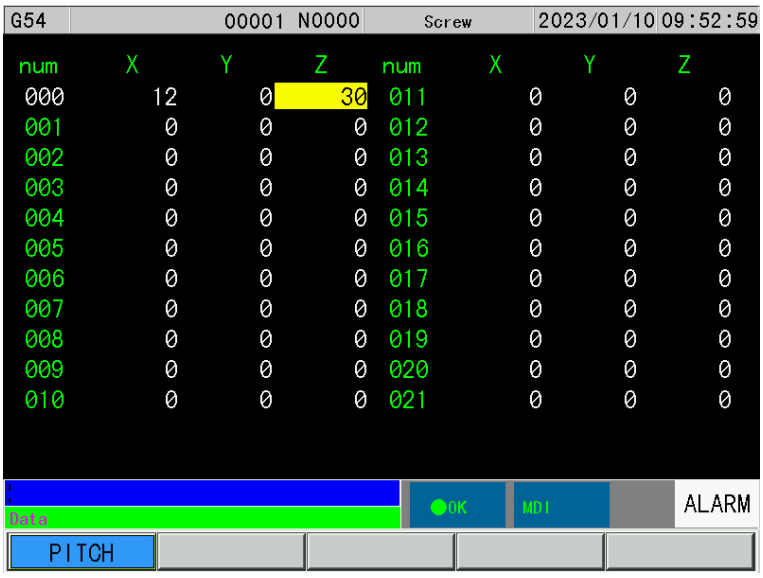
Example 2: Set the value of the X axis of the screw complement data No 000 to 12 and the value of the Z axis to 30. Follow the steps above to move the cursor over the screw data No 000

and type 12 in the prompt line.

Press the Enter key and the data modification is complete. The display page is as follows:



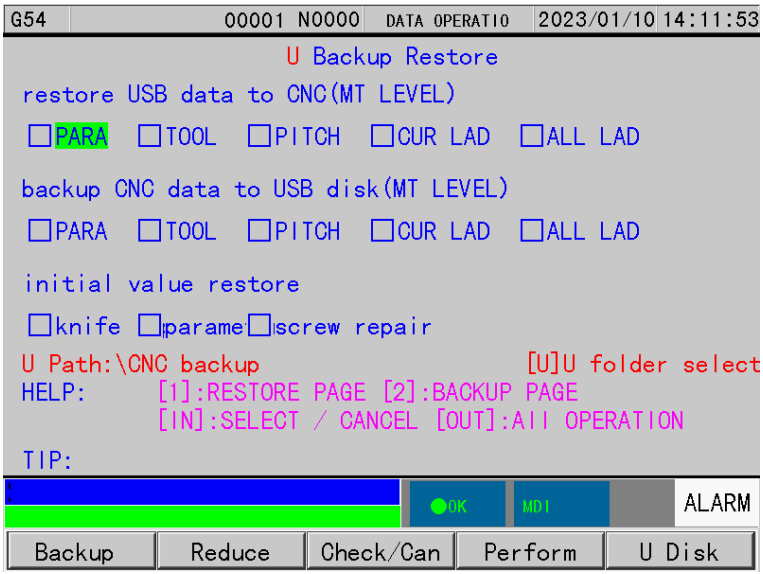
Similarly, type 30 in the prompt line, press the Enter key, and the data modification is complete. After modification, the page is displayed as follows:



9.2 Data Restoration and Backup

The user data of the system (such as status parameters, data parameters, screw data, etc.) can be backed up (saved) and restored (read). Data backup and restoration are performed without affecting the parts program stored in the CNC. The data action page is displayed as follows:

Press the screen down button  and  key to switch to find the F1 key [Settings], click to enter the setting page, and then click the F2 key [Backup] under the screen to enter the parameter backup and restore page, which has "U disk data restore to CNC", "CNC data backup to U disk", "initial restore operation" and other operation functions.



1、 Turn on the parameter switch;

2、 The system is inserted into the USB flash drive;

3、 Press the  and  key to switch between the 3 functions, and use the  ,  keys to move the cursor to the desired option;

4、 Press the F3 key [Check/Cancel] on the screen to select or deselect;

5、 Press the F4 key under the screen [Perform Operation] to start the backup or restore operation;

Note 1: Do not power off when backing up and restoring data, and it is recommended not to perform other operations until the operation is completed.

