

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

The copyright of this user manual belongs to the company, and it is illegal for any unit and individual to publish or copy, and the company reserves the right to pursue its legal responsibility.

preface

Dear Customer:

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

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

Security warnings



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

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

Notes

■ Transportation and storage

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

■ Unpacking inspection

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

■ Wiring

1. The personnel participating in the wiring and inspection must be professionals with corresponding capabilities;
2. The product must be reliably grounded, the grounding resistance should be less than 0.1Ω , and the neutral wire (neutral wire) cannot be used instead of the ground wire;
3. The wiring must be correct and firm to avoid product failure or unexpected consequences;
4. The surge absorbing diode connected to the product must be

connected in the specified direction, otherwise the product will be damaged;

5. Before plugging in and unplugging the plug or opening the product chassis, the product must be cut off.

■ Overhaul

1. The power supply must be cut off before repairing or replacing components;

2. When there is a short circuit or overload, the fault should be checked and restarted after troubleshooting;

3. Do not power off the product frequently, if you need to re-energize after the power off, the interval is at least 1min.

statement

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

warn

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

scrapping of the workpiece and even personal injury.

note

The product functions and technical indicators (such as accuracy, speed, etc.) described in this manual are only for this product, the CNC machine tool installed in this product, the actual functional configuration and technical performance are determined by the design of the machine tool manufacturer, and the functional configuration and technical indicators of the CNC machine tool are subject to the instruction manual of the machine tool manufacturer;

Although the system is equipped with a standard machine operator panel, the functions of the buttons of the standard machine panel are defined by the PLC program (ladder diagram). Please note that the functions of the buttons in the machine panel in this manual are described for standard PLC procedures!

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

Part II Operating Instructions

Introduce the operation and use of the lathe system.

Security responsibility

Manufacturer's responsibility for safety

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

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

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

User's responsibility for safety

- The user should learn and train the safe operation of the numerical control system, and be familiar with and master the content of the safe operation.

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

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

***This manual is for end-user collections. Sincerely thank you for your friendly support when using our products!**

directory

Chapter 1 Operation mode and display interface	11
1.1 Panel division	11
1.1.1 Status indication	12
1.1.2 Edit the keyboard	12
1.1.3 Display menu	14
1.1.4 Machine panel.....	15
1.2 Overview of the mode of operation.....	18
1.3 Display interface	19
1.3.1 Location interface.....	20
1.3.2 Program Interface.....	23
1.3.3 Tool length, wear, macro variable interface, system variables.....	26
1.3.4 Alarm interface.....	28
1.3.5 Settings interface.....	29
1.3.6 Status parameters, data parameters, processing technology, user parameter interface	33
1.3.7 CNC diagnostics, PLC signals, IO diagnostics, version information, alarm interface	35
1.3.8 Graphical Interface	37
Chapter II: Power On, Shutdown and Safety Protection	38
2.1 Boot.....	38
2.2 Shutdown	39
2.3 Overtravel Protection	40
2.3.1 Hardware Overtravel Protection	40
2.3.2 Software overtravel protection	40
2.4 Emergency Operations	41
2.4.1 Reset.....	42
2.4.2 Emergency Stop.....	42
2.4.3 Feed hold.....	42
2.4.4 Cut off the power supply.....	43
Chapter III: Manual Operation	43
3.1 Axis movement.....	43
3.1.1 Manual feeding	43
3.1.2 Manual fast movement	44
3.1.3 Speed Trimming	45
3.2 Other manual operations	46
3.2.1 Turn counterclockwise, turn clockwise, stop control	46
3.2.2 Spindle jogging	46
3.2.3 Coolant Control	47
3.2.4 Lubrication control	47
3.2.5 Manual tool change	49
3.2.6 Tuning of spindle magnification	49
Chapter 4 Handwheels/Manual Operation	49

4.1 Manual feeding	49
4.1.1 Selection of increments.....	50
4.1.2 Movement direction selection	50
4.2 Handwheel (Hand Crank Pulse Generator) feed	50
4.2.1 Selection of increments.....	51
4.2.2 Selection of movement axis and direction	51
4.2.3 Other Operations	51
4.2.4 Descriptions.....	52
Chapter V: Input Operations	53
5.1 Code Word Input.....	53
5.2 Code Word Execution.....	54
5.3 Setting of Parameters.....	54
5.4 Modification of Data	55
5.5 Other Operations	55
Chapter VI: Program Editing and Management	56
6.1 Establishment of Programs	56
6.1.1 Generation of program segment numbers	56
6.1.2 Input of program content.....	57
6.1.3 Cursor movement.....	58
6.1.4 Search for words and line numbers	59
6.1.5 Insertion of words	61
6.1.6 Deletion of Words	61
6.1.7 Modification of Words	61
6.1.8 Deletion of Single Program Segments.....	62
6.1.9 Deletion of Multiple Program Segments.....	62
6.1.10 Block Deletion	62
6.1.11 Replication of Single Program Segments.....	62
6.1.12 Replication of multiple program segments	62
6.1.13 Copying of Blocks	63
6.1.14 Pasting of program blocks	63
6.2 Deletion of Programs	64
6.2.1 Deletion of Individual Programs.....	64
6.2.2 Deletion of All Programs	65
6.3 Selection of Programs	65
6.4 Renaming of the Program	65
6.5 Reproduction of Programs	66
6.6 Program Management	67
6.6.1 Program Directory	67
6.6.2 Number of stored programs.....	68
6.6.3 Storage Capacity.....	68
Chapter 7 Tool to tool and wear compensation.....	68
7.1 Tool to Tool	68
7.2 Wear compensation	71
Chapter VIII Automatic Operation.....	73

8.1 Auto-Run	73
8.1.1 Selection of Runners	73
8.1.2 Startup of Autorun	74
8.1.3 Stop of Auto-Run	74
8.1.4 Run automatically from any segment	76
8.1.5 Feed, fast speed adjustment	77
8.1.6 Spindle speed adjustment	77
8.2 Status at runtime.....	78
8.2.1 Single-segment operation	78
8.2.2 Empty run.....	79
8.2.3 The machine is locked for operation	79
8.2.4 Accessibility locks operation	80
8.2.5 Program segment selection.....	80
8.3 Other Operations	81
Chapter 9 Zero Return Operation.....	81
9.1 Program Returns to Zero	81
9.1.1 Program Zero	81
9.1.2 Procedure for program zeroing	81
9.2 Machine tool returns to zero	82
9.2.1 Machine tool zero	82
9.2.2 Operation steps of machine tool zeroing	83
9.3 Other operations in zero-back mode	84
Chapter 10 Data Setting, Backup and Recovery	84
10.1 Setting of Data.....	84
10.1.1 Switch Settings	84
10.1.2 Graphics Settings.....	85
10.1.3 Setting Parameters	86
10.2 Data Restoration and Backup	92
10.3 Setting and Modifying Permissions	93
10.3.1 Entry to the Operation Level	95
10.3.2 Change of Operation Password	96
10.3.3 Downgrading the Operation Level	98
Chapter 11 U disk operation functions	100
11.1 File Directory Page	100
11.2 File Copying	101
Chapter 12 Processing Examples.....	101
12.1 Programming.....	102
12.2 Program Inputs.....	103
12.2.1 Viewing Existing Programs	103
12.2.2 Establish new procedures	104
12.3 Program Verification	105
12.3.1 Graphics Settings.....	105
12.3.2 Program Validation.....	107
12.4 Tooling, Running and Wear	107

Chapter 1 Operation mode and display interface

1.1 Panel division

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

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

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

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

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



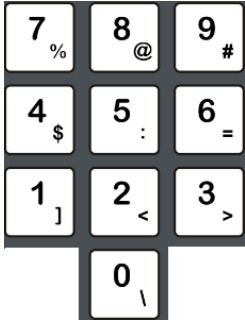
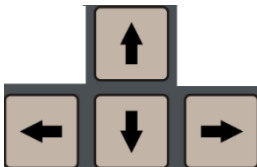
Panel division

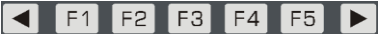
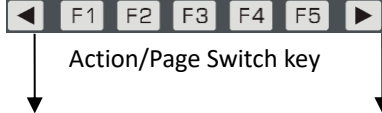
Note: Some system models have different button placement positions.

