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

preface

Dear Customer:

For the products you choose, the company is deeply honored and grateful!

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

Security warnings

 Improper operation will cause accidents and must be qualified to operate the system.

Special note: The system power supply installed on the chassis (inside) is a special power supply provided only for the CNC system manufactured by the 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 product packaging box stacking shall not exceed six layers;
2. Do not climb, stand or place heavy objects on the product box;
3. Do not use the cable connected to the product to drag or handle the product;
4. It is strictly forbidden to collide and scratch the panel and display;
5. Product packaging should be protected from moisture, exposure to sunlight and rain.

■ Check out of the box

1. After opening the package, please confirm whether it is the product you have purchased;
2. Check whether the product is damaged in transit;
3. Check whether the parts are complete and whether there is damage against the list;
4. If there is a product model inconsistency, missing accessories or shipping 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 (zero wire) cannot be used instead of the ground wire;
3. The wiring must be correct and firm, so as not to lead to product failure or unexpected consequences;
4. The surge absorption diode connected to the product must be connected in the specified direction, otherwise the product will be damaged;
5. The power supply to the product must be cut off before plugging or opening the product chassis.

■ Overhaul

1. The power supply must be cut off before overhauling or replacing components;
2. When a short circuit or overload occurs, the fault should be checked, and the troubleshooting can be restarted;
3. Do not frequently power off the product, if you must re-power after the power outage, the interval time is at least 1min.

statement

The manual describes as many different things as possible, but because of the possibilities involved, it is not possible to explain all the actions that can or cannot be done, so what is not specifically stated in this manual can be considered unusable.

warn

Before the installation connection, programming and operation of this product, it is necessary to read the product manual and the instruction manual of the machine tool manufacturing plant in detail, and strictly follow the requirements of the manual and the instructions for related operations, otherwise it may lead to damage to the product, the machine tool, the scrap of the workpiece and even personal injury.

The product functions and technical indicators described in this manual (such as accuracy, speed, etc.) are only for this product, the CNC machine tool of this product is installed, 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 instructions of the machine tool manufacturer; Although the system is equipped with a standard machine tool operation panel, the function of each button of the standard machine tool panel is defined by the PLC program (ladder diagram). Please note that the function of the keys in the machine panel of this manual is described for the standard PLC program!

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

The safety responsibility of the manufacturer

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

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

- 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 dangers caused by adding, changing or modifying the original CNC system and accessories.

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

*** This manual is for the end-user collection. Sincerely thank you for your friendly support to our company when our products!**

Part II

Operating instructions

directory

Chapter 1 Operation Methods and Display Interfaces

1.1 Panel division

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

Part 1: display menu, function menu keys (position, program, compensation and other buttons), such keys are handled by the system software, not controlled by PLC, for switching different interfaces;

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

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

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

Panel division

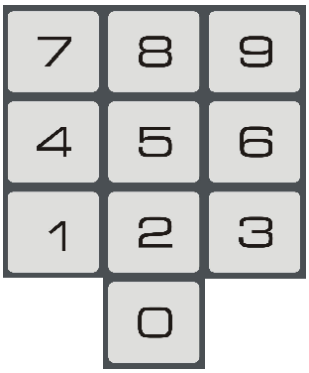
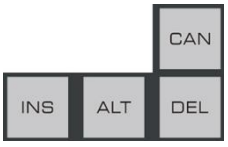
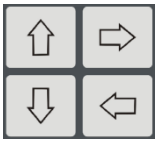
Note: Some machining center system models have different button arrangement positions.

1.1.1 Status indication

	Axis back to zero end LED
	Tri-color lights

1.1.2 Edit the keyboard

keystroke	name	Description of the feature
	Reset key	CNC reset, feed, output stop, etc
	The address key	Address input
		Double address key, press the key repeatedly, switch between the two

	Symbol key	Three address keys, press the key repeatedly, switch between the three
	Number keys	Numeric input
	Enter the key	Determination of data inputs such as parameters and compensation amounts
	Output key	Start the communication output
	Convert keys	Switch between information and display
	Edit key	Insert, modify, delete, etc. of programs, fields, etc. when editing
	EBO key	The input of the program segment terminator
	Cursor move the key	Controls cursor movement
	Page turn key	Switching of pages under the same display interface
	Soft function keys	After using the function key to switch the page set, you can use the corresponding software function key to display the content of a subpage in the current page, the schematic diagram is as follows:  Action/page toggle key Back to the Upper Menu Move the menu key right What the soft function keys do: (1) Switch subpages within the current page set;

		(2) As an action input for the currently displayed subpage, such as editing, modifying data, or displaying content.
The following button parts of the model have		
	Enter	Determination of data inputs such as parameters and compensation amounts; Input of program segment terminators; Note: Model systems with a Enter key do not have an input key, and the Enter key is used as an input and line feed key
	The first line lasts the key	In the program interface you can switch to the beginning and end of the program
	Up button	Some model numbers and symbols are composite keys, and the symbols can be switched after pressing the upper button

1.1.3 Display menus

Menu key	remark
	Enter the location interface. The location interface has four surfaces: relative coordinates, absolute coordinates, comprehensive coordinates, and process supervision
	Enter the program interface. The program interface has four pages: program content, program status, program directory, and file directory
	Enter the compensation boundary surface and macro variable boundary surface, and repeatedly press the key to switch between the three interfaces. The compensation interface can display tool offset wear; The Macro Variables interface displays CNC macro variables
	Enter the alarm interface, alarm log, and repeatedly press the key to convert between the two interfaces. The alarm interface has two pages: CNC alarm and PLC alarm; The alarm log shows the history of generated and eliminated alarms
	Enter the settings interface, graphical interface, and repeatedly press the key to convert between the two interfaces. The setting interface has switch settings, G54-G59, data operation, permission settings, and time settings; The graphical interface displays the trajectory of the feed axis
	Enter the interface of status parameters, data parameters, and screw parameters. Repeated keystrokes can be converted between interfaces

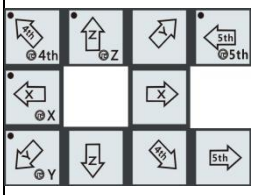
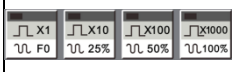
DGN	Enter the CNC diagnostic interface, PLC status, PLC data, machine tool soft panel, version information interface. Repeated keystrokes can be converted between interfaces. CNC diagnostic interface, PLC status,PLC data show CNC internal signal status PLC everywhere Information on the status of the address and data; Machine tool soft panel can carry out machine tool soft keyboard operation; The version information interface displays the version number of the CNC software, hardware, and PLC
PLC	Enter the ladder bounding surface, PLC information, PLC ladder diagram, PLC parameters, PLC diagnostic bounding surface, Repeated key presses can be converted between polygons.
GRA	Enter the graphics bounding surface.

1.1.4 Machine panels

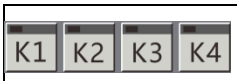
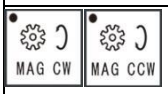
The function of the keys in the milling machine tool panel is defined by the PLC program (ladder diagram), and the specific functional significance of each button can be found in the manual of the machine tool manufacturer.

The function of each button in the machine panel defined by the standard PLC program of the milling machine series is shown in the following table:

keystroke	name	Description of the	How the function works when it
	Feed hold key	The program and MDI code are suspended	Automatic mode, input mode, DNC mode
	Cyclic start key	The program and MDI code run start	Automatic mode, input mode, DNC mode
	Feed multiplier key	Adjustment of feed speed	Automatic mode, input mode, editing mode, machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Fast multiplier key	Adjustment of fast movement speed	Automatic mode, input mode machine tool back to zero, manual mode, DNC mode
	Spindle magnification key	Spindle speed adjustment (spindle speed simulation control mode is effective)	Automatic mode, input mode, editing mode, machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode

	Manual tool change key	Manual tool change	Machine tool back to zero, handwheel mode, single step mode, manual mode
	Click to open the key	Spindle jog state on/off	Machine tool back to zero, handwheel mode, single step mode, manual mode
	Lubrication opens the key	Machine lubrication on/off	Automatic mode, input mode, editing mode, machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Coolant open key	Coolant on/off	Automatic mode, input mode, editing mode, machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Blow open key	Processing blows on/off	Automatic mode, input mode, editing mode, machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Spindle control keys	The spindle is spinning The spindle stops The spindle is reversed	Automatic mode, input mode, editing mode, machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Quick switch	Fast speed/feed speed switching	Automatic mode, input mode, manual mode, DNC mode
	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 each axis of positive / negative movement	Machine back to zero, single step, manual mode
	Handwheel/single-step incremental selection with fast multiplier selection keys	Handwheel moves per grid 1/10/100/1000 * Minimum equivalent single step per step move 1/10/100/1000 * Minimum equivalent fast rate Fo, 25%, F50%, F100%	Automatic mode, input mode, machine tool return to zero, handwheel mode, single step mode, manual mode, DNC mode
	Select Stop	When Stop is active, perform M01 Pause	Automatic mode, input mode, DNC mode

	Single-segment switch	The single-segment running/continuous running status of the program is switched, and the single-segment operation indicator is on when the single-segment is active	Automatic mode, input mode, DNC mode
	Program section select jump switch	Whether the program segment marked with a "/" sign at the beginning of the program segment skips the state switch, and the skip section indicator is lit when the program segment jump switch is turned on	Automatic mode, input mode, DNC mode
	The machine locks the switch	When the machine is locked, the machine lock indicator is on, and the feed shaft output is invalid	Automatic mode, input mode, editing mode, Machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Accessibility function locks the switch	When the auxiliary function is locked, the auxiliary function lock indicator light is on, and the M, S, T function output is invalid	Automatic mode, input mode, DNC mode
	Spindle positioning switch	Turn spindle positioning on or off	Automatic mode, input mode, editing mode, Machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Working light switch	Turn the work light on or off	Automatic mode, input mode, editing mode, Machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Chip removal machine switch	Turn the chip evacuator on or off	Automatic mode, input mode, editing mode, Machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode

	Custom output switches	Turn custom output on or off	Automatic mode, input mode, editing mode, Machine tool back to zero, handwheel mode, single step mode, manual mode, DNC mode
	Handwheel simulation development	Turn the handwheel simulation function on or off	Automatic mode, input mode, DNC mode
	Edit by select key	Enter the editing mode	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	Select the key in an automatic mode	Enter the automatic operation mode	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	Enter by select key	Enter the Entry (MDI) mode of operation	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	Machine zero mode selection key	Enter the machine zero operation mode	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	Single-step/hand wheel selection key	Enter the single-step or handwheel operation mode (one of the two operating modes is selected by the parameter)	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	Select the key manually	Enter the manual operation mode	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	The DNC mode selects the key	Enter the DNC operation mode	Automatic mode, input mode, machine tool back to zero, Handwheel mode, single step mode, manual mode, DNC mode
	Power-off memory key	Turn on the power-off memory function	Automatic mode, input mode, DNC mode
	Magazine back to zero	Turn on the tool magazine back to zero function	The machine returns to zero
	Tool magazine rotation key	Control the forward rotation and reversal of the tool magazine	Handwheel mode, single step mode, manual mode

	Magazine forward and back keys	Control the front and back of the magazine or the inverted and returned tool of the magazine	Handwheel mode, single step mode, manual mode
	Tool magazine arm key	Controls the rotary arm rotation of the disc magazine	Handwheel mode, single step mode, manual mode

1.2 Overview of the mode of operation

The milling machine series has seven operating modes, such as editing, automatic, input, machine tool back to zero, single step / handwheel, manual, DNC mode and so on.

- **Edit the operation mode**

Under the editing operation mode, operations such as the establishment, deletion and modification of the machining program can be carried out.

- **Automatic mode**

In Auto mode, run the program automatically.

- **Enter the operation mode**

Under the input mode, the input of parameters and the input and execution of code snippets can be performed.

- **Machine zero operation mode**

Under the operation mode of machine tool back to zero, the feed shaft back to the zero point operation of the machine tool can be performed separately.

- **Handwheel / single-step operation**

In the single-step/handwheel feed mode, the CNC moves in selected increments.

- **Manual operation mode**

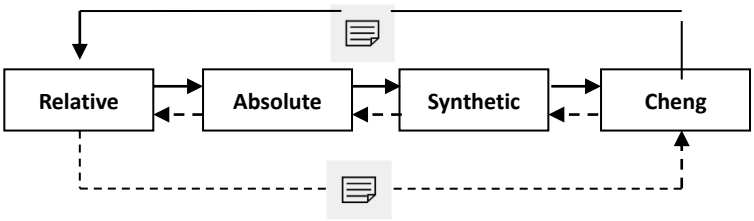
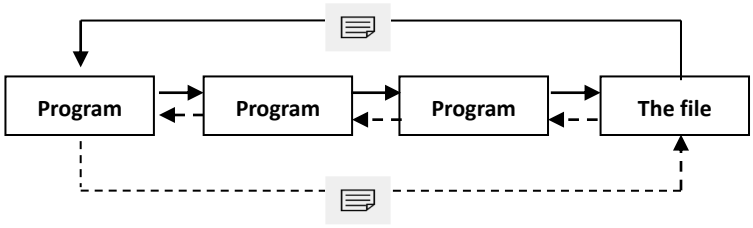
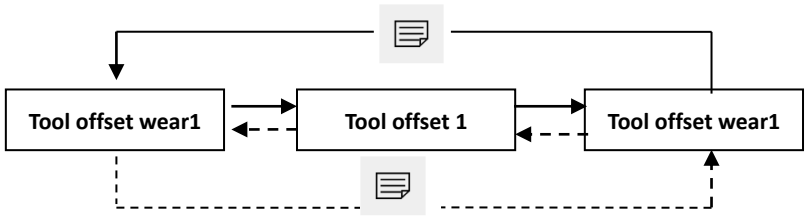
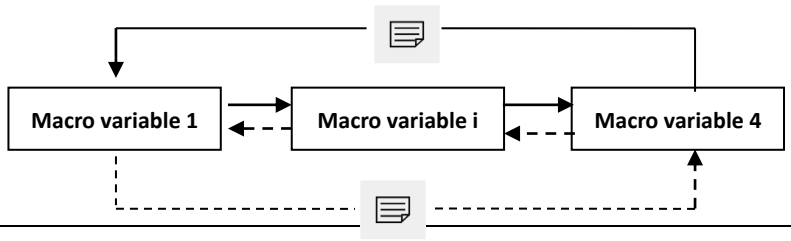
Under the manual operation mode, manual feed, manual fast, feed rate adjustment, fast rate adjustment and spindle start and stop, coolant switch, lubricating liquid switch, spindle jogging, manual loosening knife / clamping knife and other operations can be carried out.

- **DNC operation mode**

Under DNC operation mode, the DNC runs the program.

1.3 Display interface

The machining center series has 9 interfaces, such as 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 following figure

Menu key	Displays the interface	Displays the page
POS	Location interface	
PRG	Program interface	
OFT	Compensation interface	
	Macro variables interface	
ALM	Alarm interface	Alarm
	Alarm log interface	Alarm logs
SET	Settings interface	The pages are: switch settings, G54-G59, data operations, permission settings, and time settings
	password	Operation-level interface
PAR	Bit parameters	Bit parameter interface
	Number parameters	Number parameter interface
	Snail patching	Screw patch interface
DGN	CNC diagnosis	CNC diagnostic interface

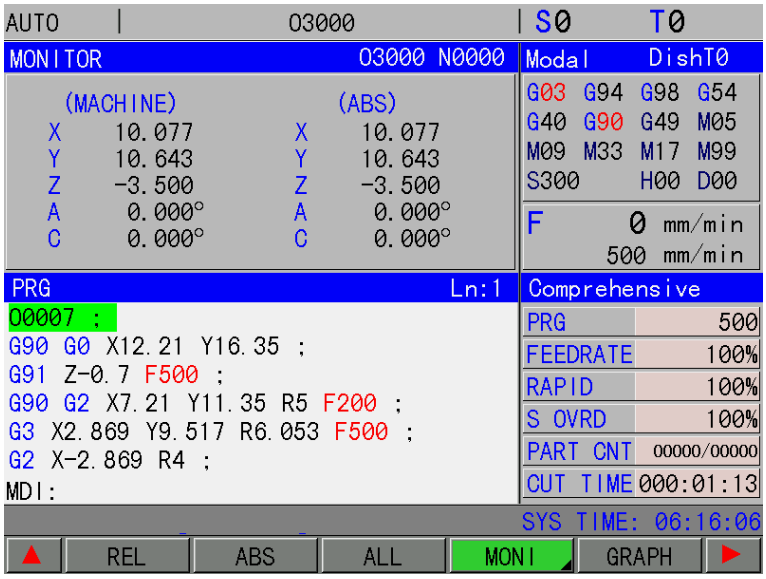
	PLC diagnosis	PLC signal diagnostic interface
	I-O diagnostics	Input/output diagnostic interface
	Version information	The system software version is displayed
	alarm	Switch to the alarm interface
	Spindle inspection	Detection spindle reaction time interface
	absolute value	The absolute value drives the monitoring interface
	Zero point detection	A turn signal back to zero monitoring interface
	CNC helps	Each interface is: operation table, alarm table, G code table, M code table, macro command
PLC	PLC information	PLC information
	Ladder diagram	Ladder diagram
	PLC parameters	PLC parameters: KEEP, TMR, DATA, CTOR
	PLC signal	
	Tool magazine control	
	Extended control	
GRA	Graphical interface	

1.3.1 Location interface

Press the POS key to enter the location interface, the location interface has absolute coordinates, relative coordinates, comprehensive coordinates and process supervision four pages, can be viewed by the POS key or soft function keys [F1-F5] key.

1) Absolute coordinates display the page

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



Note: In editing, automatic, entering, displaying "programming speed"; The "Manual Speed" is displayed in the machine to zero, the program to zero, and the manual mode. Display "Handwheel Increment" in handwheel mode; Displays Step Increments in single-step mode.

Actual speed: the actual processing speed after the feed rate operation in actual processing;
Feed magnification: the magnification selected by the feed magnification switch;
G function code: The value of the G code in the currently executing program segment;
The number of processed parts: when the program executes M30 (or M99 in the main program), the number of processed parts is increased by 1;
Cutting time: when the automatic operation starts after the start of the time, the time unit is hours, minutes, seconds;
Fast magnification: Displays the current fast magnification;
Spindle magnification: When the Bit4 bit of the parameter NO.001 is set to 0, the spindle magnification is displayed;
S0000: spindle speed feedback from the spindle encoder, the spindle encoder must be installed to display the actual speed of the spindle;
Current tool: The tool number specified by the T code in the program.
Tool offset: H00 tool length compensation for current machining programs; D00 Radius compensation for the current machining program.
The number of processed parts and the cutting time are remembered for power-down, and the zeroing method is as follows:

The number of processed parts is clear: first press and hold the CAN key, then press the N# key; Or directly press [] on the location interface to move the menu, find the [▸ number

of pieces clear] button, press clear zero;

Cutting time clearance: first press and hold the CAN key, then press the M[key; Or directly press [] on the location interface to move the menu, find the [▸ working hours clear] button, press clear zero;

2) Relative coordinates display the page

The relative coordinate values displayed are the coordinates of the current location relative to the relative reference point, and the relative coordinates are maintained when the CNC is powered on. Relative coordinates can be cleared to zero at any time. When relative coordinates are cleared, the current point is the relative reference point.

Methods for zeroing relative coordinates:

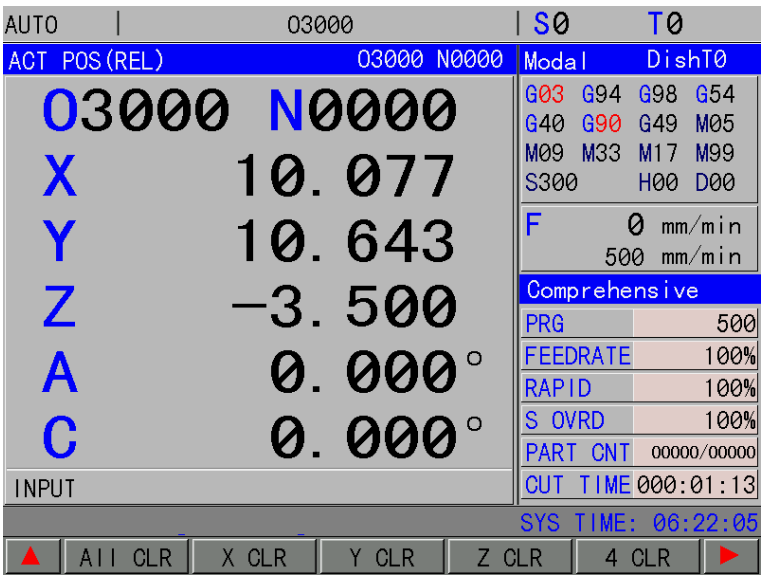
Under the relative coordinate display page, hold down the X key until X flashes in the page, press the cancel key, and the X coordinate value is cleared;

Under the relative coordinate display page, hold down the Y & key until Y flashes in the page, press the cancel key, and the Y coordinate value clears zero.