Note 2: Users **with level 3 operations and above can back up and restore status parameters, data parameters, and screw parameters.**

9.3 Setting and Modifying Permissions

In order to prevent the processing program and CNC parameters from being maliciously modified, the system has arched the permission setting function, and the password level is

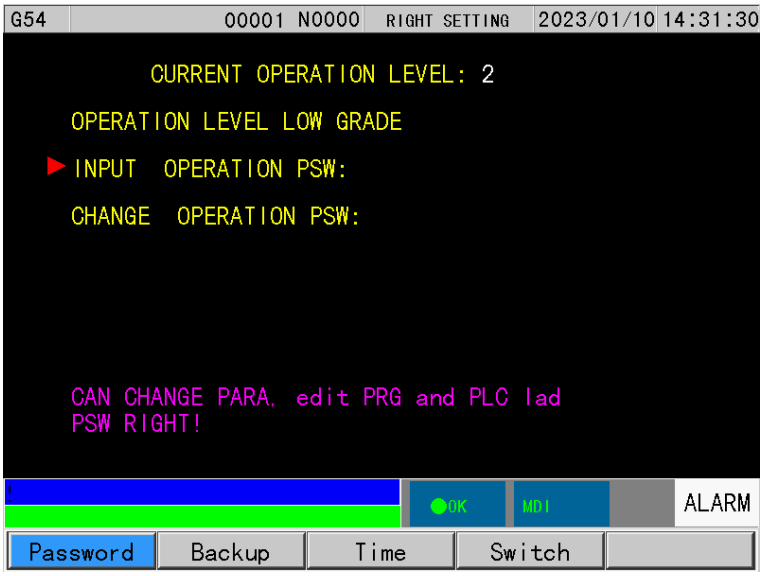
divided into 4 levels, from high to low, they are 2 (machine tool manufacturer level), 3 level (equipment management level), 4 Level (craftsman level), level 5 (machining operation level), the operation level of the CNC is currently displayed by the "current operation level:" on the permission setting page.

Level 2: Machine tool manufacturer level, allowing to modify CNC status parameters, data parameters, screw supplement data, tool supplement data, edit part programs, transfer PLC ladder diagram, etc.

Level 3: The initial password is 12345, which allows modification of CNC status parameters, data parameters, knife complement data, and editing part programs.

Level 4: The initial password is 1234, you can modify the tool supplement data (for tool setting operations), macro variables, edit the part program, and cannot modify the status parameters, data parameters and screw compensation data of CNC.

Level 5: No password level, can be operated by the machine tool operation panel, no tool supplement data can be modified, no part program can be selected, no program can be edited, and the status parameters, data parameters and screw supplement data of the CNC cannot be modified.



After entering the permission setting page, the cursor first stops on the line "Enter operation password". You can press the  key or  keys to move the cursor to the corresponding action.

1) Press the  key once to move the cursor up one line. Move the cursor up once and stop on the Action Level Demotion row (first line).

2) Press the  key once and the cursor moves down one line. Move down one line of the cursor to stop the "Change Operation Password" line (last line).

9.3.1 Entry to the Operation Level

1、 After entering the permission setting page, move the cursor to the "Enter operation password" line;

2、 Enter the operation password (for each number entered, the display adds a "*" sign);

3、 Enter and press the Enter key to enter the operation level corresponding to the password.

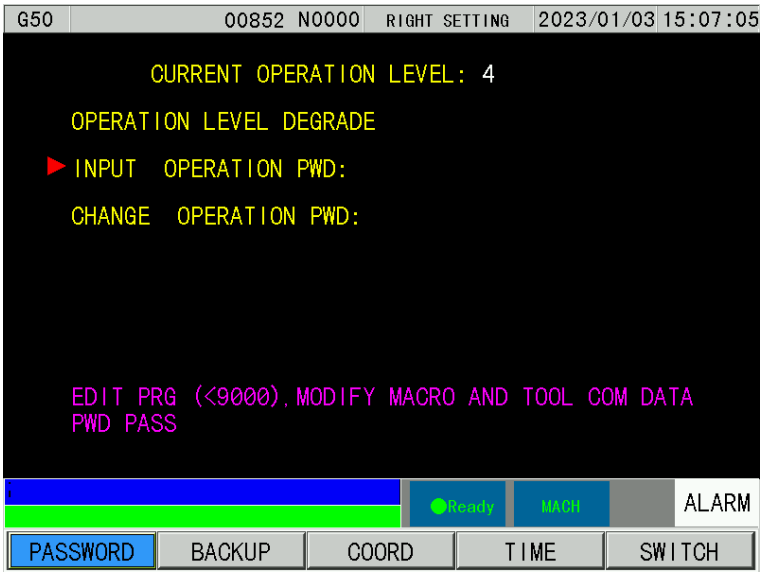
Note: The length of password data and operation level defined by the system are corresponding, and users cannot increase or decrease the length of password data according to their personal imagination.