1.1.1 Status indication

	Shaft zero end indicator
	Tricolor lamp

1.1.2 Edit the keyboard

keystroke	name	Function description
	Reset key	CNC reset, feed, output stop, etc
	Address key	Address input
		Double address keys, press keys repeatedly, and switch between the two
	Symbol keys	Two address keys, repeatedly press the key, switch between the two
	Number keys	Digital input
	Delete key	Deletion of programs, fields, etc. when editing
	Cursor movement keys	Controls cursor movement
	Page turn key	Switch between pages on the same display screen

	Function keys	After using the function keys to switch the page set, you can use the corresponding software function keys to display the content of a sub-page in the current page, as shown below:  Return to the parent menu Move the menu key right The role of soft function keys: (1) Switch subpages in the current page set; (2) As an operation input for the currently displayed subpage, such as editing, modifying data, or displaying content, etc.
Some of the following button models have		
	Enter	Determination of data input such as parameters and compensation quantities; input of the end-of-program segment; Note: Models with a carriage return key do not have an input key, and the carriage return key is used as an input and line feed key
	The first and last row keys	In the program interface, you can switch to the beginning and end of the program
	Up key	Some model numbers and symbols are composite keys, and you can switch symbols after pressing the upper gear key
	Space bar	

1.1.3 Display menu

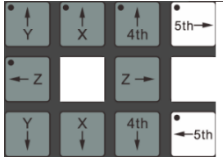
Menu key	remark
	Enter the location interface. The location interface has three surfaces: relative coordinates, absolute coordinates, and comprehensive coordinates
	Enter the program interface. The program interface has file management, teaching, deleting lines, replacing, etc
	Enter the interface of knife length, wear, tool tip, macro variables, etc., and repeatedly press keys to switch between interfaces. The wear interface shows tool offset wear; The macro variables interface displays CNC macro variables
	Enter the program supervision interface. The program monitoring interface loads the program, graphics, MDI, information settings, wear and tear pages, etc.; The graphical interface shows the movement trajectory of the feed axis
	Enter the CNC diagnosis interface, PLC status, PLC data, machine tool soft panel, version information interface. Repeated keystrokes can be used to switch between interfaces. CNC diagnostic interface, PLC status, PLC data display CNC internal signal status PLC all over the place Information on the status of address and data; The soft panel of the machine tool can be operated by the soft keyboard of the machine tool; The version information interface displays the version number of CNC software, hardware and PLC

1.1.4 Machine panel

The function of the buttons in the machine tool panel is defined by the PLC program (ladder diagram), and the specific function significance of each button is please refer to the manual of the machine tool manufacturer.

The functions of the buttons of the machine panel defined by the standard PLC program of the lathe series are shown in the following table

keystroke	name	Function description	Mode of operation when the
	Feed hold button	Program and MDI code are suspended	Automatic method, input method, DNC method
	Cycle the start button	The program and MDI code run and start	Automatic method, input method, DNC method
	Feed multiplier key	Adjustment of feed rate	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Quick magnification key	Adjustment of fast movement speed	Automatic mode, input mode, machine tool zeroing, manual mode, DNC mode
	Spindle magnification key	Spindle speed adjustment (spindle speed analog control mode is effective)	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Manual tool change	Manual tool change	Handwheel method, single-step method, manual mode
	Toggle the key	Spindle jogging status on/off	Machine tool zeroing, handwheel mode, single-step mode, manual mode
	Lubricate the key	Machine lubrication on/off	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	The coolant switch is on	Coolant on/off	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode

	Spindle control keys	The spindle rotates forward The spindle stops The spindle is reversed	Automatic mode, input mode, editing mode, machine tool zeroing mode, handwheel mode, single-step mode, manual mode, DNC mode
	Quick switch	Fast speed/feed rate switching	Automatic mode, input method, manual method, DNC method
	X-axis feed key	Manual, single-step operation mode of forward/negative movement of each axis	Machine tool zeroing, single-step mode, manual mode
	Y-axis feed key		
	Z-axis feed key		
	4th axis feed key		
	5th axis feed key		
	Handwheel/step increment selection with quick magnification selection key	Handwheel movement per cell 1/10/100/1000 * Minimum equivalent single step movement 1/10/100/1000 * Minimum equivalent fast magnification F0, 25%, F50%, F100%	Automatic mode, input mode, machine tool zeroing method, handwheel mode, single step mode, manual mode, DNC mode
	Select Stop	When you select Stop is active, perform M01 Pause	Automatic method, input method, DNC method
	Single-stage switch	The program single-stage operation/continuous operation status is switched, and the single-stage operation indicator light is on when the single-stage is active	Automatic method, input method, DNC method
	Program segment selector switch	Whether the program segment marked with "/" at the beginning of the program segment skips the state switch, and the jump indicator light is on when the program segment selection switch is turned on	Automatic method, input method, DNC method

	The machine locks the switch	When the machine is locked, the machine lock indicator is on, and the feed shaft output is invalid	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Accessibility locks the switch	When the auxiliary function is locked, the auxiliary function lock indicator is on, and the output of the M, S, and T functions is invalid	Automatic method, input method, DNC method
	C/S button	Spindle speed/position mode switching	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Work light switch	Turn the work light on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Chip conveyor switch	Turn the chip conveyor on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Custom output switches	Turn custom output on or off	Automatic method, input method, editing method, Machine tool zero return, handwheel mode, single step mode, manual mode, DNC mode
	Handwheel simulation development	Turn handwheel simulation on or off	Automatic method, input method, DNC method
	Automatic mode selection keys	Enter the automatic operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Select the key for the entry method	Enter the entry (MDI) operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Machine zero mode selection key	Enter the machine to return to zero operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method

	Single-step/handwheel selection key	Enter single-step or handwheel operation mode (one of the two operation modes is selected by the parameters)	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Select keys manually	Enter the manual operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Program zero-point mode selection key	Enter the program zero point operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Tailstock switch	Turn on/off the tailstock function	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method
	Chuck switch	Turn on/off the chuck function	Automatic mode, input mode, machine tool zeroing, Handwheel method, single-step mode, manual mode, DNC method

1.2 Overview of the mode of operation

The lathe series system has seven operation modes, such as editing, automatic, input, machine tool zeroing, single-step / handwheel, manual, and program zeroing.

- Edit the mode of operation

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

- Automatic operation mode

In automatic operation mode, run the program automatically.

- Input operation mode

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

- Machine tool return zero operation mode

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

- Handwheel/ single step operation mode

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

- Manual operation mode

In the manual operation mode, manual feed, manual fast, feed rate adjustment, fast magnification adjustment and spindle start and stop, coolant switch, lubricant switch, spindle jogging, manual tool change and other operations can be carried out.