Under the relative coordinate display page, hold down the Z key until Z flashes in the page, press the cancel key, and the Z coordinate value is cleared.

Or directly press the [Relative] key on the relative coordinate interface and enter the lower menu to clear the relative coordinates.

Note: If 4th and 5th are valid, the zeroing method is the same as above.

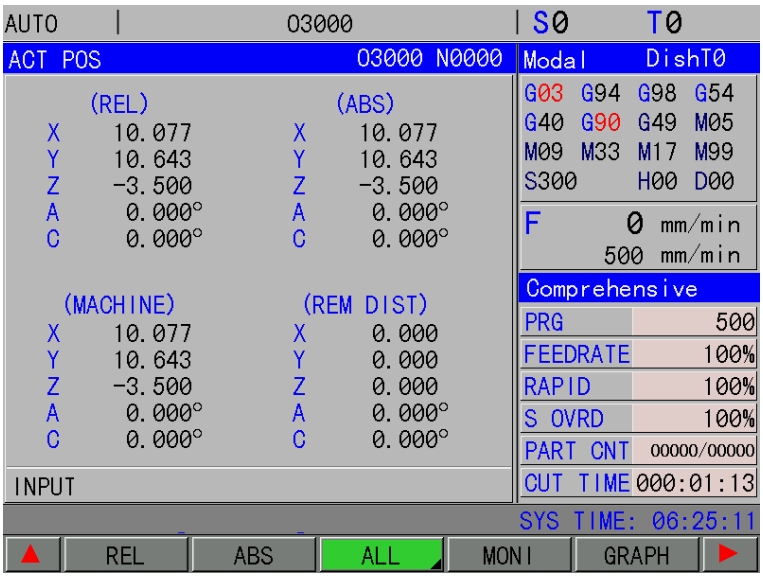


3) Comprehensive coordinate display page

On the General Location page, the relative coordinates, absolute coordinates, machine coordinates, and residual movement are displayed at the same time (the residual movement is only displayed in the automatic and input mode).

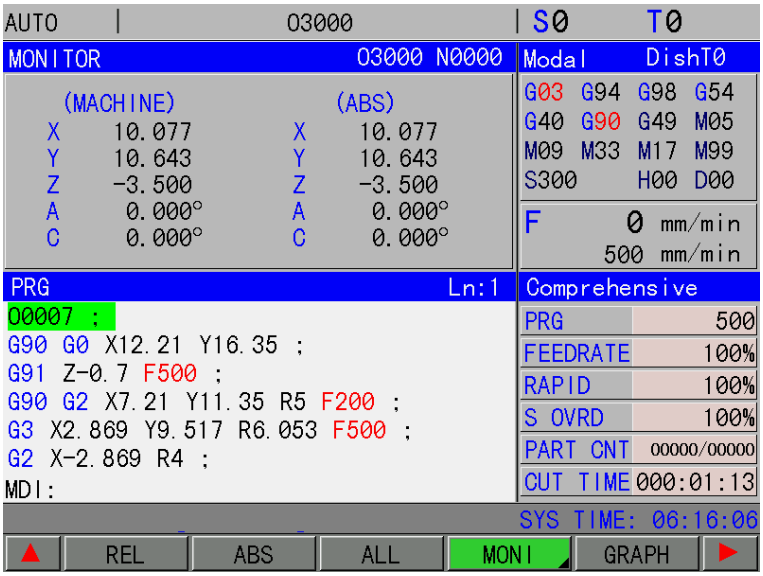
The display value of the machine coordinates is the coordinate value of the current position in the machine tool coordinate system, which is established by returning to the zero point of the machine tool. The remainder is the difference between the target position of the program segment or MDI code and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

The display page is as follows:



4) The Cheng supervisor display page

In the process supervision display page, the absolute coordinates of the current position, the relative coordinates (if the Bit0 bit of the status parameter No.25 is set to 1, the absolute coordinates of the current position and the amount of residual movement are displayed) and the 6 program segments of the current program are displayed at the same time. The cursor is located at the currently running program segment.

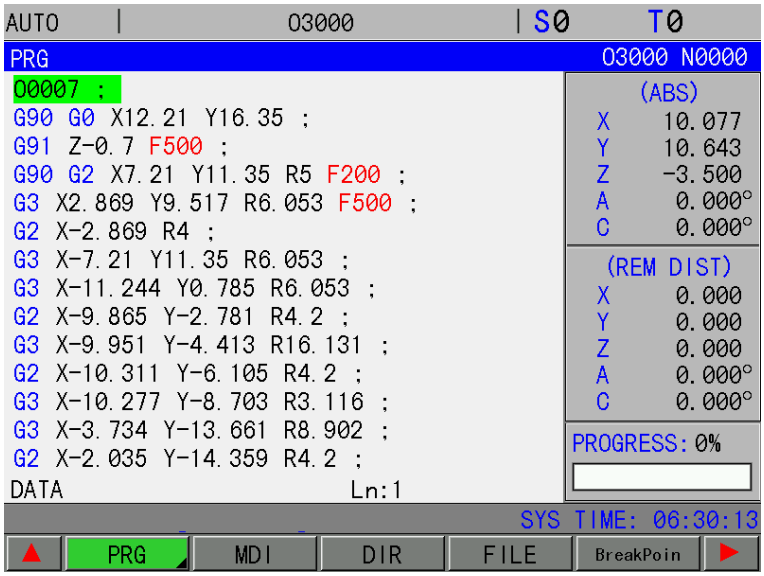


1.3.2 Program interface

Press the PRG key to enter the program interface, the program interface has four pages of program content, program status, program directory, and file directory, and switch between each page by repeatedly pressing the PRG key.

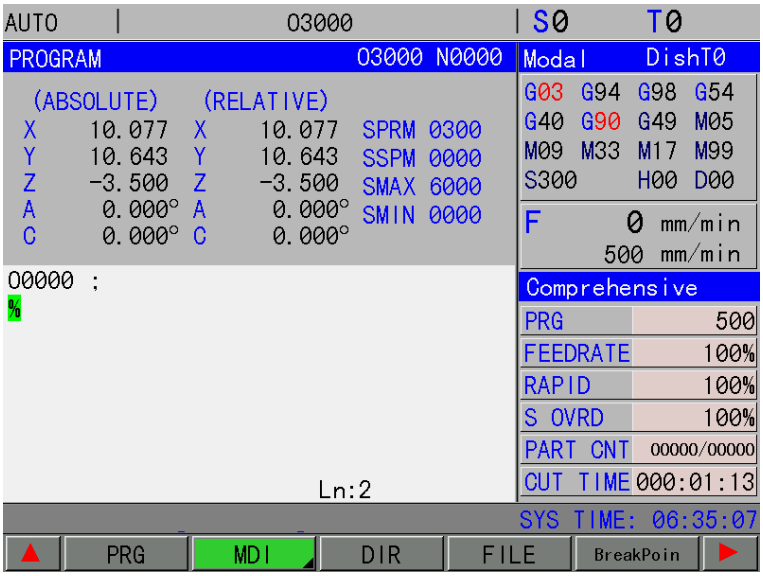
1) Program content page

On the Program Contents page, the program content, including the current program segment, is displayed. Under Edit Mode, press the key, the key forward, or backward to view the program contents.



2) Program Status page

When you are on the program content page, pressing the PRG key will take you to the program status page



3) Program catalog page

When on the program status page, pressing the PRG key will take you to the program directory page. Under this interface, all the machining programs are listed, and in order to make it easier for the user to find the program they want to choose, the first 12 lines of the current program are displayed on the right side of the page.

What the program catalog page displays:

(a) Number of programs in existence:

Displays the number of programs that have been deposited in the CNC, including subroutines

(b) Number of remaining programs:

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

(c) Amount of storage used:

Displays the amount of storage (KB) consumed by the CNC stored part program

(d) Amount of remaining storage:

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

(e) Catalogue of Programs:

The program numbers stored in part programs are displayed in order of the size of the part program name

(f) Program size:

Displays the amount of storage space occupied by the CNC program

4) File directory page

When you are on the program directory page, pressing the PRG key will take you to the file directory page. The page appears as follows:

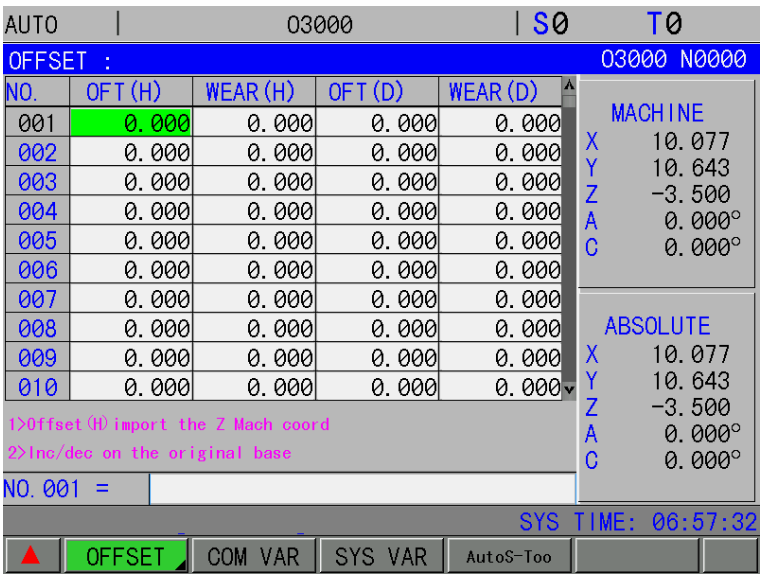
1.3.3 Tool bias wear, macro variable interface

The OFT key is a composite key, press the OFT key once from other display pages to enter the knife patch interface, and then press the OFT key to enter the macro variable interface.

1) The tool is offset against the wear interface

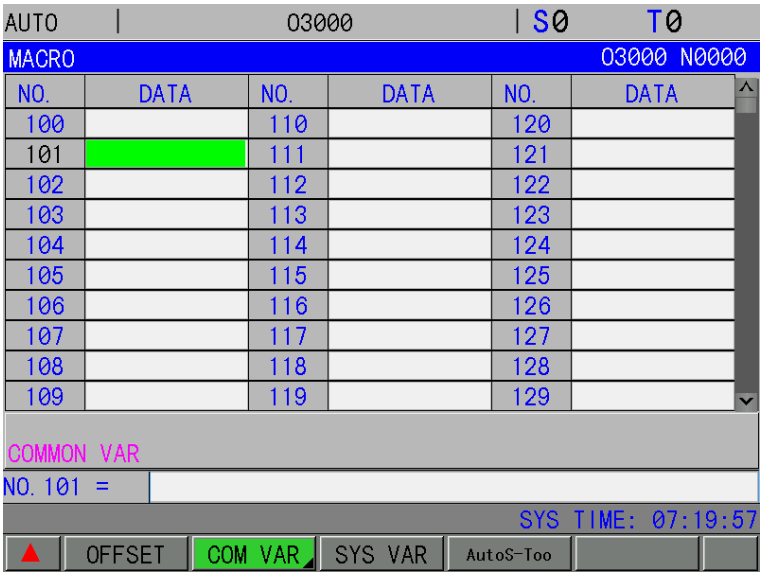
The tool offset wear interface has a total of 26 pages, a total of 256 bias, wear number

(No.001 ~ No 256) For the use of users, display each page by key, key, display page as shown in the figure:



2) Common variables interface

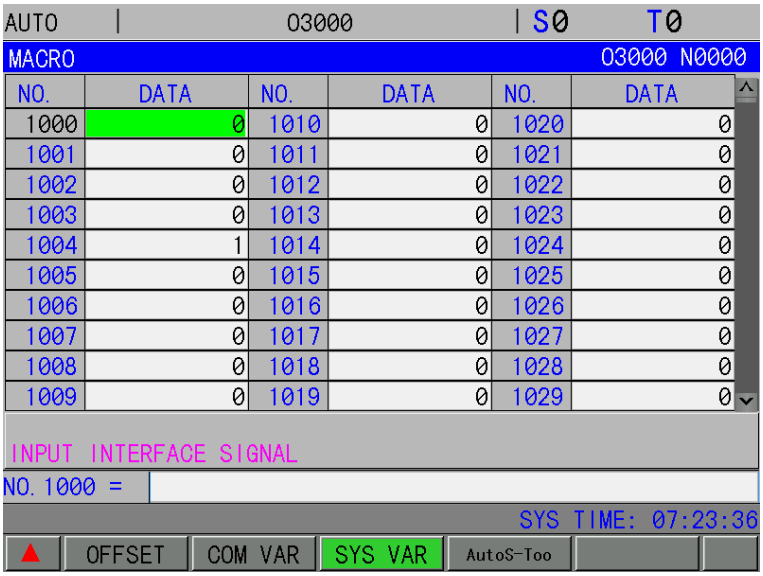
The macro variable interface has 20 pages, which can be displayed by key and key, and the macro variable page displays a total of 600 (No.100~ No.199 and No.500 ~ No.999) macro variables, macro variable values can be specified by macro code or set directly on the keyboard.



3) System variables interface

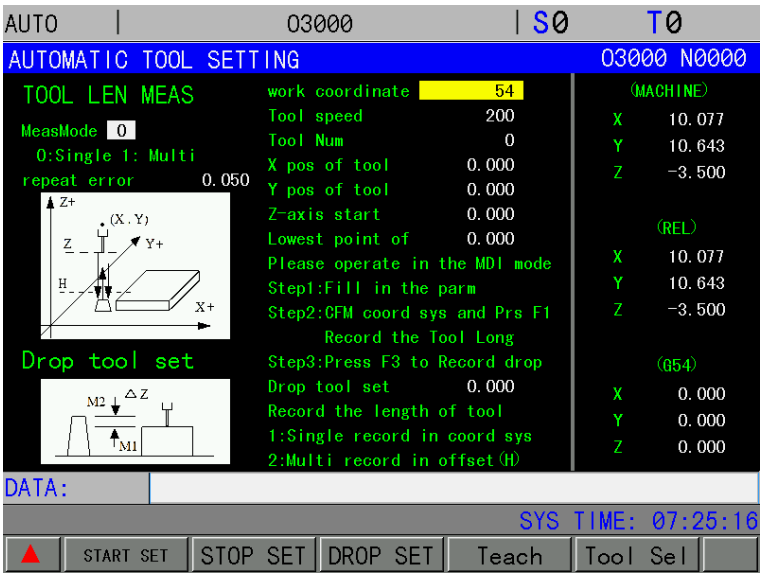
The macro variable interface has 49 pages, each page can be displayed by key and

key, and the system macro variable page is displayed (No.1000 ~ No.). 725) System macro variables, some system macro variables can only be read and not written, and some can be read and written, and cannot be entered manually.



4) Automatic tool setting interface

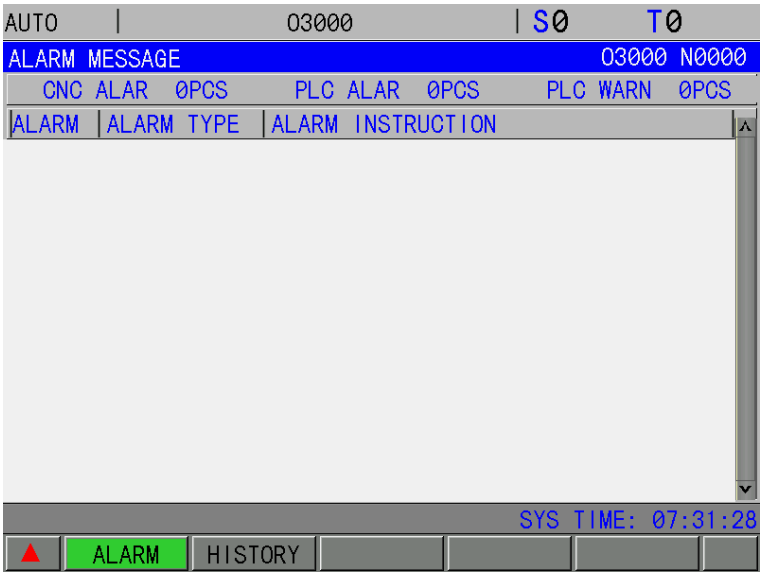
Enter the automatic tool setting interface optional parameters to achieve automatic tool setting Z-axis tool alignment.



1.3.4 Alarm interface

1) Alarm:

Press the ALM key to enter the alarm interface, through the  key,  key to view all the alarm display, the page is displayed as follows:



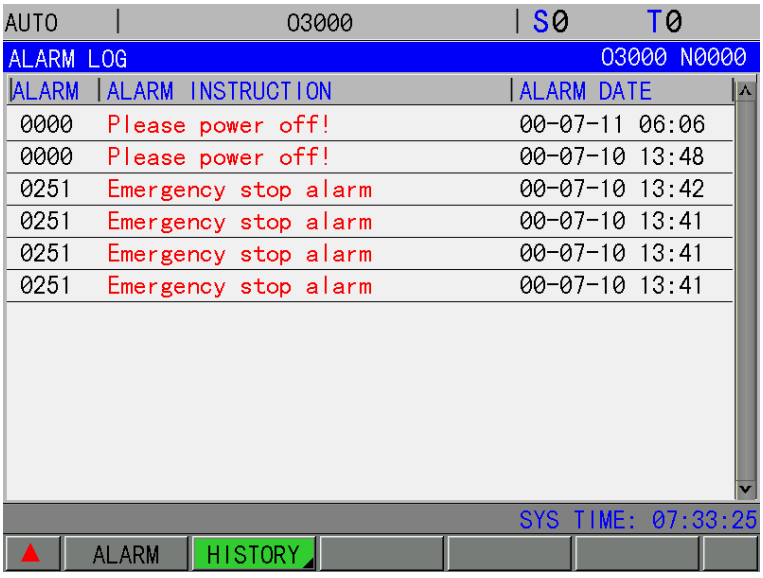
Note: Clear the alarm: When the alarm has been canceled, the RESET **key can clear the alarm content.**

(Alarm number 100 must be cleared by pressing both RESET **keys and** CAN **keys).**

2) Alarm log:

Then press the ALM key to enter the alarm log interface. Through the key, the key can view a total of 200 alarm log information;

Sort in order: The latest alarm log information is listed at the front of the first page, and then pushed in turn. When the alarm log exceeds 200, the last historical log message is cleared.



Note: Manual clearance of alarm logs: Press the CAN **+** J B **key under the level 2** password **or**

directly press the function key [Clear] to clear all log information.

1.3.5 Setup interface

Pressing the SET key once from other pages to enter the settings interface

1. Settings interface

The setup interface has six pages, which can be viewed through the SET key and the [Soft Function] key.

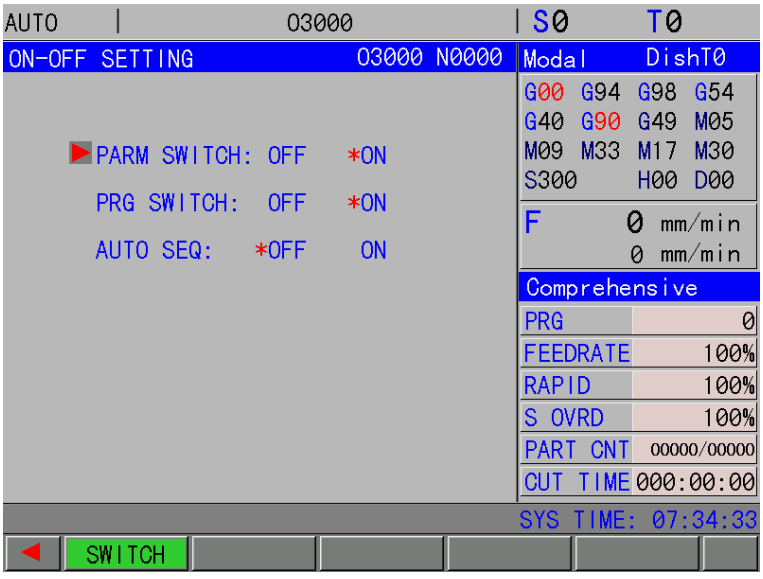
1) Switch interface

Switch settings: Displays the on-off and off-status of parameters, programs, and auto-serial numbers.