Note: Be careful to use an all-numeric password when changing the operation password.

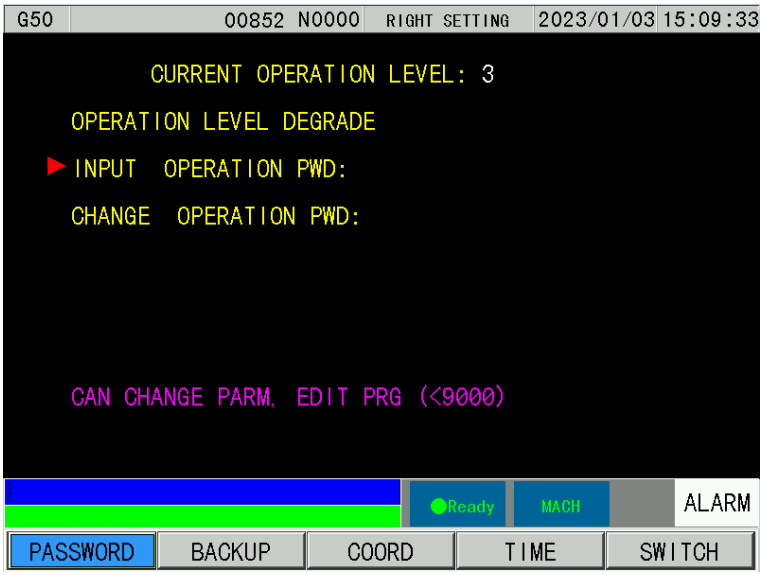
The details are as follows:

The level of	Password data	Initial
Level 3	5 digits	12345
Level 4	4 digits	1234
Level 5	not	not

Example: The current level of operation of the CNC is level 4, and the following page is displayed. The CNC3 level operation password is 12345, please change the current operation level to level 3.



Move the cursor to the "Enter operation password" line, enter 12345, press the Enter key, the CNC prompts "Parameters can be modified, programs can be edited", "Password passed", and the current operation level is changed to level 3 . The display page is as follows:



Note: If the current operation authority is less than or equal to **level** 3 (level **3,** level **4 or level 5), the operation authority will not change when it is powered on again. If the last** operation permission is level **2, the operation permission is set to level 3 by default when you power on again.**

9.3.2 Change of Operation Password

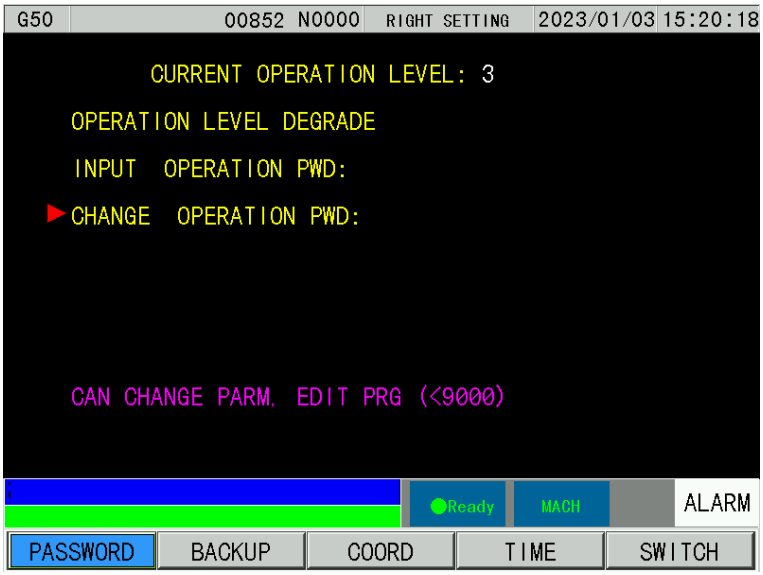
To change your password, follow these steps:

1、 After entering the permission setting page, enter the password as described in Section 9.3.1;

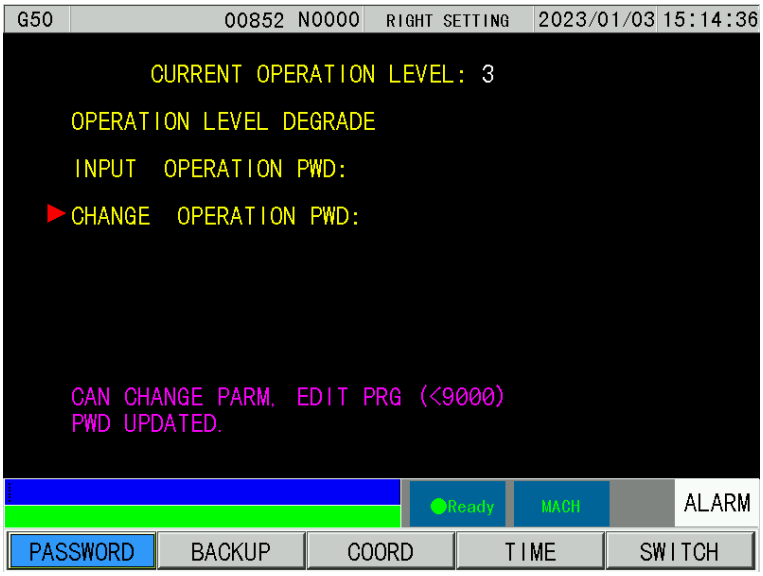
2、 Move the cursor to the line "Change operation password";

3、 Enter the new operation password and press the Enter key;

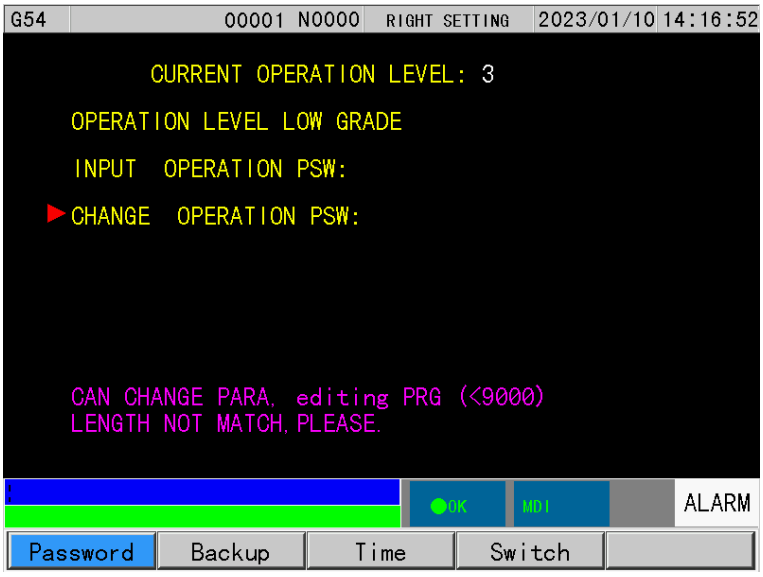
4、 The CNC prompts "Please enter the new password again" and the page is displayed as follows:



5、 After entering the operation password again, press the Enter key, if the password entered twice is the same, the CNC prompts "Password has been updated", and the operation password is changed successfully.



6、 If the two passwords entered are different, the CNC prompts "Change password verification does not match, please re-enter", and the display page is as follows:



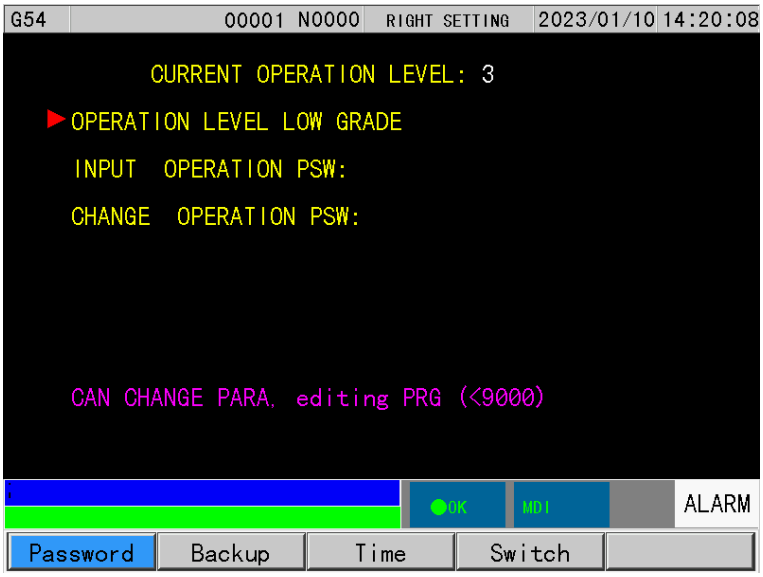
9.3.3 Downgrading the Operation Level

Action level downgrading makes it easy to downgrade from a higher level of action to a lower level of operation as follows:

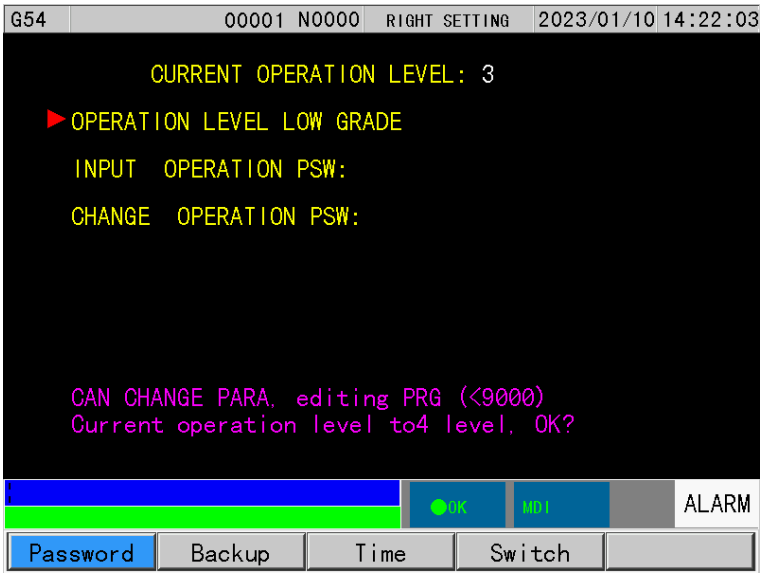
1、 After entering the permission setting page, enter the password as described in Section 9.3.1;

2、 Move the cursor to the Action Level Demotion row, and if the CNC is currently operating

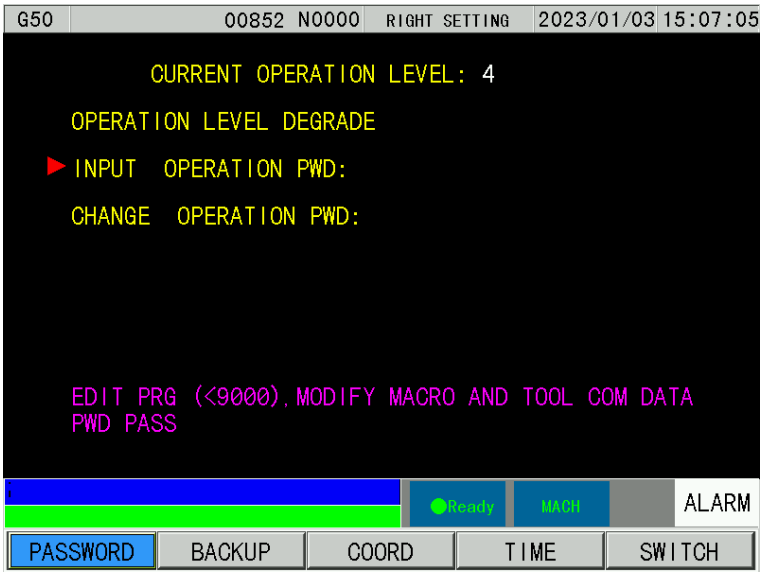
level 3, the page is displayed as follows:



3、 Press the Enter key, and the CNC prompts "The current operation level is reduced to level 4, confirm?" "； The display page is as follows:



4、 Press the Enter key again, the operation level is demoted successfully, and the page is displayed as follows:



Note: If the current operation permission is level 5 , you cannot downgrade the operation level.

Chapter 10 U disk operation function

10.1 File Directory Page

Press the 

key to enter the program page in the non-running state, press the F3 key [File Management] under the screen to enter the file directory interface, the upper part of the interface displays the number of programs, storage amount, used, remaining and other statuses, the left side of the interface displays the arrangement of each program, and the right side of the interface preview shows the program content where the cursor is located.