- Program return to zero operation mode

In the program return zero operation mode, the feed axis back program zero operation can be performed separately.

1.3 Display interface

The lathe series system 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 figure below

Menu	Display the	Display the page
	Location interface	Coordinate switching (machine coordinates→ relative coordinates→ absolute coordinates→ residual movement), 1/2 coordinates (U-axis 1/2, W-axis 1/2), relative clearing , relatively clear
	Program interface	Perform machining, file management, find/replace, delete rows, teach, region copy, jump
	Compensation interface	Knife length, wear, tip
	Macro variable interface	Macro variables, system variables
	CNC diagnosis	CNC diagnostic interface

	PLC diagnosis	PLC signal diagnostic interface
	IO diagnosis	Input/output diagnostic interface
	Version information	The system software version is displayed
	alarm	Switch to the alarm interface
	Spindle detection	Detect the spindle reaction time interface
	absolute value	Absolute values drive the monitoring interface
	Zero point detection	One turn signal back to zero monitoring interface
	CNC Help	Each interface is: operation table, alarm table, G code table, M code table, macro command
	Extended control	

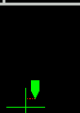
1.3.1 Location interface

Press the  key to enter the position interface, the location interface has four pages: absolute coordinates, relative coordinates, comprehensive coordinates and program supervision,

which can be viewed by  keys or soft function keys [F1-F5].

1) Absolute coordinates display the page

The displayed X, Z coordinate values are the absolute position of the tool in the current workpiece coordinate system, the X, Z coordinates are maintained when the CNC is powered on, and the workpiece coordinate system is determined by G50, G54-G59 specified.

G54		0008 N0000		MONITOR		2000/08/22 13:07:14	
(ABSOLUTE)		(REM DIST)		G Code G00		Pro Time 000:00:00	
X	-24.812	X	0.000	G18 G97 G98	All Time 000:00:00		
Z	16.406	Z	0.000	G21 G40 G54	G00Rate 100 %		
				M05 M09 M33	G01Rate 0 %		
				M41 M11 M30	SPI Rate 100 %		
F 0mm/min S 300 RPM		300 RPM		Workeds 0		T 0100	
(True) 0mm/min (True)		0 RPM		All Num 0		Start In 1	
00008 ;							
S0 M3 ;							
G1 X0 Y0 Z0 F3000 ;							
M3 S600 ;							
M8 ;							
Ln:1							
				●Ready		JOG	
				ALARM			
Load edit		GRAPH		MDI		Info/Set	
						Wear set	

Fast magnification: display the current fast magnification, selected by the panel F0 to 100% corresponding to the button;

Feed rate: the magnification selected by the feed rate switch;

G function code: modal values of group 01 G code and 03 group G code;

Number of processed pieces: when the program finishes M30 (or M99 in the main program), the number of processed pieces is increased by 1;

Cutting time: when the automatic operation starts the timing, the time unit is hours, minutes, seconds;

Spindle magnification: When the Bit4 bit of parameter NO.001 is set to 1, the spindle magnification is displayed;

S0000: spindle speed feedback from spindle encoder, spindle encoder must be installed to display the actual speed of spindle;

T0100: Current tool number and tool offset number.

2) Relative coordinates display the page

The displayed U and W coordinate values are the coordinates of the current position relative to the relative reference point, and the U and W coordinates are maintained when the CNC is powered on. U and W coordinates can be cleared at any time. After the U and W coordinates are cleared, the current point is the relative reference point. When Bit1=1 of CNC parameter No.005, use G50 to set the absolute coordinates, U, W Same as the absolute coordinate value set.

G54	0008 N0000		COORD		2000/08/22 13:41:03	
RELATIVE					(ABSOLUTE)	
U -24.812					X	-24.812
W 16.406					Z	16.406
					(REM DIST)	
					X	0.000
					Z	0.000
					(MACHINE)	
F 0 mm/min 0%		S 300RPM 100%		X	-24.812	
0 mm/min (True)		0 RPM (True)		Z	16.406	
Pro Time 000:00:00		Worker 0/0		T0100		
				●Ready JOG		
				ALARM		
Switch		1/2Coord		REL CLEAR		
				ALL CLEAR		

U, W coordinate zeroing method:

Under the position function card interface, press F1 to switch to the relative coordinate interface, click F3 to enter the relative clearing sub-page of each axis, or click F4 axis for relative total clearance.

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

3) The program supervisor displays the page

On the program monitoring display page, the absolute coordinates of the current position, the relative coordinates (if the Bit0 bit of the status parameter No.180 is set to 1, the absolute coordinates of the current position, the amount of residual movement) and 6 of the current program are displayed at the same time program segments, while the program is running, the displayed program segment is dynamically refreshed, and the cursor is located in the currently running program segment.

G54		0008 N0000		MONITOR		2000/08/22 13:07:14	
(ABSOLUTE)		(REM DIST)		G Code G00		Pro Time 000:00:00	
X	-24.812	X	0.000	G18 G97 G98	All Time 000:00:00		
Z	16.406	Z	0.000	G21 G40 G54	G00Rate 100 %		
				M05 M09 M33	G01Rate 0 %		
				M41 M11 M30	SPI Rate 100 %		
F 0mm/min S 300 RPM		Workeds 0		T 0100			
(True) 0mm/min (True) 0 RPM		All Num 0		Start In		1	
00008 ;							
S0 M3 ;							
G1 X0 Y0 Z0 F3000 ;							
M3 S600 ;							
M8 ;							
Ln:1							
				●Ready		JOG	
				ALARM			
Load edit		GRAPH		MDI		Info/Set	
						Wear set	

1.3.2 Program Interface



Press the  key to enter the program interface, which includes program content, file management (local directory, U disk directory), program editing functions (perform processing, find/replace, area copy, delete line, teach, jump, paste), etc page, press F1-F5 keys repeatedly to switch between pages.

1) Program content page

On the Program Content screen, the program contents including the current program

segment are displayed. Press  keys,  keys forward, and backward to view program content in a non-running state.



2) File Management: Program Directory page



Press the  key to enter the program interface, and then press F2 "File Management" to enter the program directory page. Under this interface, all processing programs are listed, and in order to facilitate users to find the program they want to select, the system displays the first 12 lines of the current program on the right side of the page.

What the program catalog page displays:

a) Number of existing programs:

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

b) Number of programs remaining:

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

c) Amount of storage used:

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

d) Amount of storage remaining:

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

e) Program Directory:

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

f) Program size:

Displays the amount of storage space occupied by the CNC program

G54	0008 N0000	PROGRAM (DIR)	2000/08/22 13:16:00
PRG CNT13	MEM	109 M	USED:0 M FREE:109 M
NAME	SIZE	DATE	PROGRAM PREVIEW:
00001	9 (B)	00-07-27	00008;
(00008) 0008	377 (B)	00-08-22	SOM3;
09101	824 (B)	00-07-27	G1X0Y0Z0F3000;
09102	465 (B)	00-07-27	M3S600;
09103	753 (B)	00-07-27	M8;
09104	5.63 (K)	00-07-27	G1X300Y300Z300;
09105	6.25 (K)	00-07-27	M3S1200;
09106	6.21 (K)	00-07-27	G1X800Y800Z800;
09107	5.70 (K)	00-07-27	M3S2400;
09108	6.27 (K)	00-07-27	G1X1000Y1000Z1000;
09109	5.70 (K)	00-07-27	M3S3600;
09110	6.31 (K)	00-07-27	
09201	3.03 (K)	00-07-27	
TIPS: [Enter] Open [O] New [P] Search [N] Remane [DEL] De l			
		Ready	ALARM
NEW	Copy File	Delete	USB Rename