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

Program switch: When the program switch is turned on, the program can be edited; When closed, editing programs is prohibited.

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



2) Data operation: On this page, you can back up and restore CNC data (status parameters, data parameters, screw parameters, tool bias, etc.).

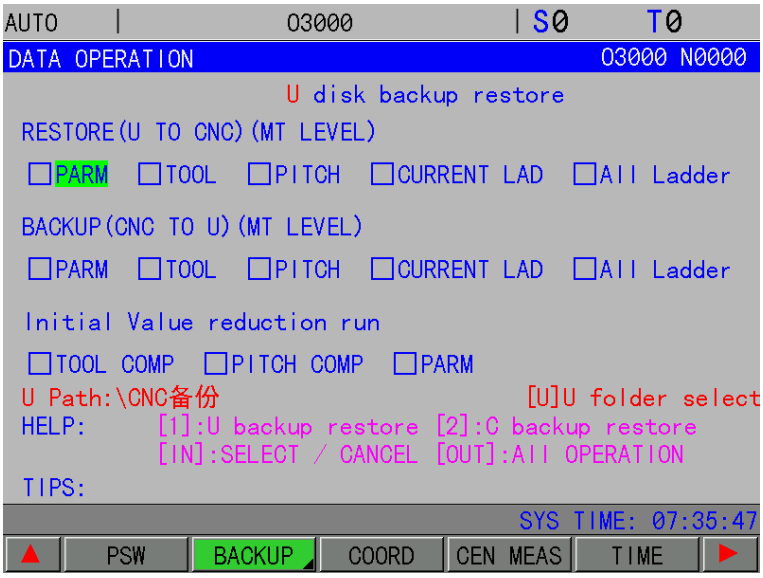
Initial value restore operation: Parameters, knife patches and screw patches can be restored to system default values.

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

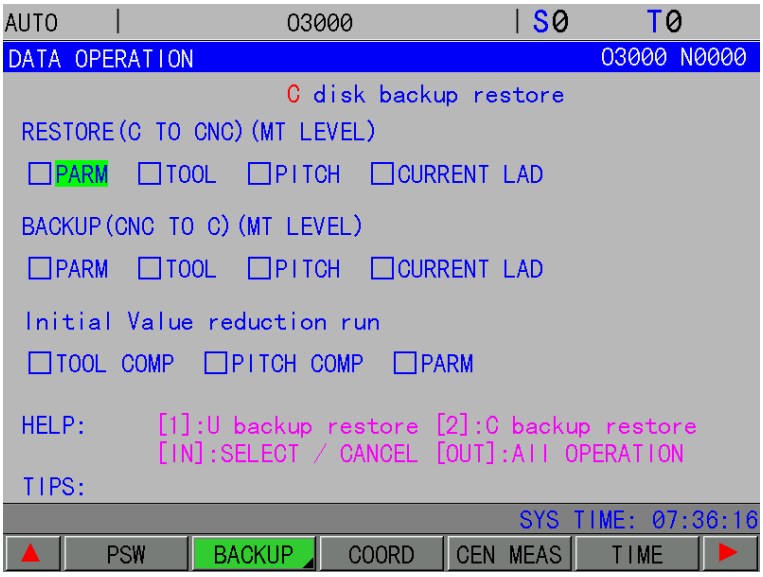
USB flash drive data restore to CNC: Return the data files backed up on the USB flash drive to the system.

CNC data backup to C drive: The current parameters, tool patches, screw patches and ladder diagrams of the system are backed up to the system disk.

CNC data backup to USB flash drive: The current parameters of the system, knife patch, screw patch and ladder diagram are backed up to a USB flash drive.



USB flash drive data backup/restore page



C drive data backup/restore page

3) Permission settings: Display, set the user action level.

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

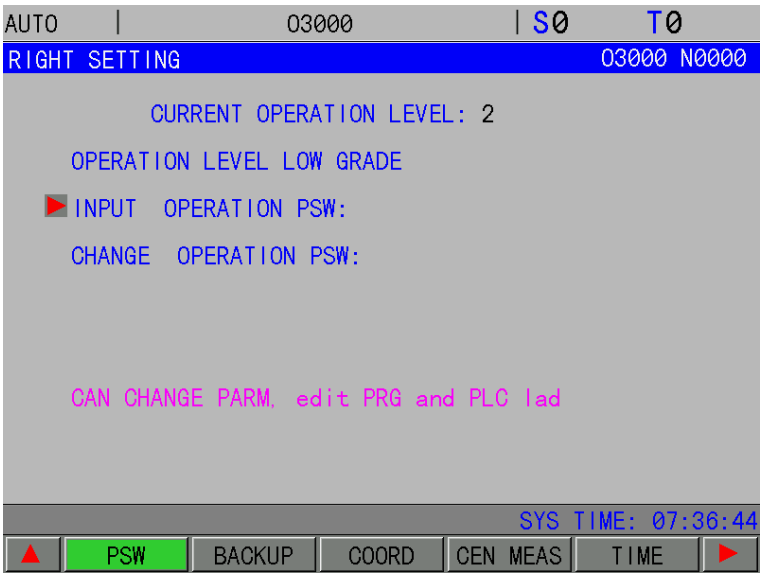
Machine tool manufacturer level: allow modification of CNC status parameters, data parameters, screw repair parameters, tool repair data, edit part programs (including macro programs), edit and modify PLC ladder diagrams, download and upload ladder

diagrams;

Equipment management level: initial password 12345, allowing modification of CNC status parameters, data parameters, knife repair data, editing procedures;

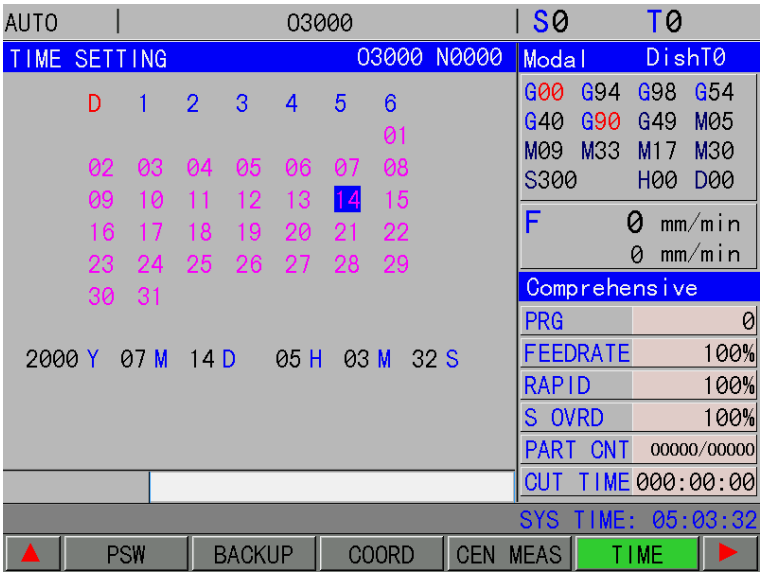
Technician level: initial password 1234, can modify the tool patch data (tool operation), macro variables, edit part program, can not modify the CNC status parameters, data parameters and screw repair parameters.

Processing 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, can not edit the program, can not modify the CNC status parameters, data parameters and screw repair parameters.



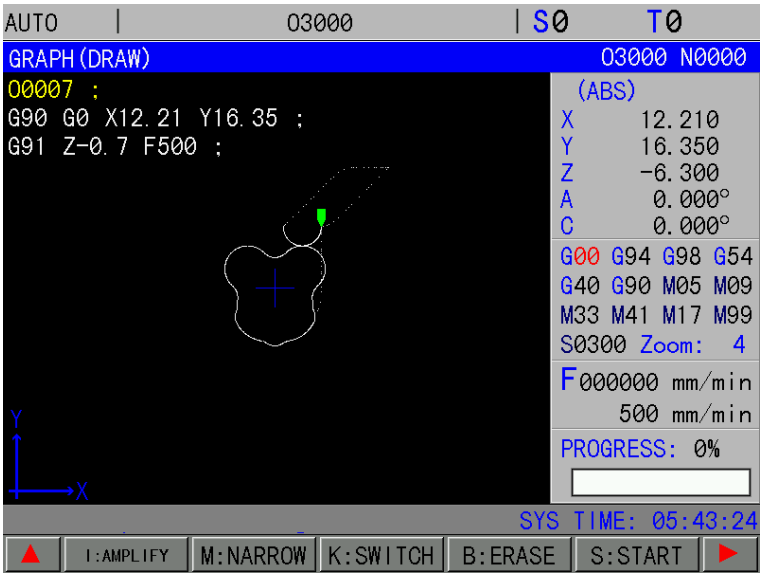
4) Time Date: Displays the current time and date.

You can press the CHG key to enter the modification mode, press ⇦ and the ⇨ key to switch the year/month/day/hour/minute / second that you need to modify, press ⇧, ⇩ keys modify their values, and to exit modify mode, press the CHG key again.



1.3.6 Graphical interface

Press the panel [graphic button] to enter the graphical interface, you can enlarge, zoom out, clear and other operations of the graph; You can also enter the graphical interface through the function key, and find the function key [graphic] in the location interface ▸.



1.3.7 Status parameters, data parameters, snail parameters boundaries

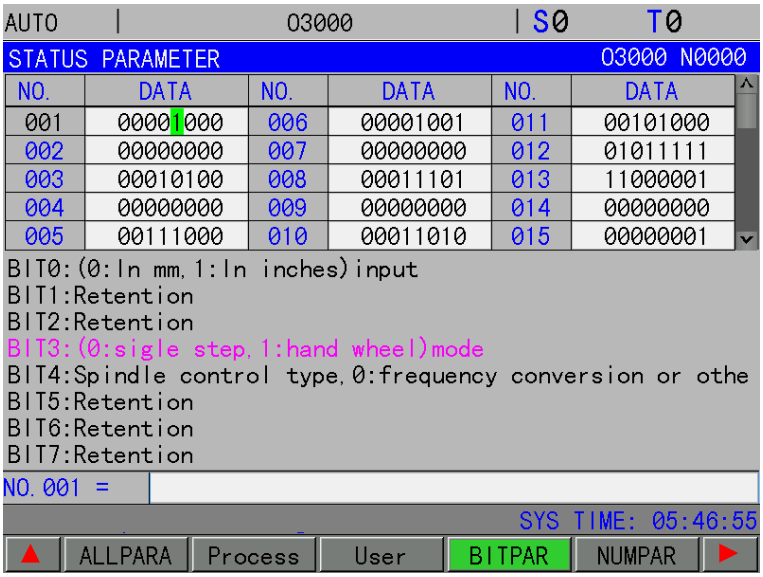
The 

key is a multiplexed key, and repeated pressing of this key can enter several interface surfaces such as parameter summary table, processing technology, user parameters, bit parameters, number parameters and screw compensation parameters.

1) Status parameters interface

Press the PAR key to enter the status parameters interface, the status parameters are displayed in two pages in total, you can enter each page through the key, key to view or modify the relevant parameters, as follows:

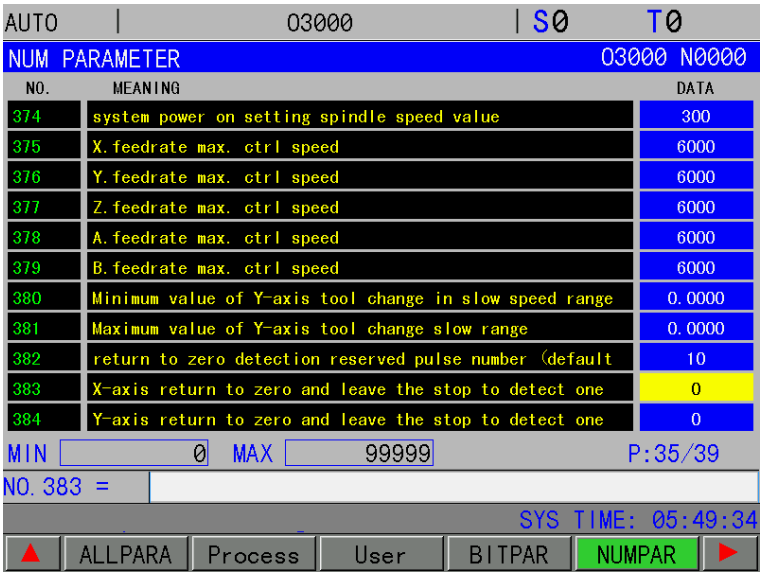
As can be seen from the status parameters page, there are two lines of parameter content display lines in the lower part of the page, and the first line displays the abbreviations of all bits of the parameter where the current cursor is located; The second line shows the Chinese meaning of a bit of the parameter where the current cursor is located, and the displayed parameter bits can be changed by pressing the S key or the D key from the left and right of the panel.



2) Data parameter interface

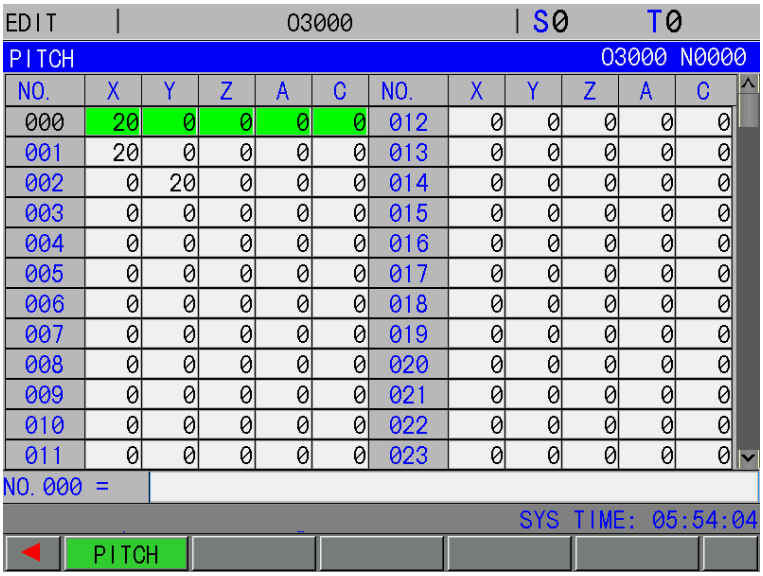
Repeatedly press the PAR key (or press the function key [number parameter]) to enter the data parameter interface, you can enter each page through the key and key to view or modify the relevant parameters, as follows:

As you can see from the Data Parameters page, there is a line in the lower part of the page Chinese prompt line, showing the meaning of the parameter pointed to by the current cursor.



3) Screw patch parameter interface

Repeatedly press the PAR key (or press the function key [screw supplement]) to enter the pitch error compensation interface, the pitch error compensation is a total of 256 points 11 pages display, you can display each page through the key and key:



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

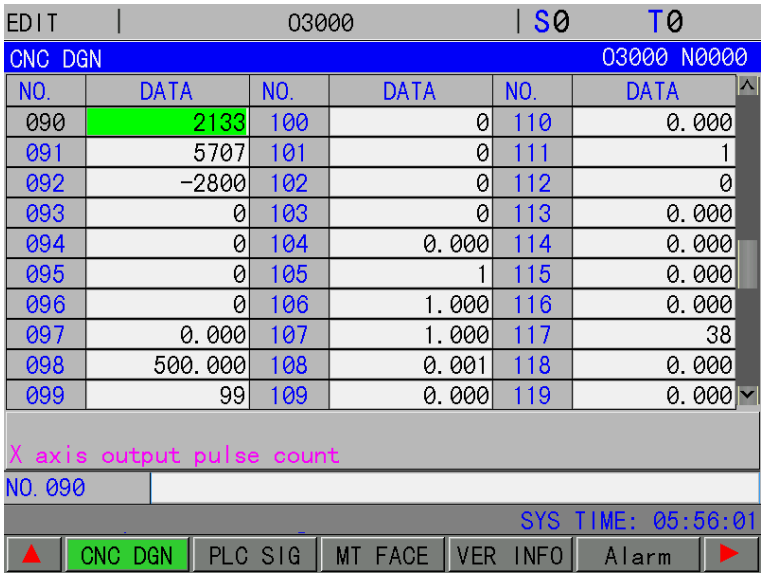
The DGN key is a composite key that can be pressed repeatedly to enter the CNC

diagnostic interface, PLC signal interface, machine tool soft panel, help information and version information interface.

1) CNC diagnostic interface

The status of the input/output signal between the CNC and the machine, the signal status transmitted between the CNC and the PLC, the internal data of the PLC and the internal status of the CNC can be displayed through diagnostics. Press the DGN key to enter the CNC diagnostic page display, CNC diagnostic page displays keyboard diagnostics, status diagnostics and auxiliary function parameters. Can be viewed by key, key.

On the CNC diagnostic display page, the lower part of the page has two lines of diagnostic number details to display the line, the second line shows the Chinese meaning of a bit of the diagnostic number where the current cursor is located, you can press the S key or D key to change the displayed diagnostic bit; The first line displays the abbreviations of all bits of the diagnostic number where the current cursor is located.

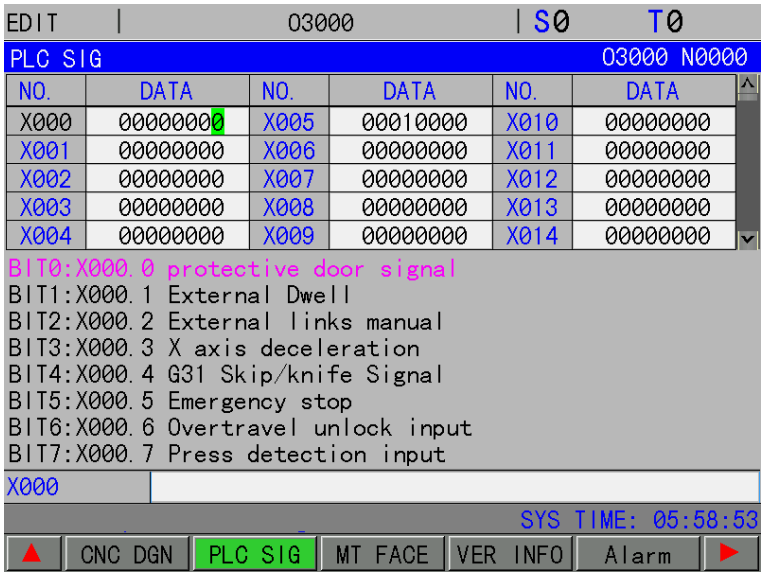


2) PLC signal interface

On the PLC signal interface page, X0000 ~ X0063, Y0000 ~ Y0047, F0000 ~ are displayed in turn F063, G0000 ~ G063 and other address status. Press the DGN key repeatedly to enter the PLC status interface. Press the key and key to view the signal status of each address of the PLC.

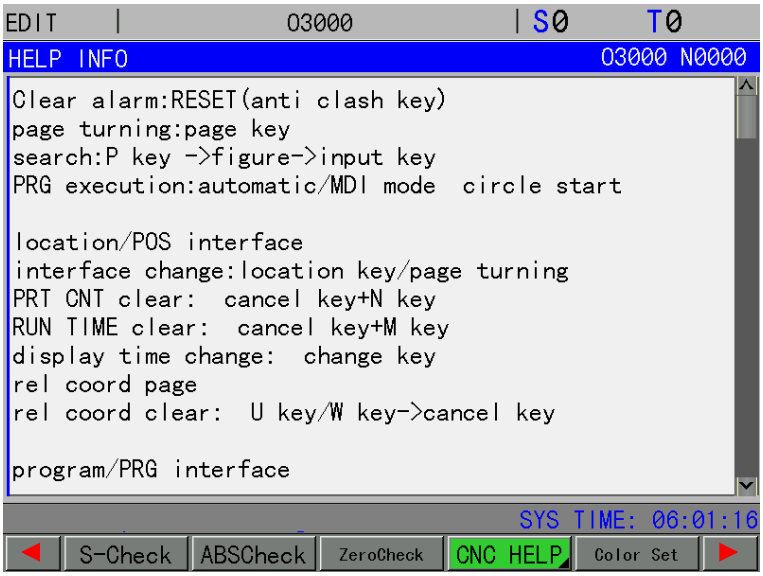
In the PLC status page, the lower part of the page has two detailed content display lines, the second line shows the Chinese meaning of a bit of the address number where the current

cursor is located, you can press the S key or D key to change the displayed address bit; The first line shows the abbreviations of all bits of the address number where the cursor is located.