The corresponding soft function key functions under the file management interface are: F1 key [New], F2 key [Search], F3 key [Copy], F4 key [U disk directory], F5 key [Perform processing], F1 key [Rename] after page turning, F2 key [Delete], F3 key [Modify number].

Function description:

New: Click and enter a new file name in the dialog box to confirm the new program;

Search: Enter the program name in the dialog box after clicking, and then press Confirm to search for the program;

Copy: After clicking, enter a new file name in the dialog box and copy the program where the cursor is located;

U disk directory: click to enter the U disk directory interface;

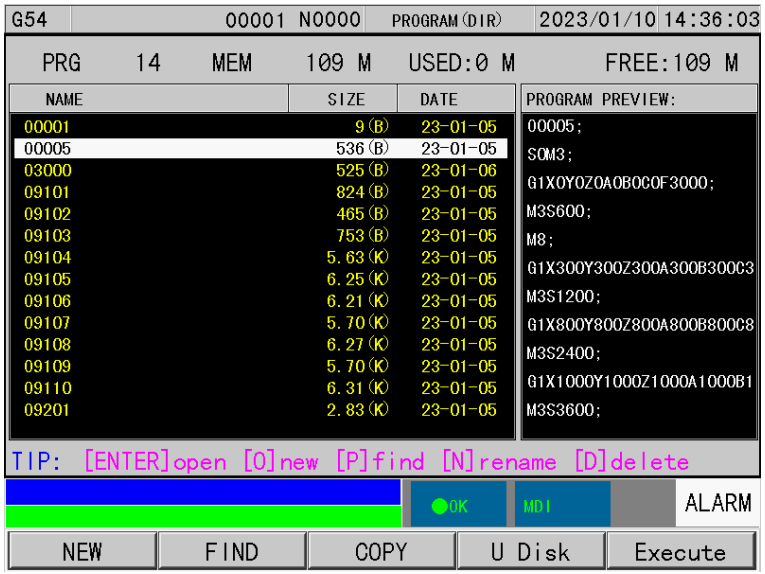
Perform processing: click to confirm the program where the cursor is located;

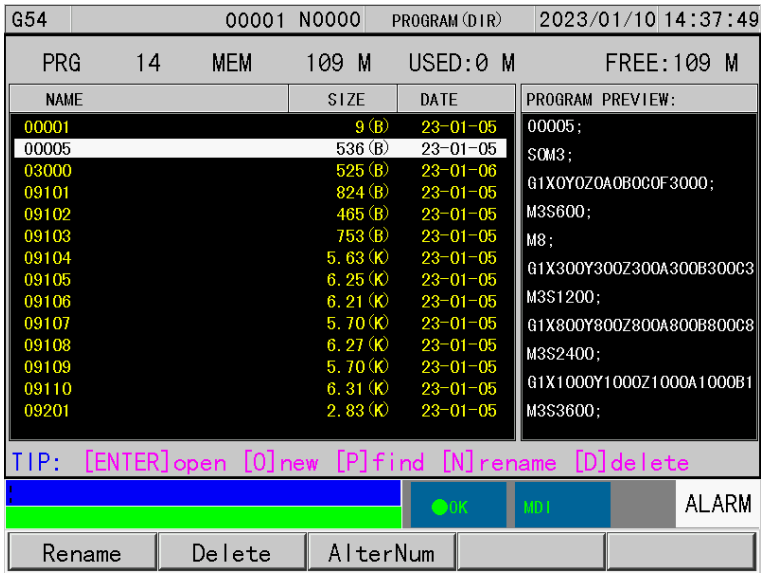
Rename: Click and enter a new file name in the rename dialog box to confirm the

modification of the program name;

Delete: Click to pop up the Delete File dialog box, and then click Confirm to delete the program;

Modification number: modify the numbering order of the program;





10.2 File Copying

Press the 

key to enter the program page in the non-running state, press the F3 key [File Management] under the screen to enter the file directory interface, and then press the F4 key [U disk directory] under the screen to enter the U disk directory interface, insert the U disk after inserting the U disk directory interface under the soft function key corresponding functions: F1 key [deposit CNC], F2 key [switch], F3 key [previous line], F4 key [next line], F5 key [Copy All],

as shown in the figure:

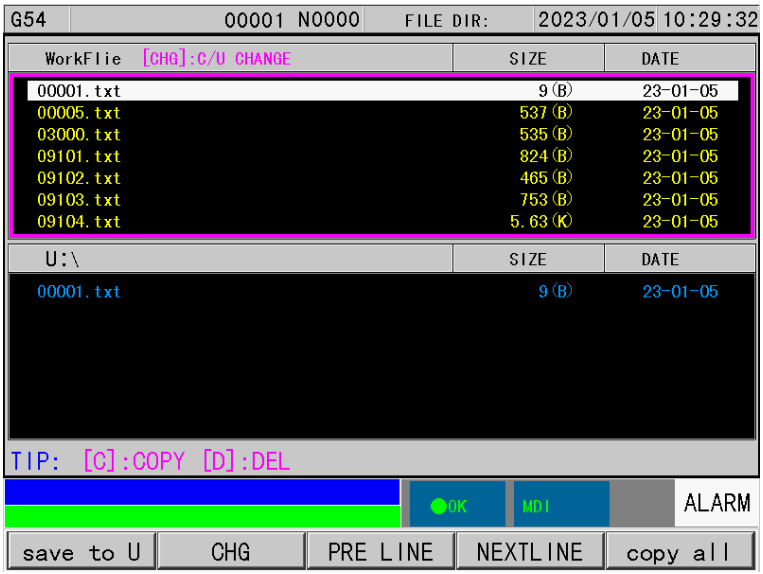
Deposit CNC: save the program where the cursor is located in the CNC drive;

Switching: Switch between CNC disk and U disk;

Previous line: Click the cursor to move up in the program arrangement;

Next line: Click the cursor to move down in the program arrangement;

Copy All: Copy all programs in the selected disk to another disk;

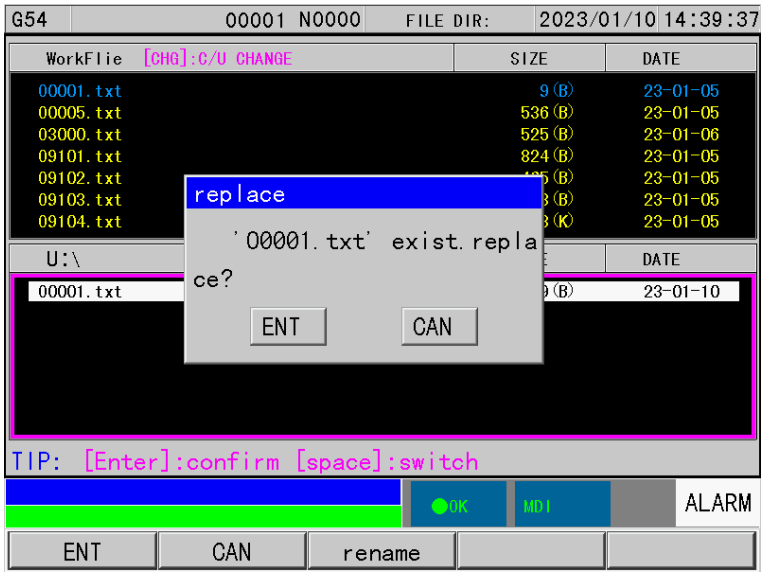


The top of the page displays the CNC disk directory information, the bottom shows the USB disk directory information, if the U disk is not detected, the lower display bar does not display the content, and displays "Please insert the U disk" in the middle display area.

Only the ". . CNC"、 ". NC" and ".txt" files.

Press the F2 key [Switch] purple drive letter under the screen to select the border will switch from the CNC disk to the USB disk, press the F3 key [Previous line] or F4 key [Next line] to move the cursor on each program name, and on the CNC disk and After the USB disk is selected, you can also press the F5 key [Copy All], then the system will copy all the programs of the current disk to another disk in turn.

Note 1: If the program name is the **same when copying the program, it will prompt "File ~~ already exists Whether to replace" through the F1 key [OK], F2 key [Cancel], F3 key [Rename] under the screen to select processing, as shown in the figure below, the U disk program is copied to the CNC disk.**



For example, copy the O0002 program in the CNC directory to the U disk directory, and operate as follows:

Move the cursor to the file to be copied, press the F1 key [Save to U Disk] under the screen to copy or enter the U disk interface and use the function key [Save CNC] to operate.