3) File management: U disk directory page

When on the program directory page, press the F4 "U disk directory" key to enter the U disk directory page. The page is displayed as follows:

G54	0008 N0000	FILE DIR:	2000/08/22 13:46:29
File	SIZE	DATE	
00001.txt	9 (B)	00-07-27	
0008.TXT	377 (B)	00-08-22	
09102.txt	465 (B)	00-07-27	
09103.txt	753 (B)	00-07-27	
09101.txt	824 (B)	00-07-27	
09201.txt	3.03 (K)	00-07-27	
09104.txt	5.63 (K)	00-07-27	
U:\	SIZE	DATE	
TIPS: [ENTER]:open [DEL]:DEL			
		Ready	ALARM
To USB	Switch	LASTLINE	NEXTLINE Copy All

G54	0008 N0000	FILE DIR:	2000/08/22 13:47:20
File	[SPACE]:C/U CHANGE	SIZE	DATE
00001.txt		9 (B)	00-07-27
0008.TXT		377 (B)	00-08-22
09102.txt		465 (B)	00-07-27
09103.txt		753 (B)	00-07-27
09101.txt		824 (B)	00-07-27
09201.txt		3.03 (K)	00-07-27
09104.txt		5.63 (K)	00-07-27
U:\		SIZE	DATE
TIPS: [ENTER]:open [DEL]:DEL			
		Ready	ALARM
Saveto CNC	Switch	LASTLINE	NEXTLINE
Copy All			

1.3.3 Tool length, wear, macro variable interface, system variables



Press the key once from other display pages to enter the knife repair interface, and then press F1-F5 keys to switch the display of tool length, wear, tool tip, macro variables, etc System variables and other pages.

1) Knife length interface

The knife length interface has a total of 5 pages, a total of 33 biases, wear numbers

(No.000 ~ No.032) for user use, display each page by key, key, display page as shown in the figure:



2) Wear interface

The wear interface has a total of 4 pages, a total of 32 biases, wear numbers (No.00 1 ~

No.032) for users to use, display each page by  keys,  keys, display pages as shown in the figure:

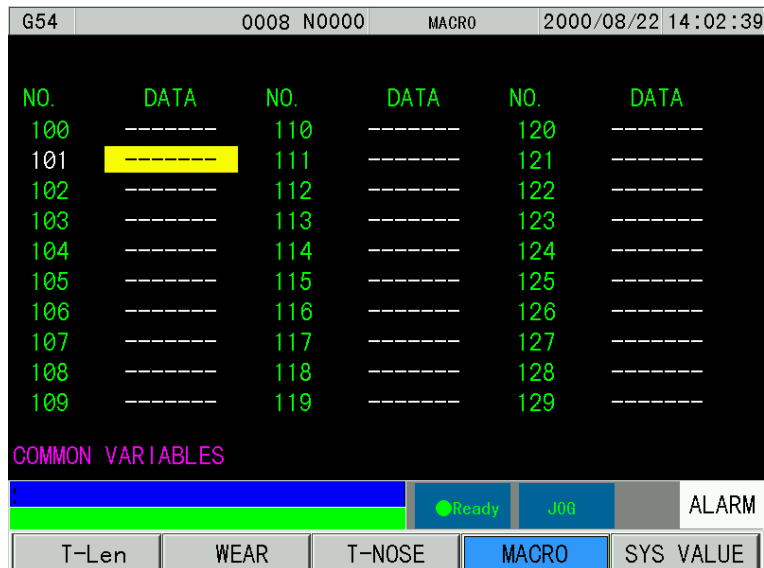


3) Macro variable interface

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



keys, and the macro variable page displays a total of 600 (No.100 ~ No.199 and No.500 ~ No.999) macro variable, the macro variable value can be specified through macro code or directly set by the keyboard.



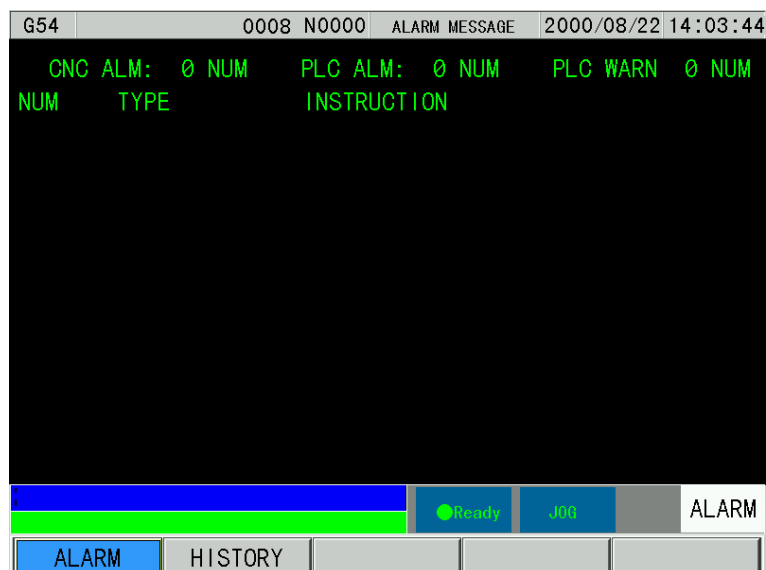
1.3.4 Alarm interface

1) Alarm:



Press the key to enter the alarm interface, and view the current alarm display

through the F1 key and F2 key, and the history is displayed as follows:



Note: Alarm clearance: When the alarm has been canceled, press the button to clear the alarm

content.**2) Alarm History/Log:**

Enter the alarm log interface on the alarm interface. Use the F2 key to view a total of 200 alarm log messages;

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

G54	0008 N0000	ALARM LOG	2000/08/22 14:05:51
NUM	INSTRUCTION	ALARM TIME	
0452	Z axis driver alarm.	00-08-22 12:39	
0451	X axis driver alarm.	00-08-22 12:39	
0452	Z axis driver alarm.	00-08-22 12:39	
1215	External cycle open time is to o long(normally	00-08-22 12:39	
0451	X axis driver alarm.	00-08-22 12:39	
1215	External cycle open time is to o long(normally	00-08-22 12:39	
0451	X axis driver alarm.	00-08-22 12:39	
0251	Emergency stop alarm	00-08-22 12:39	
0451	X axis driver alarm.	00-08-22 12:39	
1215	External cycle open time is to	00-08-22 12:37	
Ready JOI ALARM			
Empty			

Note: Manual clearing of alarm logs: Press the F1 key under the level 2 password to check the "Clear History" function key, which can clear all log information.

1.3.5 Settings interface

Find the "Settings" function by switching the "Up/Down" function at the bottom of the screen, and click the F1 key "Settings" to enter the setting interface, which contains password, backup, coordinate system, switch, time, and switch Interface.

1) Settings interface

The setting interface has five pages, which can be viewed by the corresponding function keys corresponding to F1-F5 at the bottom of the screen.