3) Help information

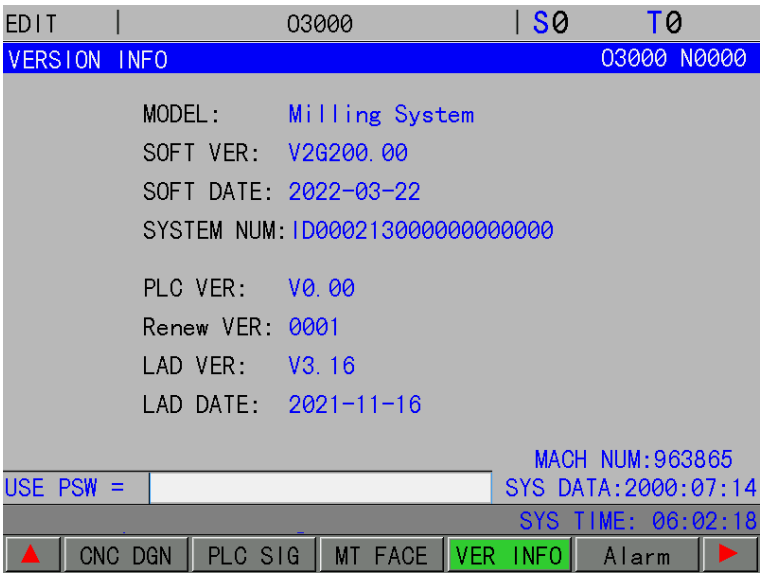
Press the DGN key repeatedly to enter the help information interface. On the help information page, you can view the current operation table, alarm table, G code table, macro and other information of the CNC, and the display page is shown in the figure.



4) Version information

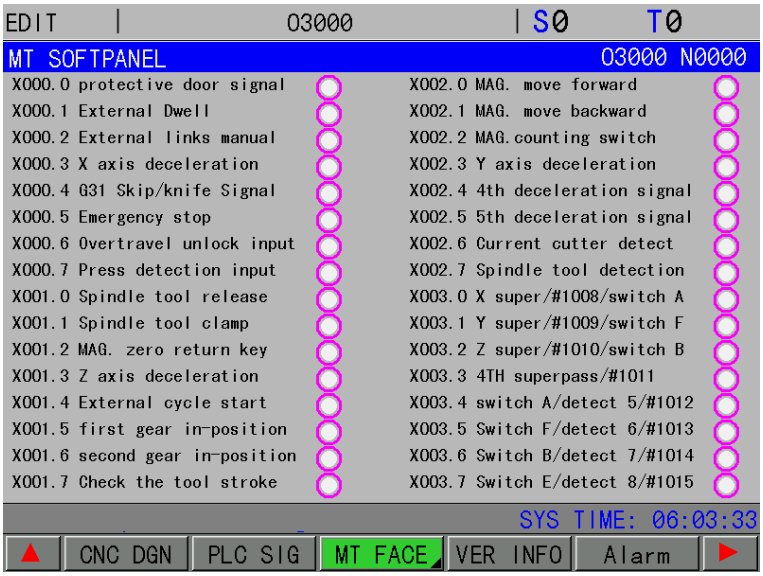
Press the DGN key repeatedly to enter the version information interface. On the version

information page, the current software, hardware, system number, PLC version information, etc. of the CNC are displayed, and the display page is shown in the figure.



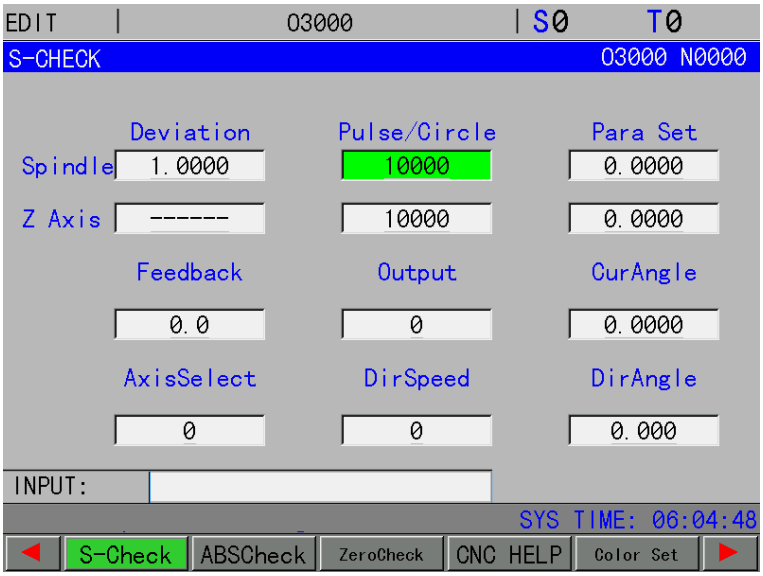
5) IO diagnostics

Press the DGN key repeatedly to enter the IO diagnostic interface. The system input and output status is displayed on the IO diagnosis page, and the display page is shown in the figure.



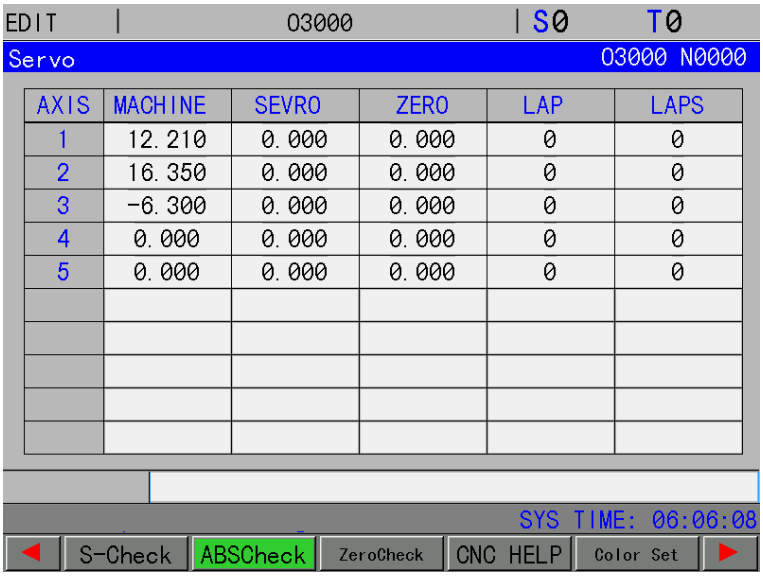
6) Spindle inspection

Press [Diagnostics] to enter the diagnostic interface and then press the function key [] to move the menu, find the function key [spindle detection] key, click into 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 as shown in the ▸ figure.



7) absolute value

Press [Diagnostics] to enter the diagnostic interface and then press the function key [] to move the menu, find the function key [absolute value] key, click into the absolute value interface, in the ▸ absolute value interface can display the coordinates of each axis of the machine tool, motor feedback coordinates, zero point position, etc. for debugging Absolute value motor, the display page is shown in the figure.



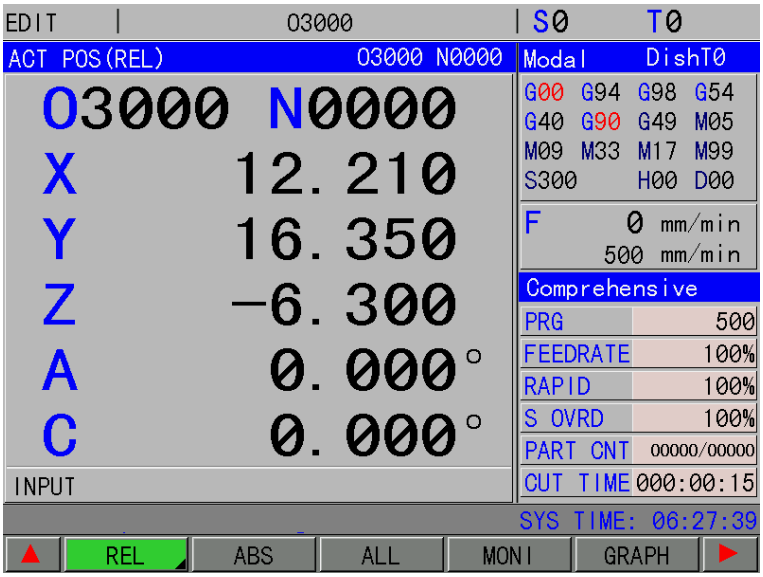
Chapter 2 Start-up, Shutdown and Security Protection

2.1 Power on

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. The wiring is correct and firm.

System self-test, initialization. After the self-test and initialization are completed, the current position (relative coordinates) page is displayed.



2.2 Shut down

Before the system shuts down, you should confirm:

1. The feed axis of the CNC is in a stopped state;
2. Auxiliary functions (such as spindles, pumps, etc.) are turned off;
3. First cut off the CNC power supply, then cut off the machine power supply.

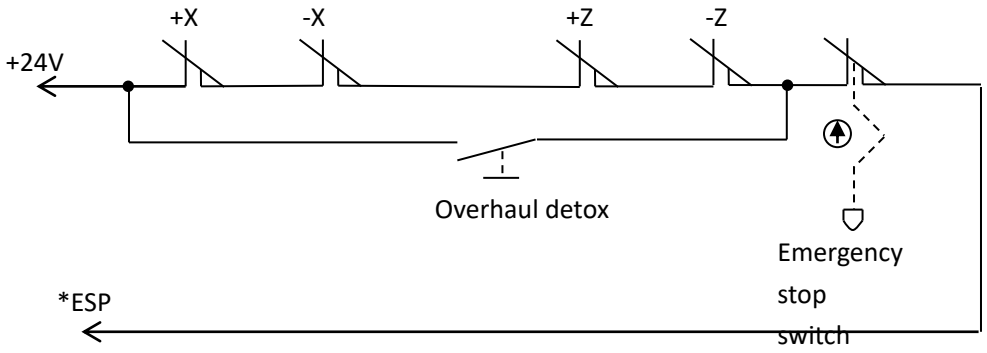
Note: Please refer to the instructions of the machine tool manufacturer for the operation of cutting off the power supply of the machine tool.

2.3 Overdrive protection

In order to avoid damage to the machine tool due to the X, Y and Z axes exceeding the stroke, the machine must take overhaul protection measures.

2.3.1 Hardware Overdrive Protection

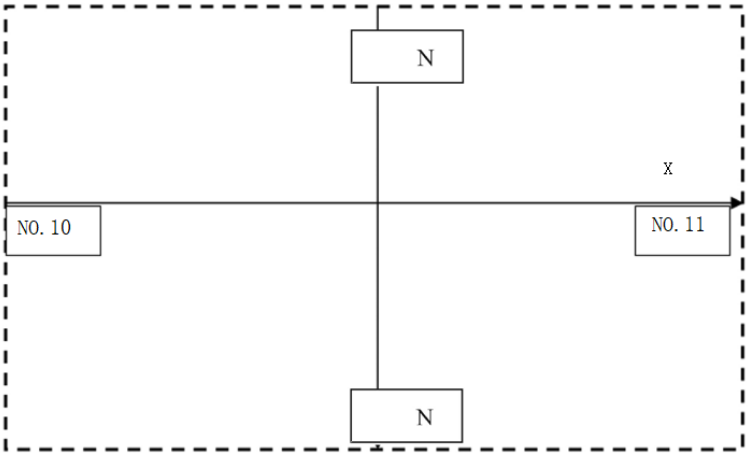
Install the stroke limit switch at the positive and negative maximum strokes of the X, Y, and Z axes of the machine tool respectively, and wire them as shown below, at which point the status parameter No.60 Bit2 (EALM) and the following figure are wiring Bit3 (LALM) must be set to 0. When overtravel occurs, the stroke limit switches and the system stops motion and displays an emergency stop alarm.



When the hardware overruns, the system will have an "emergency stop" alarm. The method of eliminating the "emergency stop" alarm is: press the overdrive button not to release, switch to the alarm information page, view the alarm information, reset and clear the alarm, move the workbench in the opposite direction (such as positive overflight, then move out in the negative direction; If the negative overrun, the positive move out) out of the stroke switch.

2.3.2 Software Overdrive Protection

The software travel range is set by the data parameters NO.010 ~ NO.019 , with the machine coordinate value as the reference value. The Bit1 (LZR) of the status parameter No.022 sets the function of the software limit before the mechanical zero return.



If the machine position (machine coordinates) exceeds the dotted area in the image above, an override alarm occurs. The method of unlocking the overhaul alarm is: press the reset key, clear the alarm display, and move in the opposite direction (if the positive overrun, the negative direction is removed; If the negative direction is overrun, then the positive direction is moved out) is enough. 2.4 Emergency Operations

During the machining process, due to user programming, operation and product failure, some unexpected results may occur, at which time the system must be stopped immediately. This section describes the handling that can be carried out by the system in case of emergency, and the handling of CNC machine tools in emergency can be found in the relevant instructions of the machine tool manufacturer.

2.4 Emergency Operations

During the machining process, due to user programming, operation and product failure, some unexpected results may occur, at which time the system must be stopped immediately. This section describes the handling that can be carried out by the system in case of emergency, and the handling of CNC machine tools in emergency can be found in the relevant instructions of the machine tool manufacturer.

2.4.1 Reset

When the system abnormal output or coordinate axis abnormal action, press the 

key to make the system in the reset state:

1. All axis movements are stopped;
2. M, S function output is invalid (can be set by the parameter setting button 

whether to automatically turn off the spindle counterclockwise rotation / clockwise rotation, lubrication, cooling and other signals, PLC ladder diagram definition);
3. Automatic operation ends, modal functions, status is maintained.

2.4.2 Emergency stop

During the operation of the machine tool, press the emergency stop button in case of danger or emergency (when the external emergency stop signal is active), 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 dismiss 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: Before lifting the emergency stop alarm, confirm that the fault has been ruled out;

Note 2: Pressing the E-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 does not have the zero point of the machine tool installed, the operation back to the zero point of the machine tool shall not be carried out);**

Note 4: An external emergency stop is only valid if **Bit2 (EALM) of the status parameter NO.60 is set to 0.**

2.4.3 Feed keeping

During machine operation, you can press the 

key to pause the operation. It is important to note that this feature does not stop the running action immediately during thread cutting and in the running of the cyclic code.

2.4.4 Cut off the power supply

During the operation of the machine in case of danger or emergency, the power supply of the machine can be cut off immediately to prevent accidents. However, it must be noted that after the power supply is cut off, the CNC display coordinates may deviate greatly from the actual position, and operations such as re-tool realignment must be performed.

Chapter 3 Manual Operation!

The function of the keys 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 manual of the machine tool manufacturer.

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

Press the 

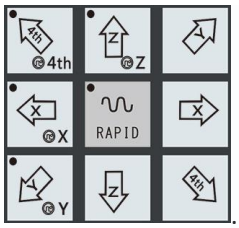
key to enter the manual operation mode, and the manual operation mode can be operated by manual feed, spindle control, magnification adjustment, tool change and so on.

3.1 Axis movement

In the manual operation mode, the two axes can be manually fed and moved quickly manually.

3.1.1 Manual feeding

Press and hold 

or  in the feed axis and direction selection key 

. The X-axis arrow key can make the X-axis negative or positive feed, and the axis movement stops when the key is released; Press and hold 

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

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

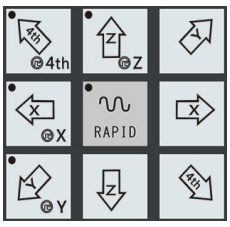
or  , the 4th-axis arrow key can make the 4th-axis negative or positive feed, and the axis movement stops when the key is released.

When a manual feed is performed, press the 

key so that the indicator light in the status indicator area lights up to enter the manual fast movement state.

3.1.2 Manual fast movement

Press the 

key in 

until the fast moving indicator in the status

indicator area lights up. Press the  or  key to make the X axis move quickly in a negative or positive direction, and the axis movement stops when the key is released; Press and hold  or  , the Z-axis arrow key can make the Z-axis negative or positive feed, and the axis movement stops when the key is released; Press and hold  or  , the Y-axis arrow key can make the Y-axis negative or positive feed, and the axis movement stops when the key is released; Press and hold  or  , the 4th-axis arrow key can 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 a manual fast movement, press the  key to turn the LED off and the fast movement ineffective, feed at manual speed.

Note 1: After powering on, if no reference point is returned, whether the fast movement speed is manually fed or fast movement speed is determined by the **Bit0 bit** (**ISOT**) **of** the **system status parameter NO.012** when the fast move switch is turned on (fast movement indicator is on). **Select;**

Note 2: The  **key is invalid in Edit / Handwheel mode.**

3.1.3 Speed trimming

In manual feed, the manual feed rate can be modified by  , a total of 16 levels. When the parameter NO.110 is set to 1260, the relationship between the feed magnification and the feed rate is shown in the following table

Feed multiplier (%)	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 fast in hand, you can press or modify the magnification of manual fast movement, and the fast magnification is Fo, 25%, 50%, 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, press this key, the spindle turns counterclockwise;

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

: In manual mode, press this key and the spindle rotates clockwise.

3.2.2 Spindle jogging

: The spindle is jogging at this point.

Function description: Press the 

key on the panel to enter the jog state, and the opening and closing of the spindle jog function requires the spindle to be in a stopped state.

Spindle jog state, press the 

buttons, counterclockwise jogging; Press the 

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

When the spindle is jogged and rotated, the 

key can stop the spindle jogging rotation, and the spindle braking signal will not be output when the jogging rotation is stopped.

K10.4 is set to 1, and the spindle jog is valid in any way. When the spindle is in jog rotation state automatically or in the input mode, the running program will turn off the spindle jog rotation and turn off the jog function.

Parameter settings:

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

Data parameter No.208: Speed of rotation when the spindle is jogged

3.2.3 Coolant control

: In any mode of operation, press this key and the coolant switches 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, the output is flipped for lubrication, press the machine operator panel 

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

When M32 is executed, the lubrication output is performed, and then M33 is executed, and the lubrication output is canceled.

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

key, lubrication output, after the time set by data parameter 112, the lubrication output is canceled; Execution of M32, lubrication output, after the time set by the data parameter DT13, lubrication output is canceled. If the time set by DT13 is not reached, and M33 is performed at this time, the lubrication output is canceled.

2) Automatic lubrication:

DT13>0: Automatic lubrication, lubrication time DT13 and lubrication interval DT53 can be set

After the system is powered up, it starts to lubricate the time set by DT13, then stops the output, after the time set by DT53, repeats the output lubrication, and then cycles in turn. When lubricating automatically, the M32, M33 codes and the machine operator panel keys are also effective.

Parameter settings:

PLC Parameter: K10.1 1/0: Spindle lubrication cooling output maintains/closes when reset

PLC Parameters: K16.2 1/0: Automatic lubrication is effective when boot 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, press this key and the spindle tool is released/ clamped.

3.2.6 Revision of spindle magnification

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

Press the 

button, adjust the spindle magnification to change the spindle speed, and realize the spindle magnification of 50% ~ 120% in 8 levels of real-time adjustment.

Chapter 4 Handwheel / Single Step Operation

In the handwheel/ single-step operation mode, the machine moves at the selected increment value.

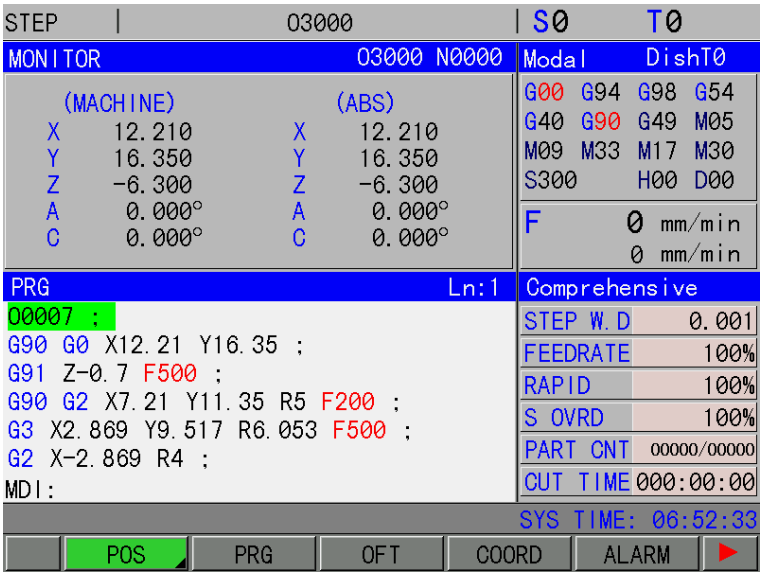
The function of the keys in the system machine panel is defined by the PLC program (ladder diagram), and the functional significance of each button can be found in the manual of the machine tool manufacturer.

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

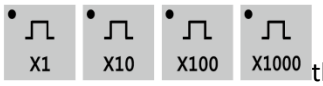
4.1 Single step feed

Set the bit3 bit of system parameter No.001 to 0, press the 

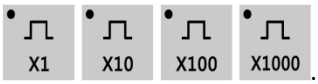
key to enter the single-step mode, and the page is displayed as follows:



4.1.1 Incremental selection

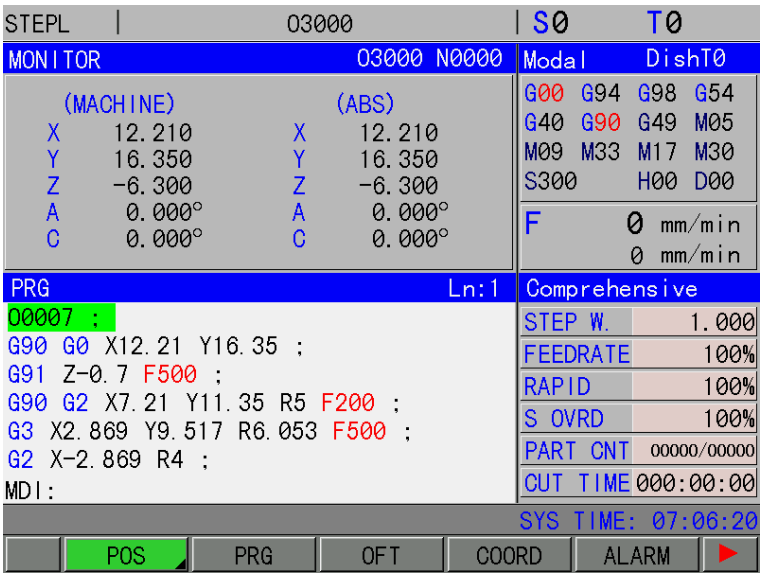
Press 

the key to select Move Increment, which is displayed on the page. When the BIT 7 bit (SINC) of the PLC state parameter K016 is 1, the 

step value is invalid; All are valid when BIT7 is 0 

. For example, if button 

is pressed, the following information is displayed:

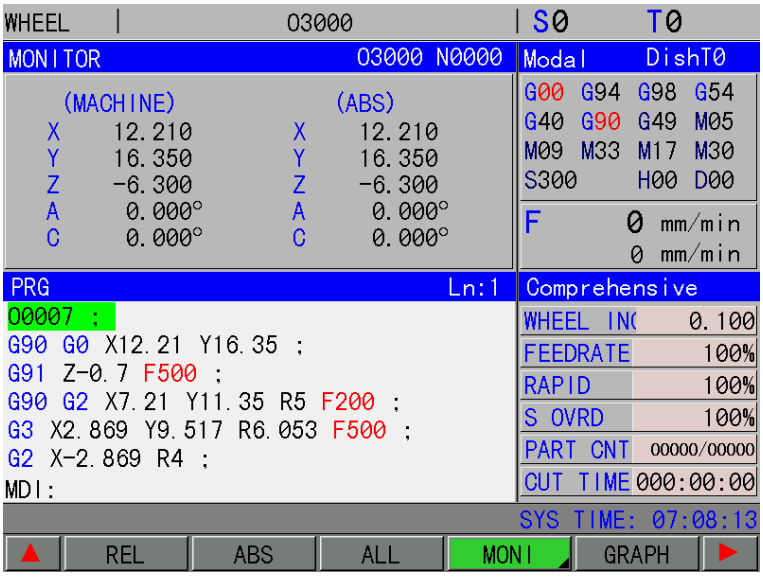


4.1.2 Movement direction selection

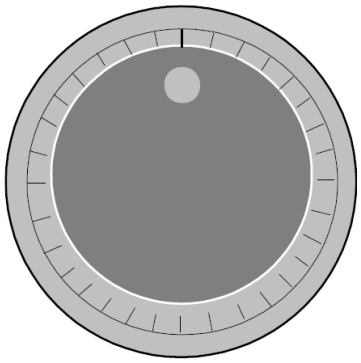
Press  or  key once to make the X axial negative or positive feed once in a single step increment; Press  or  key once to make the Z axial negative or positive feed once in single step increments. Press  or  key once to make the Y axial negative or positive feed once in a single step increment.

4.2 Handwheel (Hand Pulse Generator) feed

Set the system parameter No.001 bit3 bit to 1, press the  key to enter the handwheel operation mode, and the page is displayed as follows:

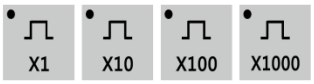


The shape of the handwheel is shown in the following figure:

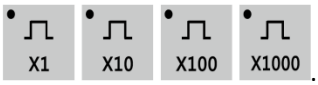


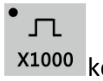
Handwheel shape drawing

4.2.1 Incremental selection

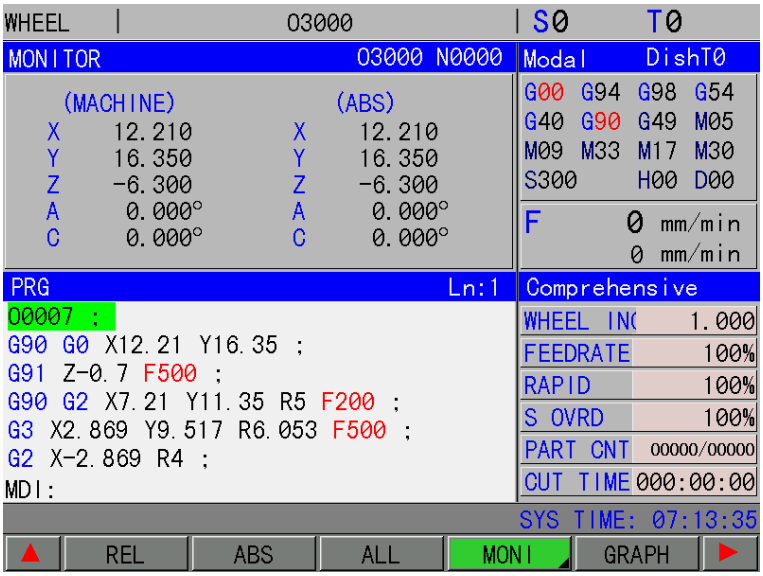
Press the 

key to select Move Increment, which appears on the page. When the BIT 7 bit (SINC) of PLC parameter K016 is 1, the 

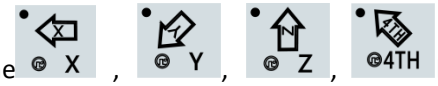
step value is invalid; All are valid when BIT7 is 0 

. If you press the 

key, the page appears as follows:



4.2.2 Selection of the axis and direction of movement

Under the handwheel operation mode, press the 

key 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 a negative feed clockwise and the counterclockwise is a positive feed, the A and B signals at the handwheel end can be exchanged. The direction of feed when the handwheel is rotated can also be selected by the BIT0 bit of parameter No 013.

4.2.3 Other operations

1. Turn counterclockwise, turn clockwise, stop control

:Handwheel / Single step mode, press this key, the spindle turns counterclockwise;

: Handwheel / Single Step mode, press this key, the spindle stops;

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

2. Spindle joking

: The spindle is jogging at this point.

In the spindle jog state, press the 

button, turn the jog counterclockwise; Press the 

button to turn the jog clockwise. The spindle jog speed is set by the data parameter No.208. See Section 3.2.2 of this Article.

3. Coolant control

See Section 3.2.3 of this Article

4. Lubrication control

See Section 3.2.4 of this Article

5. Manual tool change

: In handwheel/ single step mode, press this key and the tool is released / clamped.

6. Adjustment of spindle magnification

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

Press the 

button, adjust the spindle magnification to change the spindle speed, and realize the spindle magnification of 50% ~ 120% with a total of 8 levels of real-time adjustment.

4.2.4 Notes

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

	The amount of movement per moment on the handwheel			
Handwheel increments	0.001	0.01	0.1	1
The coordinates specify a value	0.001mm	0.01mm	0.1mm	1mm

(Minimum input increment 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 the handwheel rotation must not be higher than 5r/s, **if more than 5r/s, it may cause a discrepancy between the scale value and the amount of movement.**

Chapter 5 Entry Operations

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

The function of the keys 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 manual of the machine tool manufacturer.

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

5.1 Input of code words

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

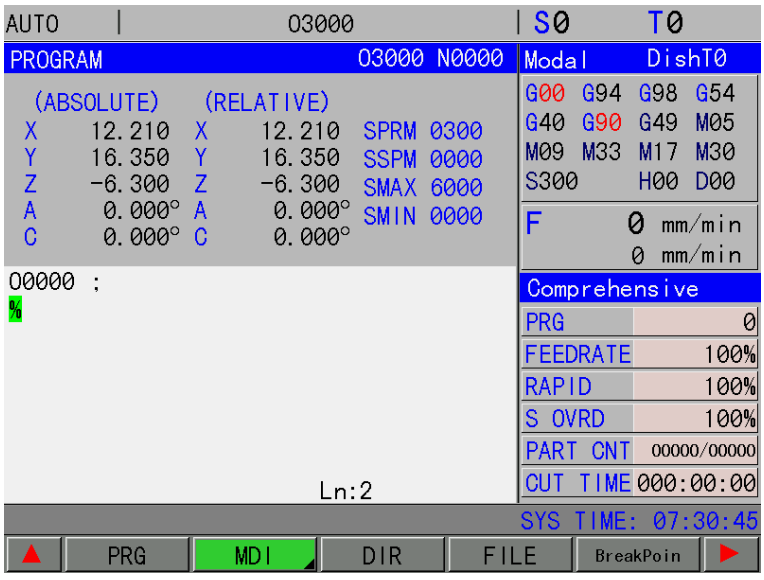
1. Press the 

key to enter the input operation mode;

2. Press the 

key (press the  key or  key again if necessary, or press the 

key multiple times) to enter the program status page:



3. Type the address key 

, the number key 

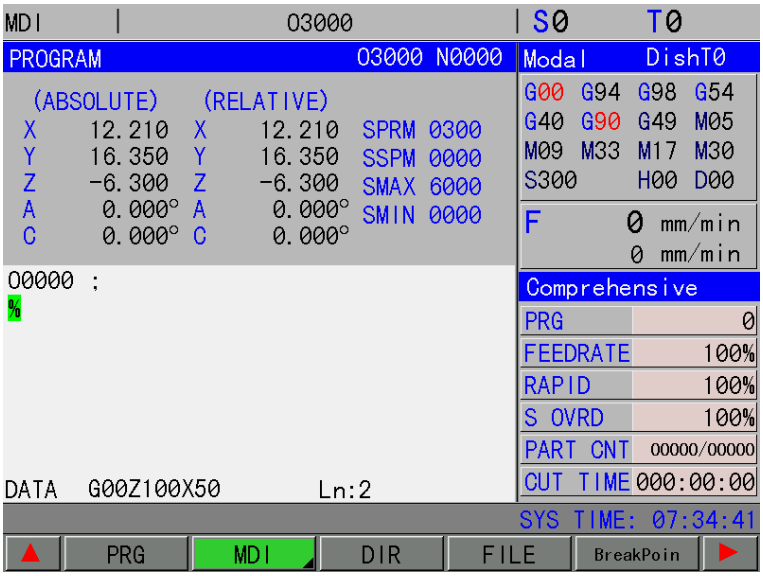
, and then type 

;

4. Type the address key Z, the number key 1, 0, and then 0;

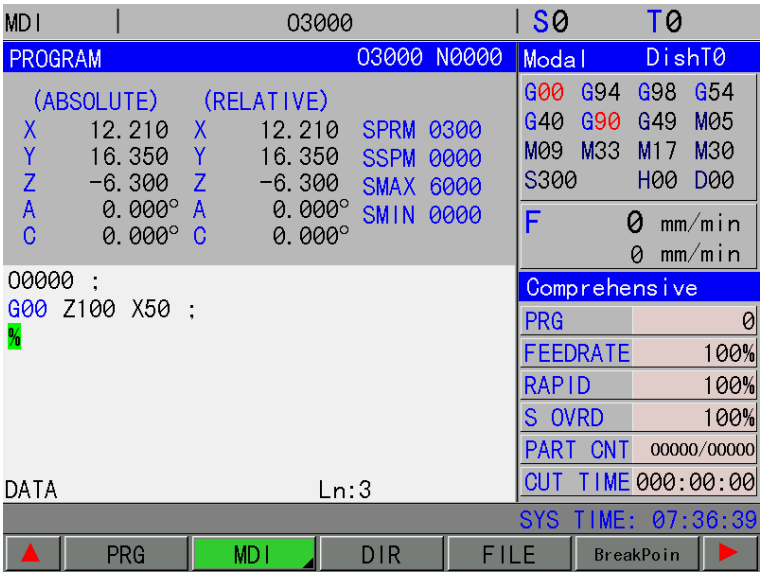
5. Type the address key X, the number key 5, and then type 0;

After performing the above operations, the page is displayed as follows (can enter 4-paragraph program, display 6-segment program):



5.2 Execution of code words

After the code word is entered, press the IN key or Enter key, and the page appears as follows:



Pressing the CYCLESTART key to execute the input program segment. During operation, you can

press 

keys, 

keys and E-STOP buttons to stop the program segment.

Note: Subroutine call code (M98 P; etc.) is invalid executed under MDI.

5.3 Setting parameters

In the input mode, enter the parameter interface to modify the parameter value, see Chapter 10 of this article.

5.4 Modification of Data

On the program status page, before executing the input program segment, if there is an error in the field input process, you can press the 

key to cancel the inverted state and modify the program, or press the 

key to clear all the contents and re-enter the correct program segment.

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.

   Press the button 

, adjust the spindle magnification to change the spindle speed, can achieve the spindle magnification of 50% ~ 120% a total of 8 levels of real-time adjustment.
2. Adjustable fast magnification button, trim fast movement feed speed, can achieve fast movement speed 4 levels of real-time adjustment.
3. Adjustable feed magnification

   Under the input operation mode, press the 

button, adjust the feed magnification to change the feed speed, and realize the real-time adjustment of the actual speed of 0 ~ 150%

of the feed speed specified by the F code.

Chapter VI: Program Editing and Management

Under the editing operation mode, programs can be established, selected, modified, copied and deleted, and two-way communication between CNC and CNC, 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 a 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 rights settings. You must have more than 4 levels of operation (level 4, level 3, and so on) to open the program switch and edit the program. The operations allowed for each action level are described in Section 10.3.

6.1 Establishment of the Program

6.1.1 Generation of program segment numbers

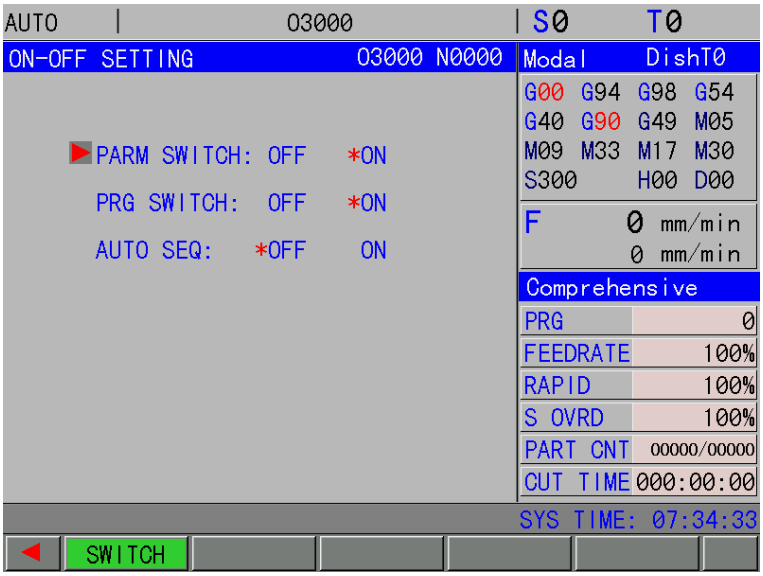
In a program, the program segment number can be and cannot be programmed, and the program is executed in the order in which the program segment is programmed (except when called).

When the Auto Sequence Number switch on the switch setup page is off, the CNC does not automatically generate program segment numbers, but program segment numbers can be manually programmed at programming.

When the Auto Sequence Number switch on the switch settings page is on, the CNC automatically generates the program segment number, and when editing, the 

key or 

key automatically generates the program segment number for the next program segment, and the increment value of the program segment number is set by the CNC data parameter No 316 (The setting of the auto-sequence number is described in Section 10.1.1 of this article.)

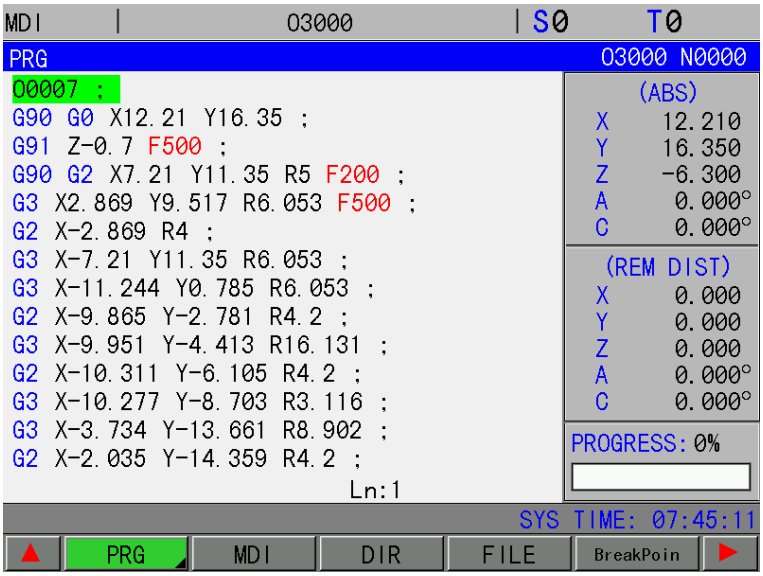


6.1.2 Input of program content

1. Press the 

key to enter the editing operation mode; Press the 

key to enter the program interface.



2. Type the address key 

, the number key 

, 

, 

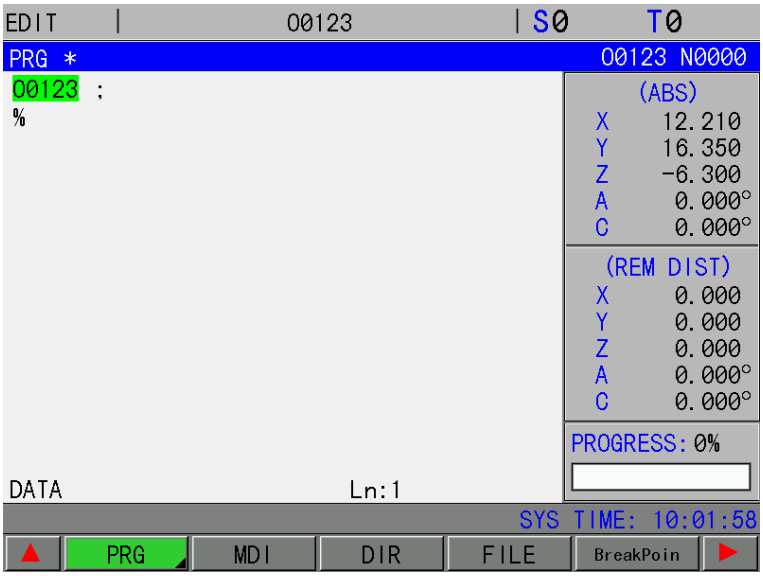
, 

(for example, to build the O 0123 program).

3. Press a 

key or 

key to create a new program.



4. According to the prepared parts of the program one by one input, each input of a character, on the screen immediately given the display input of the character (the processing of the composite key is to repeatedly press this composite key to achieve alternate input), after the completion of the need to switch to other working methods to store the program.
5. Follow the method in step 4 to complete the input of other program segments.