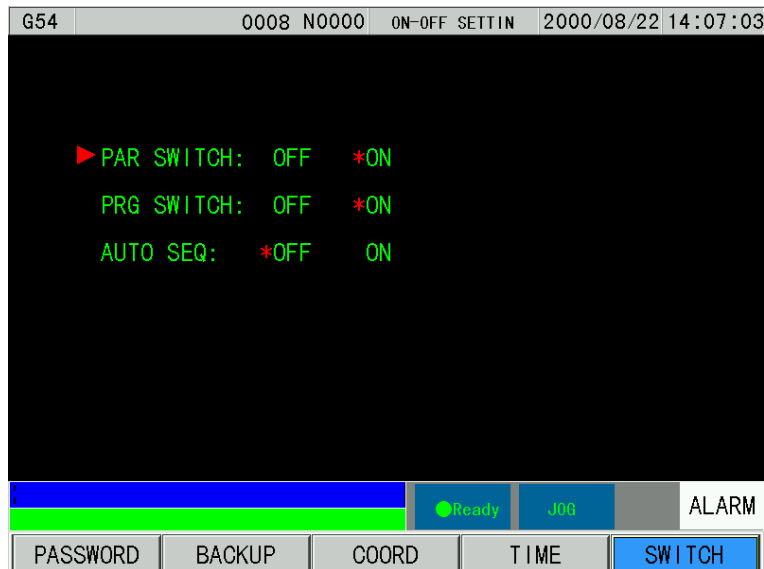
Switch settings: display parameters, programs, automatic serial number on and off status.

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

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

Automatic serial number: when the automatic serial number switch is turned on, the program segment number is automatically generated when editing the program; The program

segment number is not automatically generated when the automatic serial number switch is turned off, and must be entered manually if needed.



2) Backup interface

Data manipulation: From this page, you can back up and restore CNC data (status parameters, data parameters, screw parameters, tool offsets, etc.).

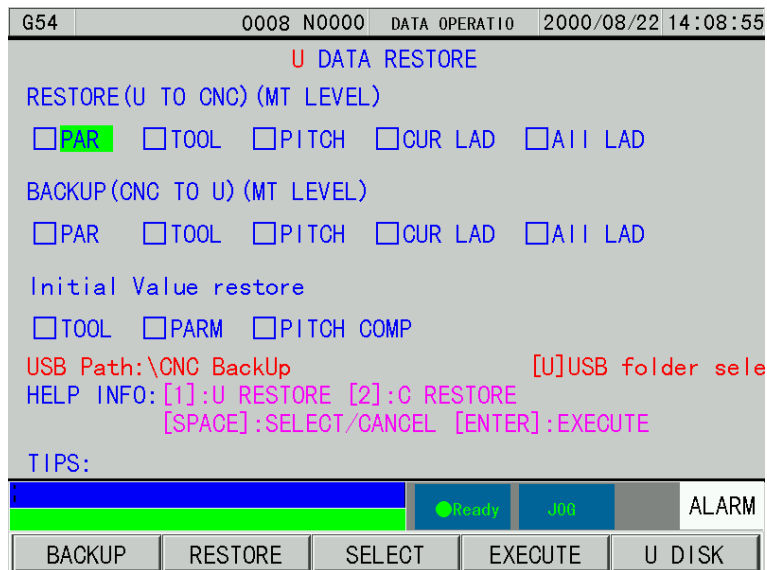
Initial value restore operation: You can restore parameters, knife compensation, and screw compensation to system defaults.

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

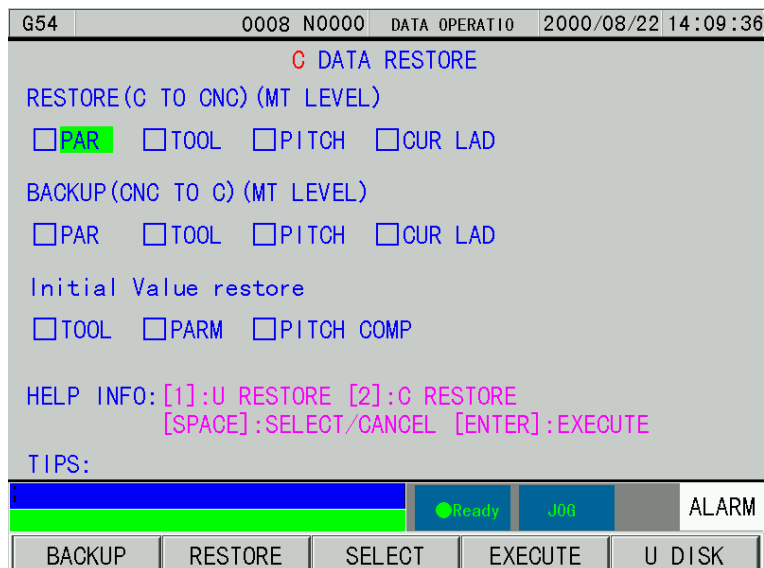
Restore U disk data to CNC: Return the data files backed up on the U disk to the system.

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

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



USB flash drive backup restore display page



C drive backup restore display page

3) Password interface

Permission settings: Display and set the user operation level.

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

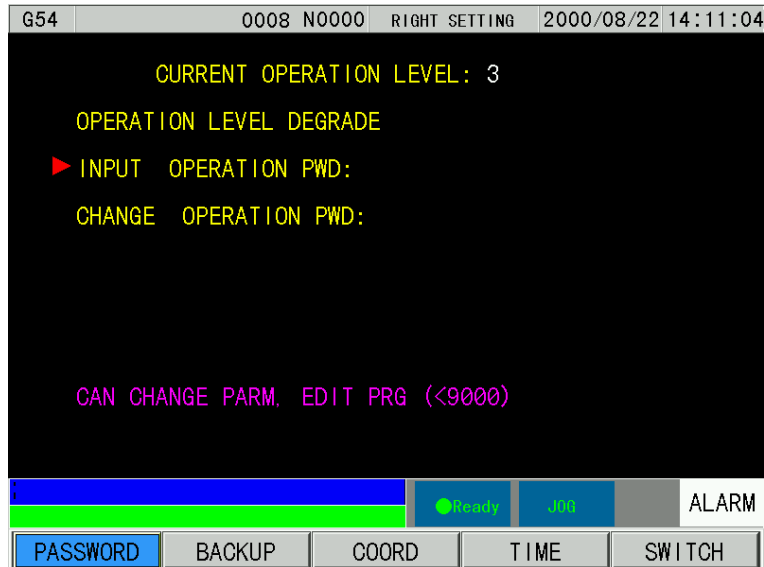
Machine tool manufacturer level: allow to modify CNC status parameters, data parameters, screw parameters, tool compensation data, edit part programs (including macro programs), edit and modify PLC ladder diagrams, download and upload ladder charts;

Device management level: initial password 12345, allowing modification of CNC status parameters, data parameters, knife supplement data, editing programs;

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

parameters, data parameters and screw parameters of the CNC.

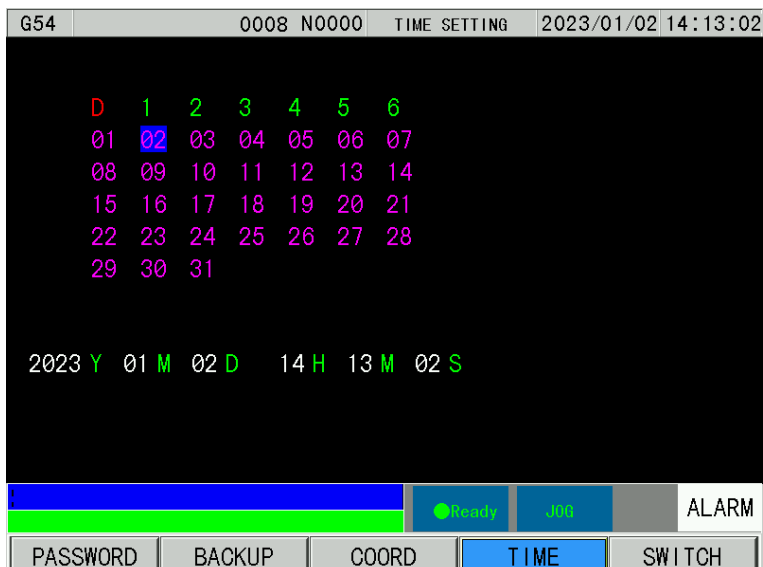
Machining operation level: no password level, can be operated by the machine tool operation panel, can not modify the tool supplement data, can not select the part program, cannot edit the program, can not modify the status parameters, data parameters and screw parameters of the CNC.



4) Time interface

TimeDate: Displays the current time and date.

At the first-level operation level, you can press the  key to enter the modification mode, and press the  key or  key to switch the year/month/day/hour/minute/ to be modified seconds, press the number key to modify its value, if you need to exit the modification mode, press the  key again.



1.3.6 Status parameters, data parameters, processing technology, user parameter interface

Find the "Parameter Settings" function through the "Up/Down" function key switch at the bottom of the screen, click the F2 key "Parameter Settings" to enter the parameter interface, which contains the parameter general table, processing technology, user parameters, bit parameters, and number parameters, and enter the F1-F5 key corresponding to its function under the interface .

1) Status parameter interface

Under "Parameter Settings", press the F4 key "bit parameter" to enter the status parameter interface, a total of 48 status parameters are displayed in two pages, and the relevant

parameters can be viewed or modified by  key and  key to enter each page, as follows:

As can be seen from the status parameter page, there are 8 lines of parameter content display lines in the lower part of the page, respectively showing the corresponding Chinese meaning of BIT0 to BIT7 8 bits, and the Chinese meaning of a bit of the parameter where the current cursor is located, you can press the left shift key or Move the right key to change the displayed parameter bits.

G54	0008 N0000		STATUS PARAM		2023/01/02 14:16:46	
NO.	DATA	NO.	DATA	NO.	DATA	
001	00011000	006	01100000	011	00000000	
002	01000010	007	10101111	012	00111001	
003	00110111	008	00011110	013	11000000	
004	01000000	009	10000100	014	01000000	
005	00110001	010	00011000	164	00001001	
BIT0:(0:Metric 1:Inch)Input						
BIT1:NULL						
BIT2:(0:Diameter 1:Radius)Programming						
BIT3:(0:Step 1:Handwheel)Mode						
BIT4:Speed Control_Spindle(0:Swtich 1:Analog)						
BIT5:NULL						
BIT6:NULL						
BIT7:NULL						
Data			●Ready	JOI	ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM	NUMPARM		

2) Data parameter interface

Press the F5 key "Number Parameters" to enter the data parameter interface, and you can

enter each page through  keys and  keys to view or modify relevant parameters, as follows:

As you can see from the data parameters page, the middle of the page table shows the meaning of the parameters Chinese, and you can move the cursor up or down to select different parameters.

G54	0008 N0000	NUM PARAMETER	2023/01/02 14:17:26
NO.	MEANING	DATA	
015	X multiplication coefficient (CMR)	10	
016	Z multiplication coefficient (CMR)	10	
017	X command frequency division coefficient (CMD)	10	
018	Z command frequency division coefficient (CMD)	10	
019	Back tail length TCH when thread cutting (back width=TCH)	5	
020	Null	0	
021	Offset comp. of spindle analog voltage output voltage	0.0000	
022	X-Axis rapid movement speed (radius)	5000	
023	Z-Axis rapid movement speed	5000	
024	S front acc/dec time setting when X rapid movement	100	
025	S front acc/dec time setting when Z rapid movement	100	
026	Acc/dec time constant of short axis when thread back ta	60	
MIN	1	MAX	65536
		P: 1/33	
Data		●Ready	JOG
		ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM
		NUMPARM	

3) Processing process interface

Press the F2 key "Processing Technology" to enter the processing process setting interface,

the processing technology is displayed on a total of 2 pages, you can use the  key,  key to display each page:

G54	0008 N0000	PROC TE	2023/01/02 14:18:24
NO.	MEANING	DATA	
6100	Alter Offset during processing(0:Invalid 1:Valid)	1	
6110	(0:Diameter 1:Radius)Programming	0	
6130	Value of Tool Offset(0:Diameter 1:Radius)	0	
6140	Processing Mode Between blocks(0:Continuous 1:Exact)	1	
6150	Back tail length TCH when thread cutting(back width=TCH)	5	
6160	Max.Value of Tool offset/Wear	50.0000	
6170	G71/G72 cycle turing single tool feeding value	0.0010	
6180	G71/G72 cycle turing sigle tool retraction value	0.0000	
6190	Tool retraction value of X when G73 cycle turing	0.0000	
6200	Tool retraction value of Z when G73 cycle turing	0.0000	
6210	Cutting count of G73 cycle turning	1	
6220	Tool retraction value of Z/X of G74/G75 cycle turing	0.0000	
P: 1 / 2			
		●Ready	JOG
		ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM
		NUMPARM	

4) User parameter interface

Press F3 key "User Parameters" to enter the user parameter setting interface, user

parameters are displayed on a total of 2 pages, and each page can be displayed by



key and



key:

G54	0008 N0000	USERPARM	2023/01/02 14:21:16
NO.	MEANING	DATA	
7110	Processed total workpieces	0	
7111	Required process total workpieces	0	
7120	1/0:Yes/No alarm after reaching process numbe	1	
7130	Counting Of Pieces with M99 (0:NO 1:YES)	1	
7131	Counting Of Pieces with M31 (0:NO 1:YES)	0	
7140	Memory Pieces Number When Power Off (0:YES 1:NO)	0	
7141	Clear finished Count of Pieces (0:NO 1:YES)	0	
7210	Auto Clear Processing Time of Each piece (0:NO 1:YES)	1	
7220	1/0:reseting spindle lubricant and coolant ou	0	
7230	Lubrication open time(0 shows that Lubrication is not r	0	
7231	Auto lubricant interval time	3600	
7240	Cursor Return to Beginning after M30 (0:NO 1:YES)	1	
MIN	0	MAX	99999
		P: 1 / 2	
		●Ready	JOG
		ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM
		NUMPARM	

1.3.7 CNC diagnostics, PLC signals, IO diagnostics, version information, alarm interface



The key is a diagnostic key, which can enter the CNC diagnosis interface, PLC signal interface, IO diagnosis, version information and alarm interface.

1) CNC diagnostic interface

The status of input/output signals between CNC and machine tools, the status of signals transmitted between CNC and PLC, PLC internal data and CNC internal status can be

displayed through diagnostics. Press the key and then press the F1 key to enter the CNC diagnosis page, which displays keyboard diagnosis, status diagnosis and auxiliary function

parameters. It can be viewed by



key,



key.

On the CNC diagnostic display page, there are two lines of diagnostic number details display lines in the lower part of the page, and the second line shows the Chinese meaning of a certain bit of the diagnostic number where the current cursor is located, and you can press the left shift key or right shift key to change the displayed diagnostic bit; The first line shows the abbreviation of all bits of the diagnostic number where the current cursor is located.