Note: 1, in the editing mode, only the input of the complete instruction word can be entered, a separate letter or number, the system does not support.

2. When entering the program, it is found that the input instruction word is wrong, and the input instructions can be cancelled one by one by pressing the 

key.

6.1.3 Cursor movement

1) Press the 

key to enter the editing mode, and press the 

key to select the program content display page.

2) Press the  key, the cursor moves up one line; If the number of columns in which the current cursor is located is greater than the total number of columns in the previous row, after pressing the  key, the cursor moves to the end of the previous program paragraph;

3) Press the  key, the cursor moves down one line, if the number of columns in the current cursor is greater than the total number of columns in the next row, after pressing the  key, the cursor moves to the end of the next row;

4) Press the  key, move the cursor to the right by a column; If the cursor is at the end of the line, the cursor moves to the beginning of the next program segment;

5) Button  , the cursor moves one column to the left; If the cursor is at the beginning of the line, the cursor moves to the end of the previous program segment;

6) Press the  button to turn the page upwards;

7) Press the  button to turn the page down;

8) On the Edit Mode and Program Display page, press the  key and the cursor returns to the beginning of the program.

6.1.4 Retrieval of word and line numbers

Word retrieval: Starts at the current position of the cursor and looks up or down for the specified character

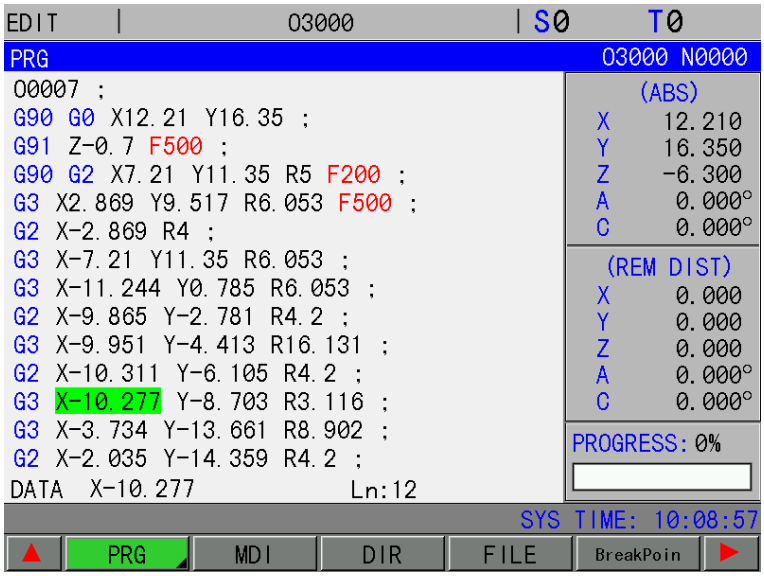
The steps to find the method are as follows:

1) Press  to select the edit operation mode;

2) Press the  key to display the program content page;

3) Enter the word you want to retrieve.

4) Press the  key (to determine whether a  key or  key is based on the position of the character you want to find and the character at which the current cursor is located), the page is displayed as follows:



5) Press the up/ down arrow keys again to find the next position of the word.

Note 1: If not found, the retrieved word entered disappears.

Note 2: Character retrieval and scanning cannot be performed in macro editing mode.

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

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

The steps to find the method are as follows:

1) Select the editing operation mode, and the program content display page;
2) Press the 

key (address bar data should be empty);
3) Enter the program line number;
4) Press the 

key or 

key, and the cursor will jump to the program line number entered.

6.1.5 Word insertion

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Enter the word you want to insert, press the 

key, and the system will insert the input to the left of the cursor.

6.1.6 Deletion of words

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Position the cursor to the location that needs to be deleted, press the 

key, the system will delete the content where the cursor is located, and if the 

keys are pressed continuously, the program content on the right side of the cursor will be deleted continuously.

6.1.7 Modification of words

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor to where you need to modify it, enter the modified word, and then press

the ALT key to replace the content of the cursor positioning with the content entered.

6.1.8 Deletion of Single Program Segments

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor to the beginning of the delete program segment (column 1), press the N# key, and then press the DEL key to delete the segment where the cursor is located.

6.1.9 Deletion of multiple program segments

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor to the beginning of the deleted program segment;
3) Enter the sequence number of the last program segment of the section to be deleted;
4) Press the DEL key to delete the program segment between the cursor and the tag address.

6.1.10 Block removal

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the marker to the first code word of the block to be deleted;
3) Enter the last code word to delete the block;
4) Press the DEL key to delete the block between the cursor and the tag address.

Note 1: If the deletion is successful, the code word entered will disappear, otherwise the deletion fails. If there are multiple identical characters down the specified character, the default one is closest to the current cursor.

6.1.11 Copying of Single Program Segments

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor to the beginning of the line of the copied program segment;
3) Press the N# key, then the CHG key, to copy the program segment where the cursor is located.

6.1.12 Copying of multiple program segments

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor over the first code word of the segment you want to copy;
3) Enter the sequence number of the last paragraph of the program segment to be copied;
4) The program segment between the CHG key, cursor and the input character is copied.

6.1.13 Copying of Blocks

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor over the first code word of the block you want to copy;
3) Enter the last code word of the block you want to copy;
4) The block copy between the CHG key, cursor and input characters is complete.

6.1.14 Paste of program blocks

The steps to do this are as follows:

1) Select the editing operation mode, and the program content display page;
2) Move the cursor to the pasted program position;
3) Press the OUT key to finish pasting the last copied program content before inserting it into the cursor.

6.2 Removal of Programs

6.2.1 Deletion of individual programs

The procedure is as follows:

1) Select the editing operation mode and enter the program display page;
2) Type the address key O and the number key 0, 0, 0, 1 (take the O0001 program as an example);
3) By pressing the DEL key, O 0001 program is removed.

6.2.2 Removal of all programs

The procedure is as follows:

1) Select the editing operation mode and enter the program display page;
2) Type the address key O, the symbol key -+, the number key 9, 9, 9, 9 in turn;
3) Press the DEL key and all programs are deleted.

6.3 Selection of Programs

When there are already multiple programs in the CNC, there are three ways to select the programs.

6.3.1 Search Method

1) Select the editing or automatic operation mode;
2) Button PRG and enter the program content display screen;
3) Press the address key O and type the program number;
4) Press ⇩ or EOB key or Enter key or press ⇩ in automatic mode to display the retrieved program on the display screen, if the program does not exist, the CNC appears alarm.

Note: In step 4, under Edit Mode, if the program does not exist, the CNC creates a new program **after** pressing **the** EOB **key or** Enter key**.**

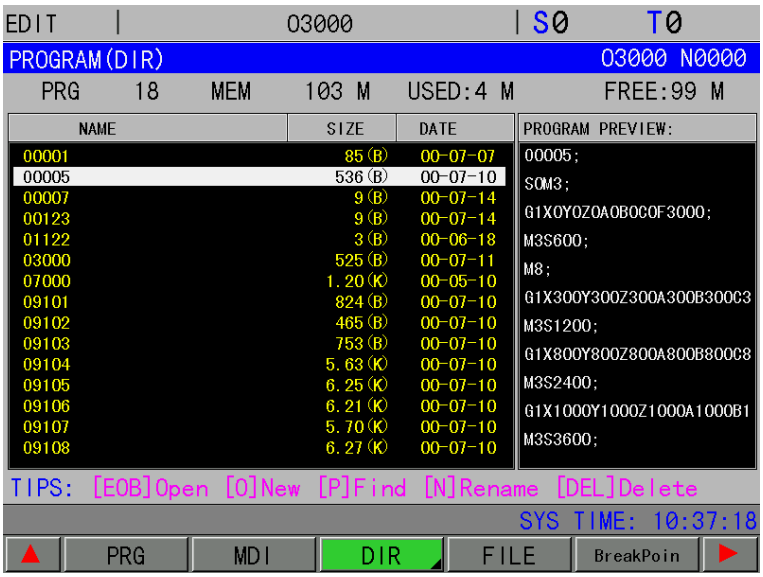
6.3.2 Scanning Method

1) Select the editing or automatic operation mode;
2) Press the PRG key and enter the program display screen;
3) Press the address key O;
4) Press ⇩ or ⇧ key to display the next or previous program;
5) Repeat steps 3 and 4 to display the stored programs one by one.

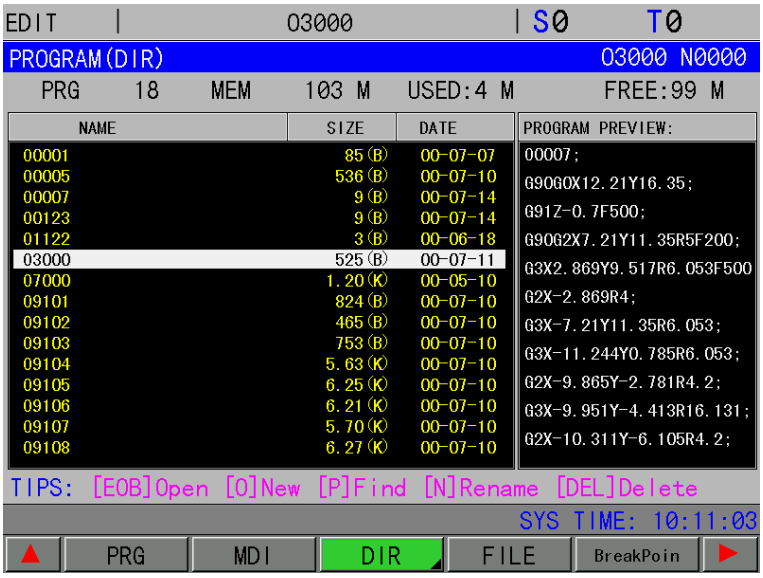
6.3.3 Cursor Confirmation Method

1) Select the automatic mode of operation (must be in a non-running state);

2) Press the PRG key to enter the program directory display page;



3) Press ⇧ or ⇩ key cursor move up/down, or press or key to turn the page up/ down, or press ✕ or |A key cursor to move to the first/ last program in the directory, move the cursor to the name of the program to be selected (as the cursor moves, the program content also changes);



4) Press the EOB key or IN key or Enter key.

6.4 Renaming of the Program

1) Select the editing operation mode to enter the program content display page;

2) Press the address key O and type the new program name;

3) Press ALT key.

6.5 Copying of Programs

To save the current program:

1) Select the editing operation mode to enter the program content display page;

2) Address key O, type the new program number;

3) Press CHG key.

6.6 Program Management

6.6.1 Program Directory

Press the PRG key to enter the program directory display page. On this page, CNC stored program names are displayed as a table of contents, with a maximum of 10 program names on one page, and when the number of stored programs exceeds 10, you can press the up/down page key to display another page of the program directory.

1) Open the program

Open the specified program: O + Serial Number + Enter (or EOB) or

Sequence Number + Enter (or EOB).

In edit mode, programs are created if the ordinal number entered does not exist.

2) Removal of the program:

1. Edit mode Press DEL to delete the program specified by the cursor.
2. Edit by O + ordinal + DEL or ordinal + DEL.

6.6.2 Number of Stored Programs

The number of stored programs in this system is up to 400, which displays the number of parts that the CNC can currently store and remain.

6.6.3 Storage Capacity

The program storage capacity of this system (total capacity is 56M), which displays the storage capacity and remaining storage capacity that have been occupied.

Chapter VII Automatic Operation

The function of the keys 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 manual of the machine tool manufacturer.

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

7.1 Autorun

7.1.1 Selection of running programs

1. Search method

1) Select the editing or automatic operation mode;

2) Button 

and enter the program content display screen;

3) Press the address key  and type the program number;

4) Press the  key to display the retrieved program on the display screen, if the program does not exist, the CNC appears alarm.

2. Scanning

1) Select the editing or automatic operation mode;

2) button 

and enter the program display screen;

3) Press the address key  ;

4) Press  or  key to display the next or previous program;

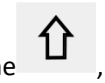
5) Repeat steps 3), 4) to display the stored programs one by one.

3. Cursor confirmation method

1) Select automatic mode of operation (must be in a non-running state)

2) Press the 

key to enter the program directory display page (press the  key and  key if necessary);

3) Press the  ,  ,  ,  key to move the cursor to the program name to be

selected.

4) Press 

key or 

key.

7.1.2 Start of Autorun

1. Press the 

key to select the automatic operation mode;

2. Press the 

key to start the program, and the program runs automatically.

Note: The running of the program starts from the line where the cursor is located, so you should check whether the cursor is on the program segment that needs to be run before pressing the 

key to run.

7.1.3 Stopping of Automatic Operation

* Code stop (M00).

1. M00

After the program section containing M00 is executed, the automatic operation is stopped, and the modal functions and states are all saved. After pressing the panel 

key or the external run key, the program resumes.

2. M01

Press the 

button, select the stop indicator is on, and select the stop function is active. After the execution of the program section containing M01, the automatic operation is stopped, and the modal functions and states are all saved. After pressing the panel 

key or the external run key, the program resumes.

* Press phase key to stop

1. After the automatic operation of the in-process 

button or the external pause key, the machine is in the following state:

1) Machine feed deceleration stop;

2) Modal functions and states are saved;

3) After pressing the 

key, the program continues.

2. Press the reset key 

1) All axis movements are stopped;

2) M, S function output is invalid (It can be set by parameter whether to automatically turn off

the spindle counterclockwise/clockwise, lubrication, cooling and other signals after pressing 

key);

3) Automatic operation ends, modal functions, status is maintained.

3. Press the E-STOP button during the operation of the machine in a dangerous or emergency situation (when the external E-STOP signal is effective), the CNC will enter the E-STOP state, at which time the machine tool movement immediately stops, and all outputs (such as spindle rotation, coolant, etc.) are all turned off. Release the emergency stop button to dismiss the emergency stop alarm and the CNC enters the reset state.

4. When the conversion operation mode is converted to the machine tool to zero, handwheel / single step, manual, program back to zero during the automatic operation, the current program segment is immediately "suspended"; When you convert to edit or enter mode during the autorun process, "Pause" is displayed only after the current program segment has been run.

Note 1: Before lifting the emergency stop alarm, confirm that the fault has been ruled out;

Note 2: Pressing the E-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 does not have the zero point of the machine tool installed, the operation back to the zero point of the machine tool shall not be carried out);**

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

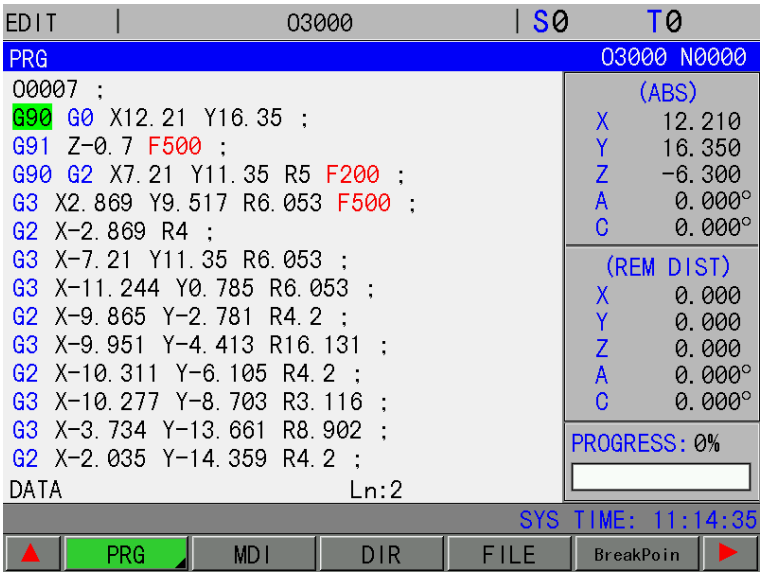
7.1.4 Run automatically from any segment

Press the 

key to enter the editing operation mode, press the 

key to enter the program interface, press the  key or  key to select the program content page;

1、 Move the cursor to the segment of the program that is ready to start running (e.g. from the second line to the beginning of the second line);



2、 If the modality (G, M, T, F code) of the program segment where the current cursor is located is default and inconsistent with the modality of the program segment running the program, the corresponding modal function must be performed before proceeding to the next step;

3、 Press the 

key to enter the automatic operation mode, and press the 

key to start the program to run.

7.1.5 Feed, fast speed adjustment

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

* Feed rate adjustment

Press 

key to realize the feed rate of 16 real-time adjustment.

Note 1: The value specified in **F in the feed magnification adjustment procedure;**

Note 2: The actual feed rate = the value specified in F × feed magnification.

* Quick multiplier adjustment

Press 

or 

key, can achieve fast magnification FO, 25%, 50%, 100% four-stop real-time adjustment.

Note 1: CNC parameters No.090 ~ **No.094 set the fast movement rate of each axis separately;**
Actual fast movement rate = parameter set value × fast rate
Note 2: When the fast magnification is F0, the minimum rate of fast movement is set by the 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.

Press the 

button, adjust the spindle magnification to change the spindle speed, and realize the spindle magnification of 50% ~ 120% A total of 8 levels of real-time adjustment.
Note: The actual output of the analog voltage value = the analog voltage value calculated by the parameters × spindle magnification

7.2 DNC runs

The CNC has a DNC function, and can be connected to the CNC through DNC communication software or USB flash drive to achieve the operation of large-capacity programs.

Press the machine panel DNC key to enter the DNC mode, press the DNC transmission when the PC side is ready, and then press key 

or key 

on the CNC terminal to receive data, and then press the machine panel cycle start key to start the program DNC machining. The specific operation method can be found in the introduction of DNC communication software

7.3 Runtime State

7.3.1 Single-segment operation

When executing a program for the first time, to prevent accidental programming errors, you can choose a single-segment run.

In the automatic operation mode, the method of opening the single-segment program switch is as follows:

Press the 

key to light up the single-segment operation indicator in the status indicator area  , indicating that the single-segment operation function is selected;

When a single segment is running, the CNC stops running after executing the current program segment; When proceeding to the next program segment, you need to press the 

key again, and so on until the program finishes running.

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

Note 2: The single program segment that executes the call subroutine (M98) and the subprogram call return code (M99) is invalid. However, in **the M98 and M99 program segments, except for** other addresses other than **N, O, P, the single segment stop is valid.**

7.3.2 Null operation

7.3.3 The machine is locked for operation

Under the automatic operation mode, the method of machine tool locking switch is as follows:

Press the 

button to make the machine tool lock running indicator light in the status indication area  , indicating that the machine tool is locked and running;

Machine locking runs are often used for program calibration together with the accessibility locking function. When the machine is locked:

1) The machine drag plate does not move, the "machine coordinates" in the comprehensive coordinate page under the location interface do not change, and the relative coordinates, absolute coordinates and residual movement display are constantly refreshed, the same as

when the machine tool locks the switch in the off state;

2) M, S, T code executes normally.

7.3.4 Accessibility Locking Operation

Under the automatic operation mode, the method of machine tool locking switch is as follows:

Press the 

key to make the auxiliary function lock operation indicator light in the status indicator area 

, indicating that the auxiliary function is locked and running;

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

Note: Accessibility locking does not affect the execution of M00, M30, M98, M99 when active.

7.3.5 Program section selection

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

In the automatic operation mode, the method of opening the program section jump switch is as follows:

Press the 

key to make the program section jump indicator light in the status indicator area 

illuminate;

Note: When the program section selection switch is not turned on, the program **segment with a "/" sign at the beginning of the program segment will not be skipped in automatic operation, and it will be executed as usual.**

7.4 Other Operations

1、 Automatic operation mode, press key 

, coolant on/ off switching;

2、 Press any key in the 

, 

, 

, 

, 

, 

or 

key to achieve the conversion of the mode of operation;

3、 Press 

key to reset CNC.

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

Chapter 8 Zero-Back Operation

The function of the keys 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 manual of the machine tool manufacturer.

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

8.1 Machine zero return

8.1.1 Machine zero point

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 return switch on the installation machine tool, usually the zero point switch or return zero switch is installed in the X axis and the maximum stroke of the positive direction of the Z axis.

8.1.2 The operation steps of the machine tool to zero back

1、 Press the 

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

2、 Press the 

, 

, 

or 

key to select the zero point of the X, Y, Z or 4th axis machine;

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 being detected by the deceleration signal and the zero point, at which time the axis stops moving, and the indicator light at the end of the return to zero is on.

Indicator indicating the end of return to zero

Note 1: If the CNC machine tool does not have the machine tool zero point installed, the machine tool shall not be used to zero operation;

Note 2: The zero-back end LED turns off in the following cases:

1) Move out of the zero point;

2) CNC power outage;

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

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

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

8.2 Other operations in zero-back mode

1、 Buttons 

, spindle turn counterclockwise;

2、 Button 

, spindle stop;

3、 Buttons 

, spindle clockwise;

4、 Button 

, coolant on/ off switch;

5、 Lubrication control (see Chapter 3 of this Part for details);

6、 Buttons 

, manual loosening / clamping knife;

7、 Revision of spindle magnification;

8、 Fast multiplier adjustment;

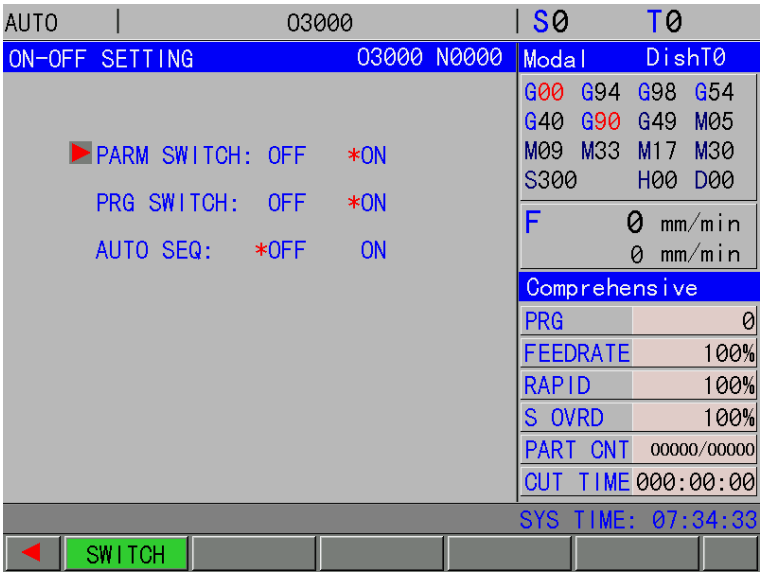
9、 Correction of feed magnification.

Chapter 9 Setup, Backup, and Recovery of Data

9.1 Settings for Data

9.1.1 Switch settings

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



1、 Press the SET key to enter the setting interface, press the SET key or [soft function] key to enter the switch setting page;

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

3、 Press  and  key toggle switch state, press  button, "*" move left, turn off switch, press  button, "*" move right, turn on switch;

Parameters can only be modified when the parameter switch is on; The program can only be edited when the program switch is on; Only when the Auto Sequence Number switch is turned on, the program segment sequence number is automatically added when the program edits.

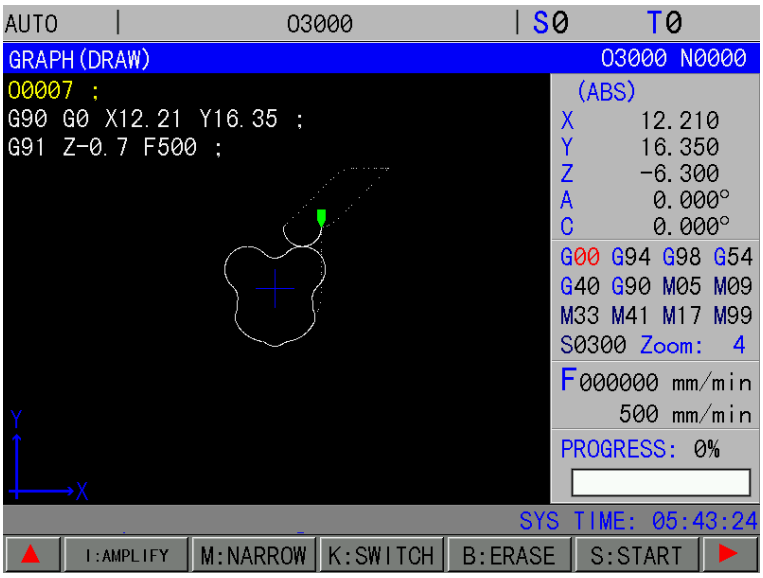
Note: When the parameter switch is switched from "off" to "on", the CNC will appear alarm, first press 

and hold the 

key **and then hold the key to eliminate the alarm, if you switch the parameter switch state, then do not alarm. For safety reasons, after the parameter modification is completed, be sure to set the parameter switch to "off".**

9.1.2 Graphics Settings

Press the panel [graphic button] to enter the graphical interface, you can enlarge, zoom out, clear and other operations of the graph; You can also enter the graphical interface through the function key, and find the function key [graphic] in the location interface ▸.



The meaning of the graphical parameter

Coordinate system settings: according to different coordinate display can switch multiple graphics direction;

1) Zoom in and out of graphics tracks

On the graphical display page, you can zoom in and out of the graphics track in real time by editing the keyboard 

, 

or the [function key to zoom in/out] key.

2) Graph track display starts, stops, and clears

On the graphical track display page, press the S key once or the [Function Key Start] key to start the drawing; Press the T key once or [Function Key Stop] key to stop drawing; Press the J key once or the Function Key to clear the current graphics track.

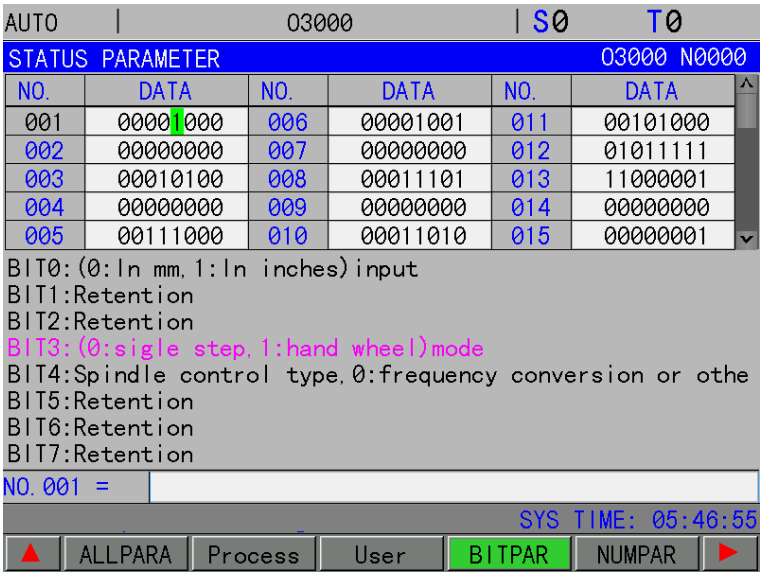
3) The movement of the graphic track is displayed

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

9.1.3 Setting parameters

With parameter setting, the relevant characteristics of the drive unit, the machine, etc. can be adjusted. The significance of each parameter is detailed in Appendix I

Press the PAR key to enter the parameter interface, press or key to switch the parameter page, as shown in the following figure:



A. Bit parameter modification settings

1、 Byte modification:

1) Turn on the parameter switch;

2) Select the input method;

3) 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, 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 or 

key.

4) Enter the new parameter values;

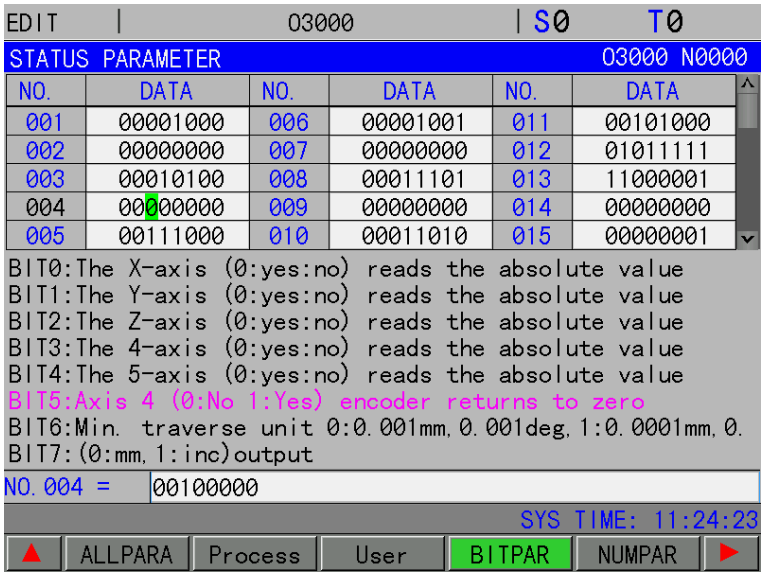
5) Press a key 

or 

key, the parameter value is entered and displayed;

6) For safety reasons, after all the parameters have been set, the parameter switch needs to be turned off.

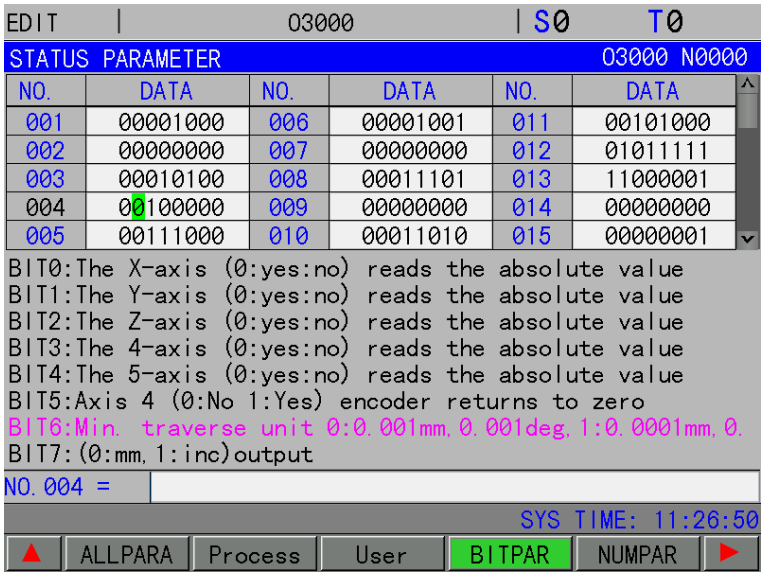
Example: Set the state parameter No 004 to Bit5 to 1 and leave the rest unchanged. Follow these steps to move your cursor over No 004 and type 0 0 100000 in the prompt line, as shown in the following figure:



Press the 

key or 

key, and the parameter modification is complete. The display page is as follows:



2、 Modify bits:

1) Turn on the parameter switch;
2) Select the input method;
3) Move the cursor over the parameter number to be set;

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

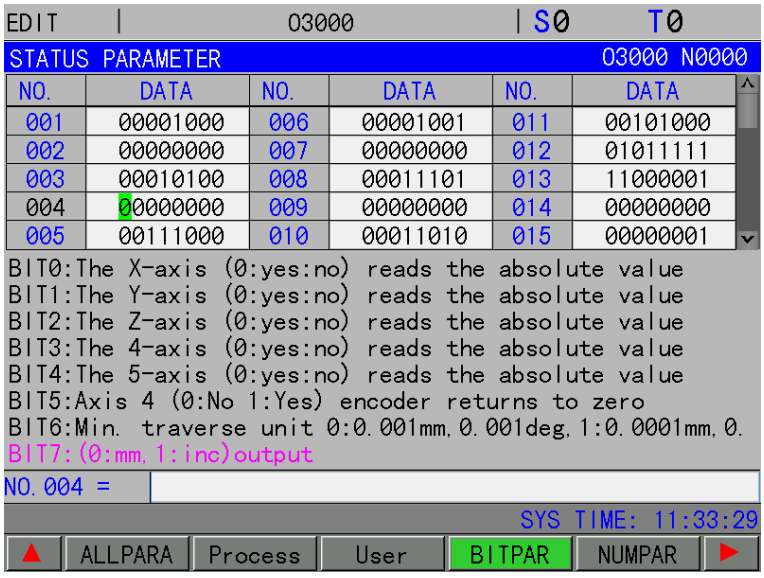
Method 2: Press the address key  , parameter number, and  key or  key.

4) Press  to jump into one of the bits of the parameter, which is inverted. Press  or  key to move the cursor to the bit you want to modify, type 0 or 1 as needed;
5) For safety reasons, after all the parameters are set, the parameter switch needs to be turned off;

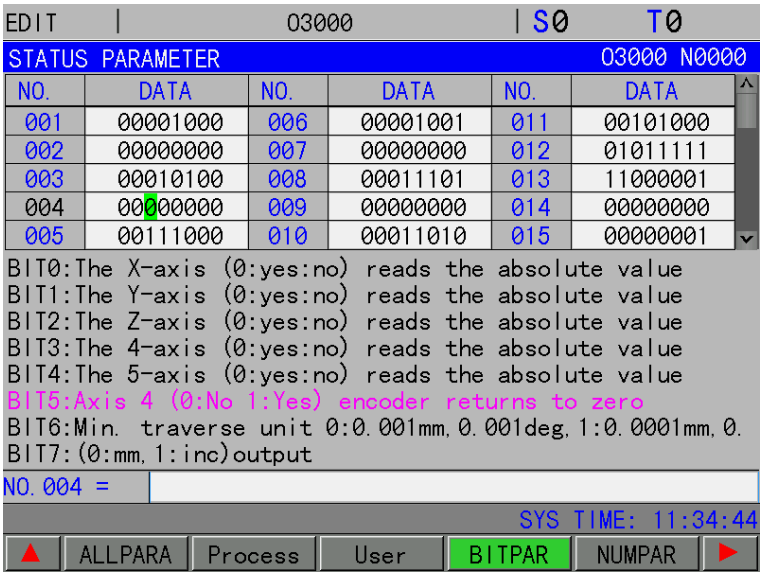
Note: After entering a certain bit of the parameter, press the  key to jump out of the bit to enter the parameter number.

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

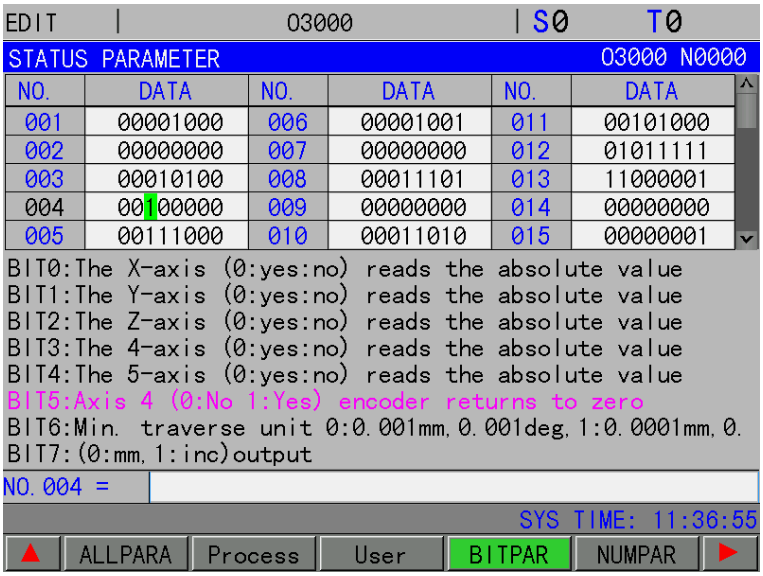
Follow these steps to move the cursor over No 004 and press  Step into one of the parameters. As shown in the following figure:



Press  or  move the cursor onto the Bit5 bit, as shown in the following figure:



Enter 1 to modify the parameters.



B. Modification settings of data parameters and screw supplement data

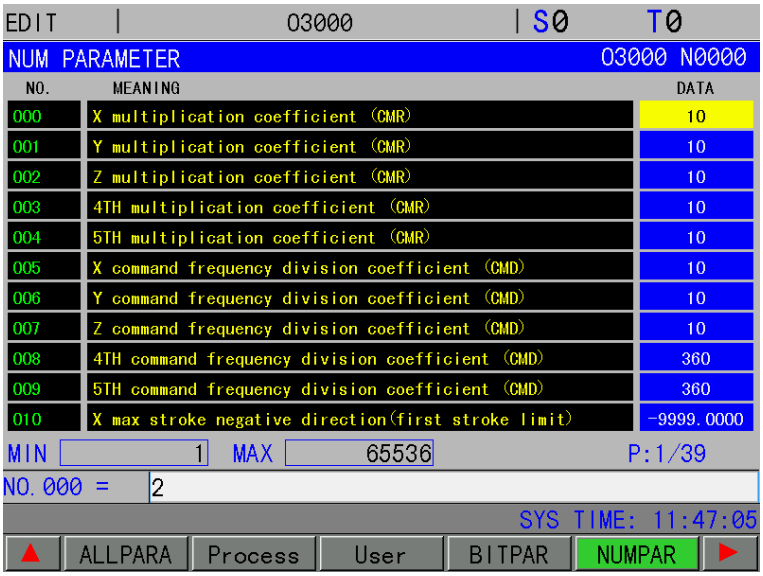
1) Turn on the parameter switch;
2) Select the input method;
3) Move the cursor over the parameter number to be set;
4) Enter the new parameter values;
5) Press a key 

or 

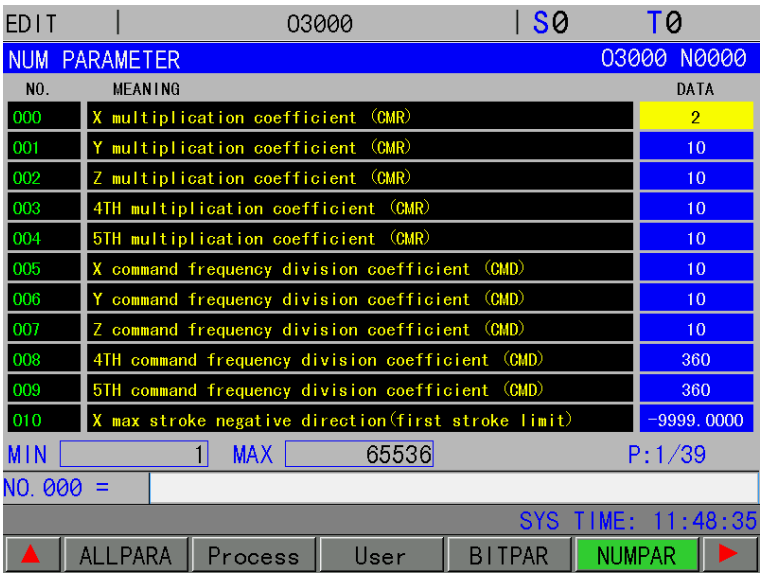
key, the parameter value is entered and displayed;
6) For safety reasons, it is recommended to turn off the parameter switch after all the parameters have been set.

Note: Screw compensation data can only be modified under the second-level operation permission.

Example 1: Set the data parameter No 000 to 2. Follow these steps to move the cursor over No 000 and type 2 in the prompt line, as shown in the following figure:

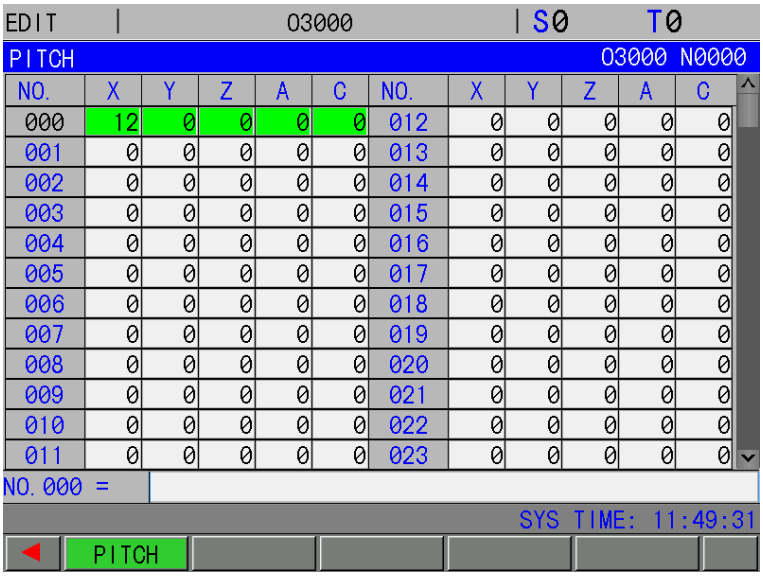


Press the IN key or Enter key, and the parameter modification is complete. The display page is as follows:

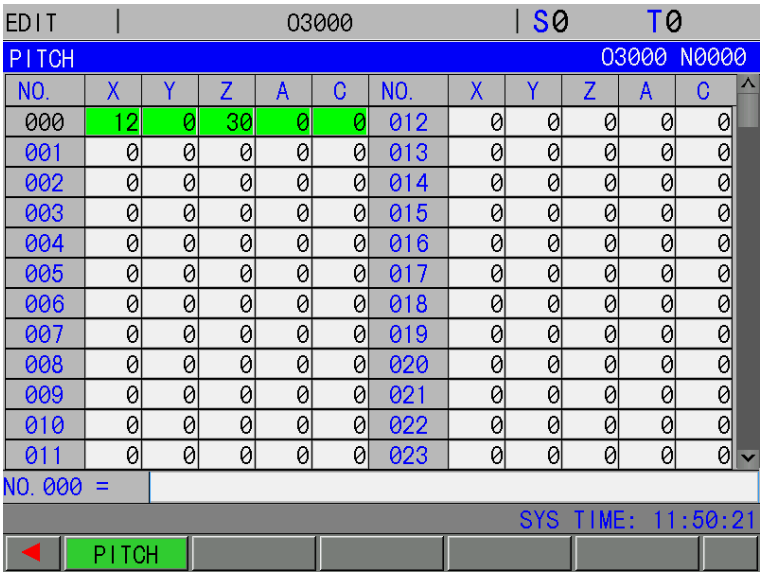


Example 2: Set the value of the X axis of the screw data No 000 to 12 and the value of the Z axis to 30. Follow these steps to move the cursor over the patch data No 000 and type X12 in the prompt row.