G54	0008 N0000		CNC DGN	2023/01/02 14:22:17	
NO.	DATA	NO.	DATA	NO.	DATA
090	-12406	100	-1	110	0
091	16406	101	300	111	-1
092	0	102	300	112	0
093	0	103	300	113	0.000
094	0	104	0.000	114	0.000
095	0	105	1.000	115	0.000
096	0	106	0.000	116	0.000
097	0.000	107	1.000	117	19
098	0.000	108	0.001	118	0.000
099	-1	109	0.000	119	0.000
Spindle speed analog voltage output value					P:4/6
			Ready	JOI	ALARM
CNC DGN	PLC SIG	IO DGN	VER INFO	ALARM	

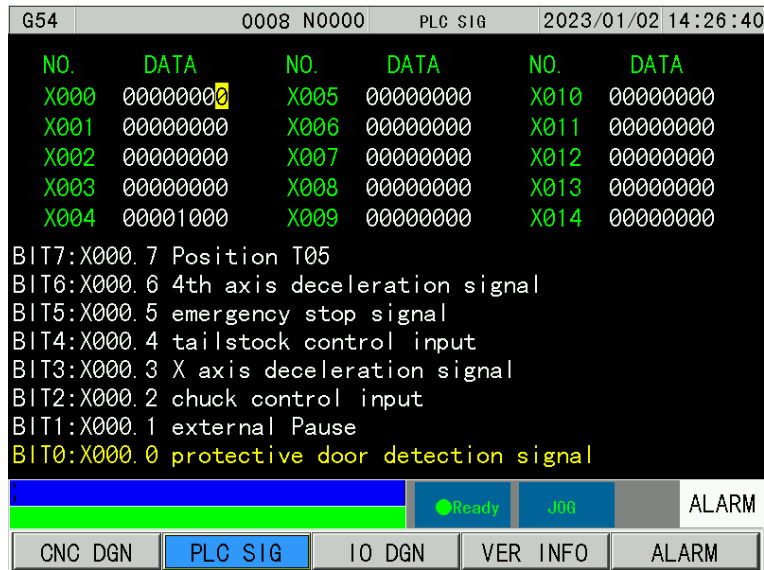
2) PLC signal interface

On the page of the PLC signal interface, X0000 ~ X0063, Y0000 ~ Y0047, F0000 ~ are

displayed in turn Address status such as F063, G0000 ~ G063. Press the  key and then

press the F2 key 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, there are two detailed content display lines in the lower part of the page, the second line shows the Chinese meaning of a certain bit of the address number where the current cursor is located, you can press the left shift key or the right shift key to change the displayed address bit; The first line displays the abbreviation of all digits of the address number where the current cursor is located.

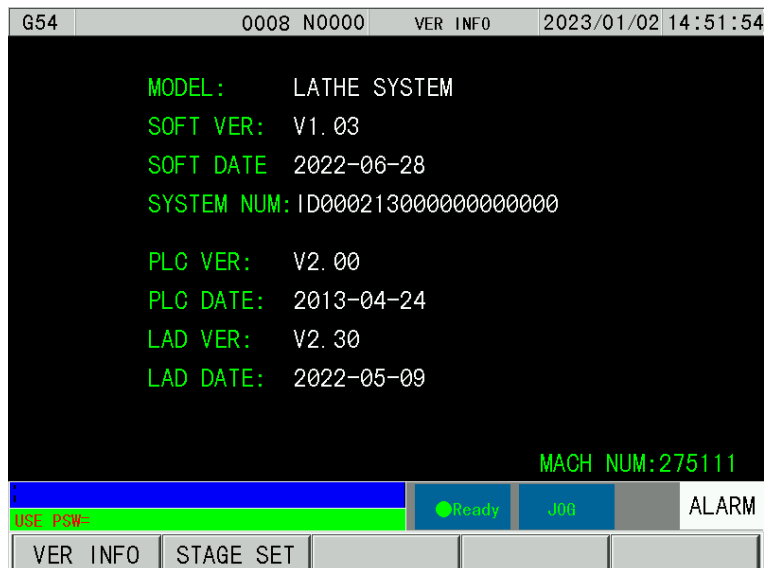


3) Version information



Press the key and then press the F4 key to enter the version information interface.

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

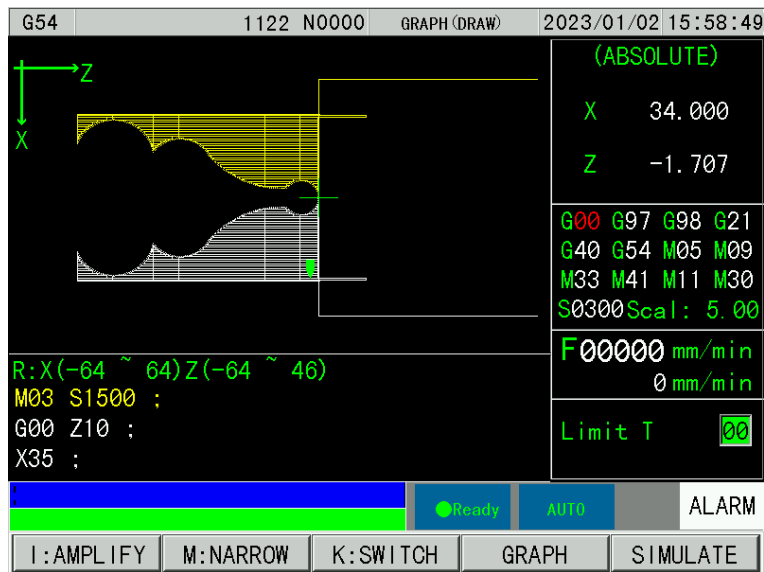


1.3.8 Graphical Interface

Find the "Graphic" function by switching the "Up/Down" function at the bottom of the



screen or click on the program supervision interface to find it, and then click the F2 key "Graphic" to enter the graphical interface, which can enlarge, zoom out, Graph generation, graph simulation, clearing, and more.



Chapter II: Power On, Shutdown and Safety Protection

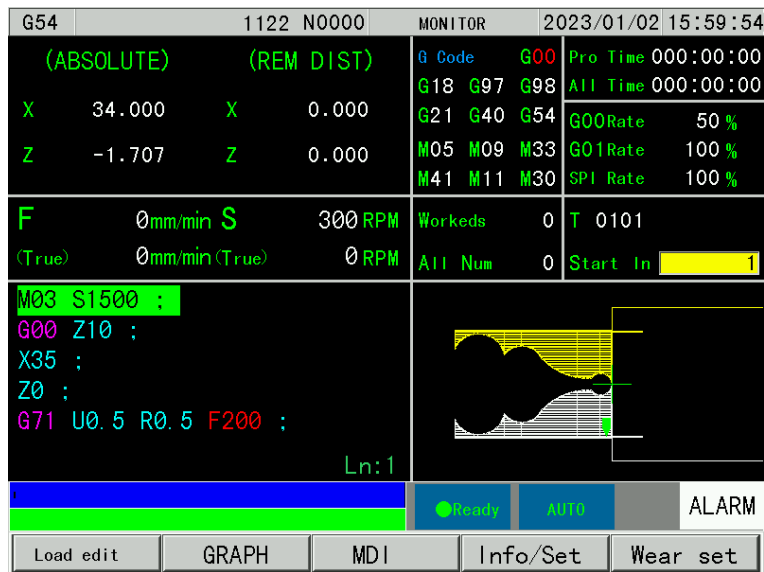
2.1 Boot

Before powering on, it should be confirmed: 1. The machine tool status is normal.