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

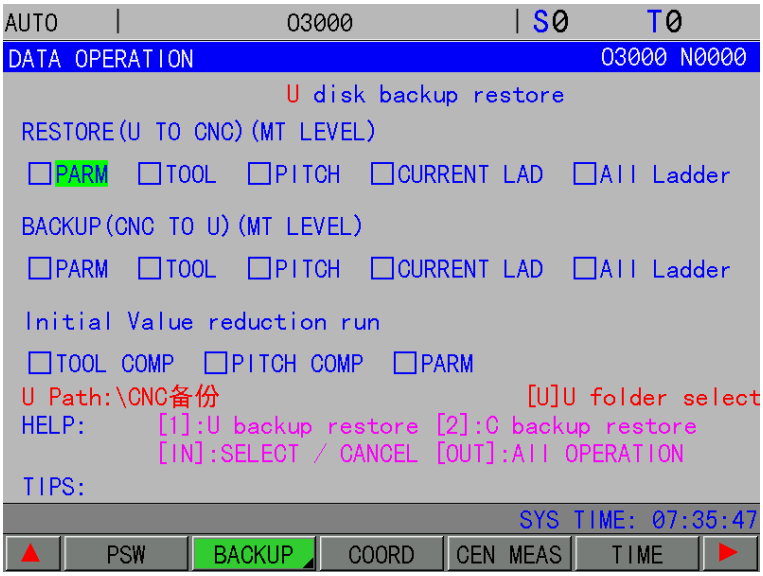


Similarly, type Z30 in the prompt line in turn, press the IN key or Enter key, and the data modification is complete. After the modification, the page is displayed as follows:



9.2 Data Restore 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). Backup and restore of data without affecting part programs stored in the CNC. The Data Operations page appears as follows:



1、 Turn on the parameter switch;

2、 Press the MDI key to enter the input operation mode, press the SET key (press or key if necessary) to enter the data operation page;

3、 Press the 2 key to enter the C disk backup and restore page, and press the 1 key to enter the U disk backup and restore page.

4、 Move the cursor to the option you want to operate, press the IN key or Enter key to select/cancel the operation option;

5、 Press the OUT key to perform all selected actions.

Note 1: When performing data backup and restore operations, please do not power off, and it is recommended not to perform other operations until the prompt operation is completed;

Note 2: Users at level 3 operation and above can back up and restore status parameters, data parameters and screw compensation parameters.

9.3 Setting and Modifying Permissions

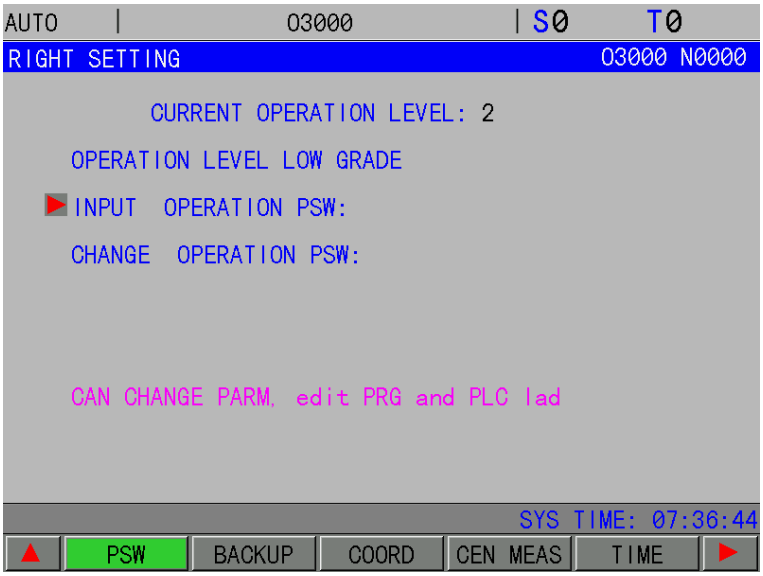
In order to prevent the machining program and CNC parameters from being maliciously modified, the system has raised the permission setting function, and the password level is divided into 4 levels, from high to low, it is 2 levels (machine tool manufacturer level), 3 levels (equipment management level), and 4 levels Level (Technician level), Level 5 (Machining Operation Level), the current operation level of the CNC is displayed by the "Current Operation Level:" on the permission settings page.

Level 2: Machine tool manufacturer level, allowing modification of CNC status parameters, data parameters, screw compensation data, tool repair data, editing part programs, transmission PLC ladder diagram, etc.

Level 3: The initial password is 12345, allowing modification of CNC status parameters, data parameters, tool compensation data, and editing part programs.

Level 4: The initial password is 1234, which can modify the tool patch data (tool counterpart operation), macro variables, edit part programs, and cannot modify CNC status parameters, data parameters and screw patch data.

Level 5: 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, can not edit the program, can not modify the CNC status parameters, data parameters and screw compensation data.



When the permission setting page is displayed, the cursor stays in the Enter Operation password line. Press key ⇧ or key ⇩ to move the cursor to the corresponding operation.

1) Press the ⇧ key once and move the cursor up one line. If the current cursor is in the "Operation Level Downgrade" line (the first line), after pressing the ⇧ key once, the cursor moves to the "Change Operation Password" line (the last line);

2) Press the ⇩ key once and move the cursor up one line. If the current cursor is in the trailing line, press the ⇩ key once and move the cursor to the first row.

9.3.1 Operation-level entry

1、 After entering the permission settings page, move the cursor to the "Enter Operation Password" line;

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

3、 Enter the completion button IN or Enter key to enter the operation level corresponding to the password.

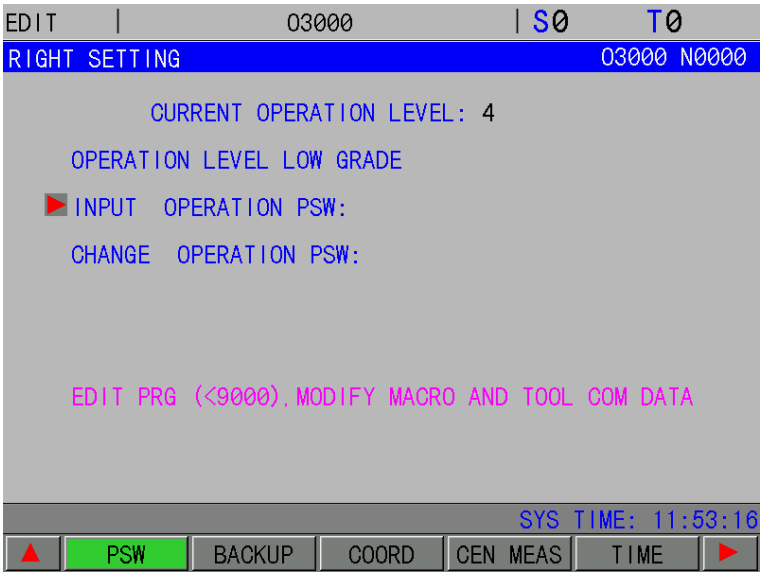
Note: The length of password data **and the operation level defined by the system are corresponding, and users cannot increase or decrease the length of password data according to personal imagination.**

Note: Be careful to use a full numeric password when changing the operation password.

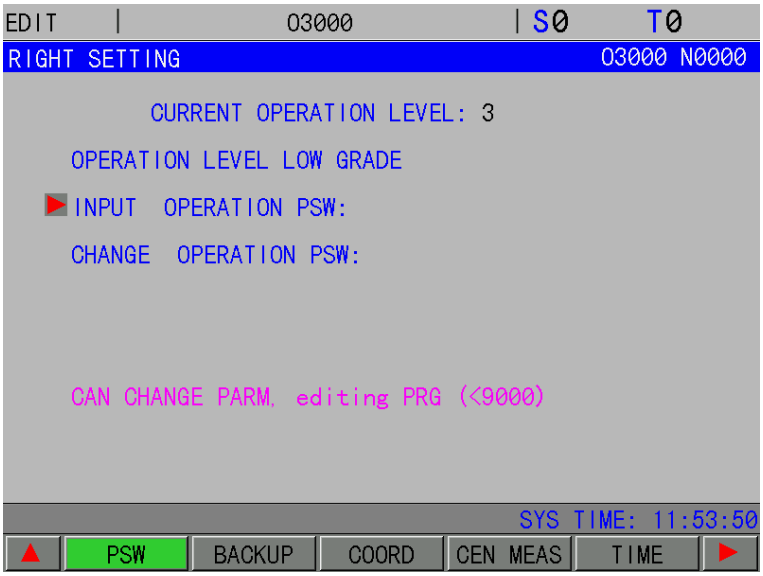
The details are as follows:

The level of	The password data	The initial
Level 3	5 digits	12345
Level 4	4 digits	1234
Level 5	not	not

Example: The current operation level of CNC is level 4, and the display page is as follows. 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 IN or Enter key, CNC prompt "Modifiable parameters, editable program. , "Password Passed", the current operation level is changed to level 3. The display page is as follows:



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

9.3.2 Operational Password Changes

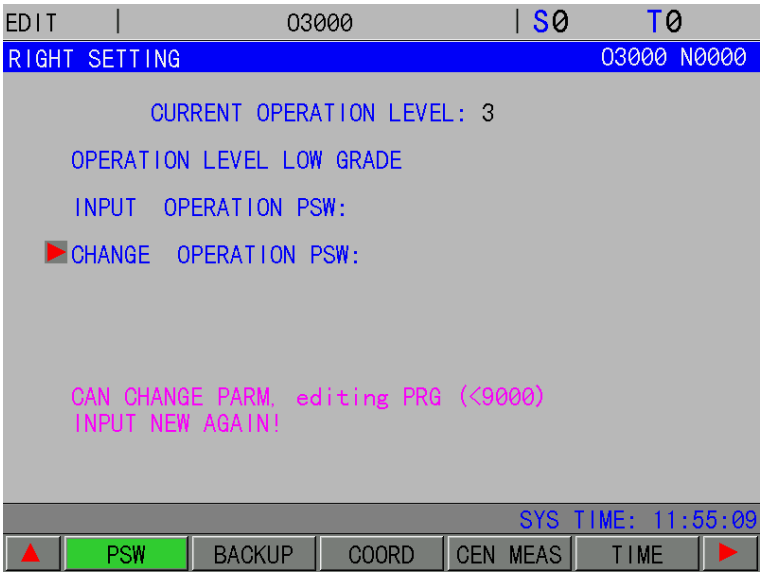
The steps to change the password are as follows:

1、 Once you are on the Permission Settings page, enter your password as described in Section 9.3.1;

2、 Move the cursor to the "Change Operation Password" line;

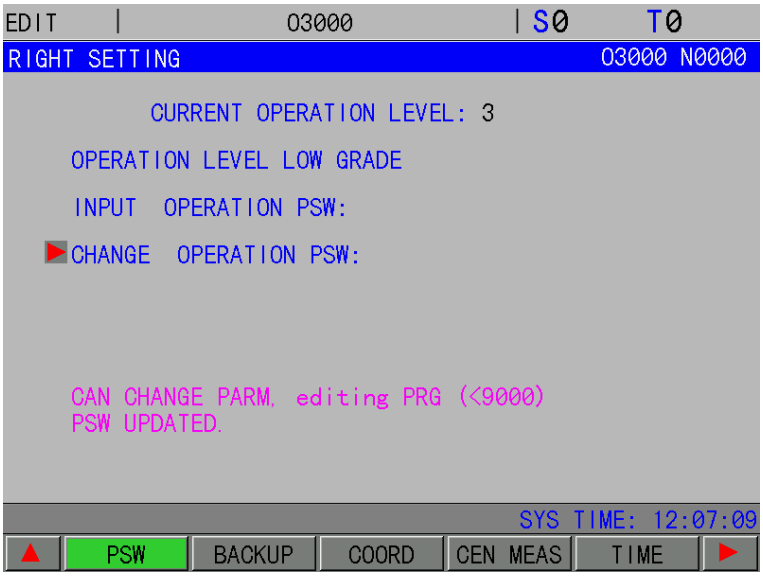
3、 Enter a new operation password, key IN or Enter key;

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

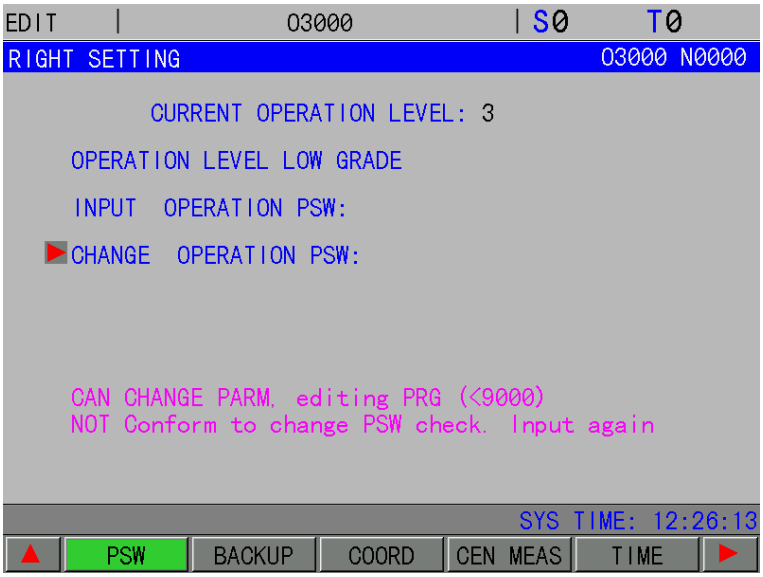


5、 After entering the operation password again, press the IN key or Enter key, if the

password entered twice is the same, the CNC prompts "password has been updated", and the operation password is changed successfully.



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

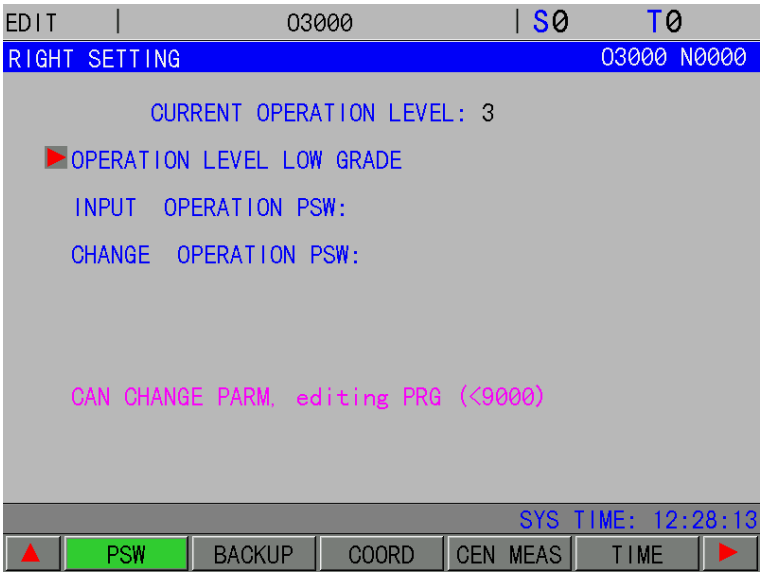


9.3.3 Operation Level Downgrade

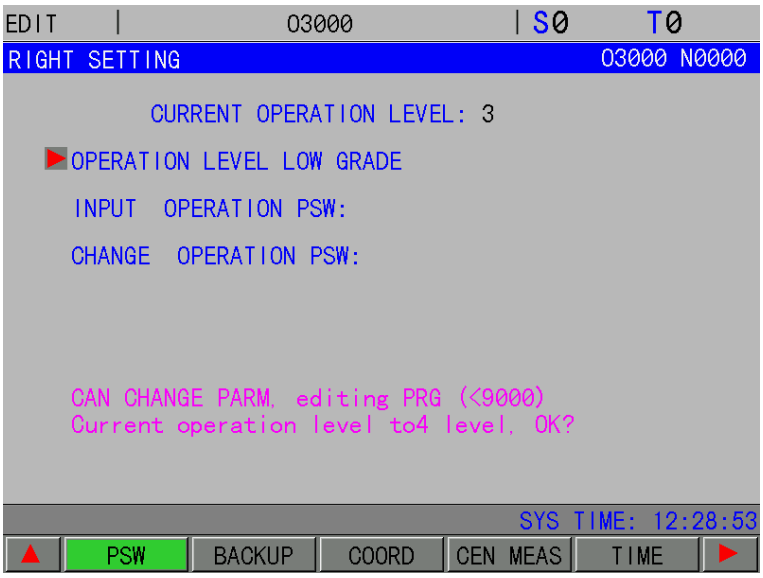
Lowering the operation level allows the user to reduce the operation level from the higher level to the lower operation level as follows:

1、 Once you are on the Permission Settings page, enter your password as described in Section 9.3.1;

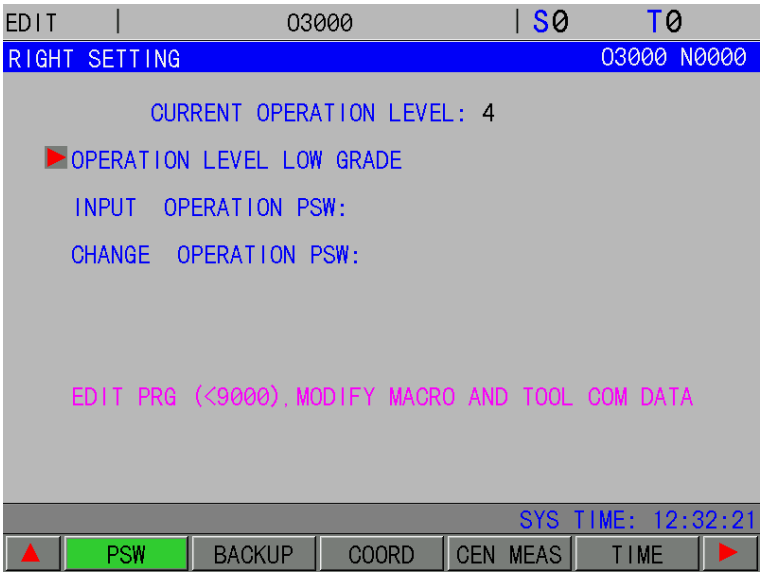
2、 Move the cursor to the Action Level Downgrade row, and if the current CNC action level is Level 3, the page is displayed as follows:



3、 Pressing the IN key or Enter key, the CNC prompts "The current operating level is reduced to 4 levels, OK? ”； The display page is as follows:



4、 If you press the IN key or Enter key again, the operation level is demoted successfully, and the page is displayed as follows:



Note: If the current operation permission is already level 5 , the operation level cannot be downgraded.

Chapter 10 U disk operation function

10.1 File Directory Page

In the non-editing state, press the PRG key to enter the program page, press the key to enter the [file directory] interface, insert the USB flash drive as shown in the figure:

The top of the page displays CNC disk directory information. The USB flash drive directory information is displayed below, if the USB flash drive is not detected, the right display bar does not display the content.

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

The key CHG or soft function key cursor switches from the CNC drive to the USB drive, and the key ⇧ or ⇩ key moves the cursor.

10.2 File Copying

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