2. The power supply voltage meets the requirements.

3. The wiring is correct and firm.

Self-test and initialization after power-on. After the self-test and initialization are complete, the Current Location (Relative Coordinates) page is displayed.



2.2 Shutdown

Before shutting down, you should confirm that:

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

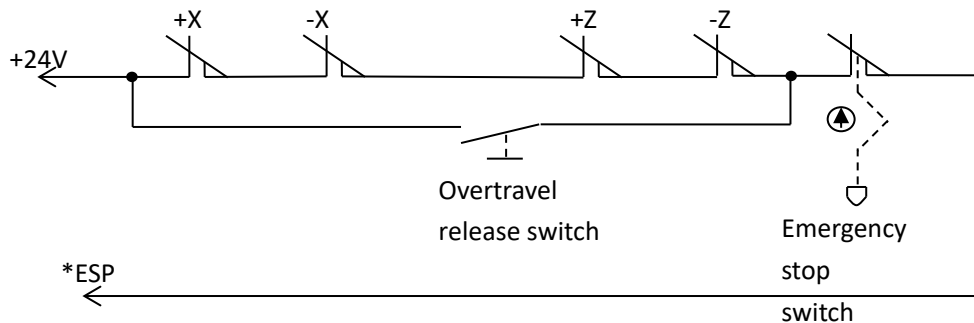
tool.

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

2.3 Overtravel Protection

2.3.1 Hardware Overtravel Protection

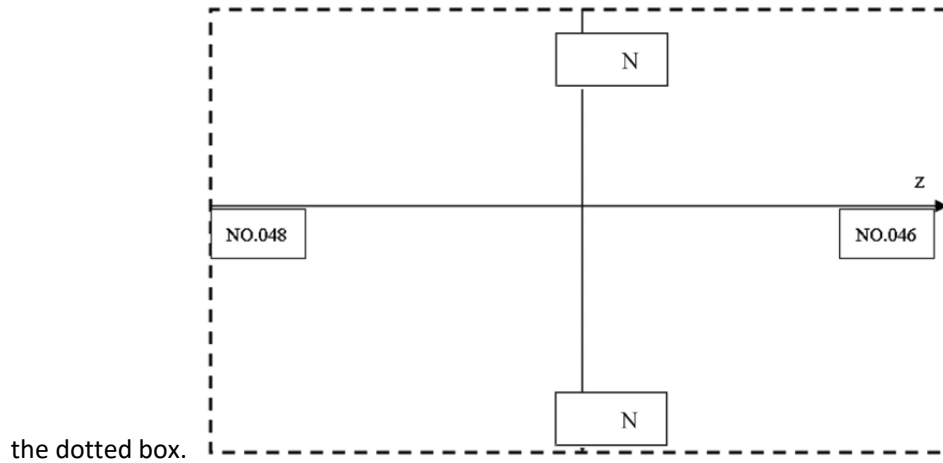
Install the stroke limit switches at the positive and negative maximum strokes of the X and Z axes of the machine tool respectively, and wire them as shown below, at this time Bit2 (EALM) of the status parameter No.215 must be set to 0. When there is an overtravel, the stroke limit switch operates, stops the movement and displays the emergency stop alarm; In addition, the external method X and Z travel limit switches are connected to the X, Z corresponding hard limit overtravel input points.



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

2.3.2 Software overtravel protection

The travel range of the X, Z axis software is set by the data parameters NO.045, NO.046, NO.047, NO.048, and the machine coordinate value is used as the reference value. As shown in the figure below, X and Z are the two axes of the machine tool coordinate system, and NO.045 and NO.047 are the positive and negative maximum strokes of the X axis. NO.046 and NO.048 are the positive and negative maximum travel of the Z axis, and the software travel range is in



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

Additional axes

The travel range of the Y-axis software is set by the data parameters NO.192 and NO.195, and the machine coordinate value is used as the reference value. NO.192 is the maximum stroke in the positive direction of the Y axis, and NO.195 is the maximum stroke in the negative direction of the Y axis.

The 4th axis software travel range is set by the data parameters NO.193 and NO.196, and the machine coordinate value is used as the reference value. NO.193 is the 4th axis positive maximum stroke, and NO.196 is the 4th axis negative maximum stroke.

The 5th axis software travel range is set by the data parameters NO.194, NO.197, and the machine coordinate value is used as the reference value. NO.194 is the 5th axis positive maximum stroke, and NO.197 is the 5th axis negative maximum stroke.

2.4 Emergency Operations

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

2.4.1 Reset

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

the  key to make the system in a reset state:

1. There is an axial movement stop;
2. M, S function output is invalid (can be defined by whether the spindle turns off counterclockwise / clockwise turn, lubrication, cooling and other signals after the

parameter setting button , PLC ladder diagram);

3. The automatic operation ends, and the modal function and state are maintained.

2.4.2 Emergency Stop

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

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

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

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

Note 4: External emergency stops are valid only if Bit2(EALM) of the status parameter NO.215 is set to 0.

2.4.3 Feed hold

The operation can be paused by  pressing a button during machine operation. It is

important to note that this function does not stop the running action immediately during thread cutting, tapping, and cyclic code running.

2.4.4 Cut off the power supply

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

Chapter III: Manual Operation

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

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

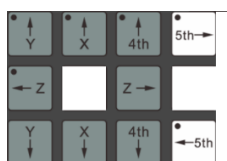


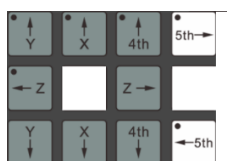
Press the  key to enter the manual operation mode, under the manual operation mode, manual feeding, spindle control, rate adjustment, tool change and other operations can be carried out.

3.1 Axis movement

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

3.1.1 Manual feeding

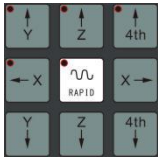


Press  and hold the feed axis and the direction selection key  or

Press the X axis arrow key to make the X axis negative or positive feed, and the axis movement stops when the button is released; Press  and  hold the Z axis arrow key to make the Z axis negative or positive feed, and the axis movement stops when the button is released; Press  and  hold the Y axis arrow key to make the Y axis negative or positive feed, and the axis movement stops when the button is released; Press  and  hold or 4th axis arrow key to make the 4th axis negative or positive feed, and the axis movement stops when the button is released; The 5th axis operation is consistent with the above axis operation.

When manual feeding is performed, press the  key to make the indicator light in the status indicator area light up to enter the manual fast move state.

3.1.2 Manual fast movement

Press  the key in the middle until the  fast movement indicator in the status indicator area is on, press the  or  key to make the X axis move quickly in the negative or positive direction, and the axis movement stops when the button is released; Press the  or  key to make the Z axis move quickly in the negative or positive direction, and the axis movement stops when the button is released; Press  and  hold the or Y axis arrow key to make the Y axis negative or positive feed, and the axis movement stops when the button is released; Press  and  hold or 4th axis arrow key to make the 4th axis negative or positive feed, and the axis movement stops when the button is released; The 5th axis operation is consistent with the above axis operation; Fast magnification

real-time trimming is effective.

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

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

Note 2: In edit/handwheel mode, the  key is invalid.

3.1.3 Speed Trimming



In manual feeding, the manual feed rate can be modified according to a total of 16 steps. When parameter NO.031 is set to 1260, the relationship between feed rate and feed rate is shown in the following table

Feed rate (%)	Feed rate (mm/min).
0	0
10	126
20	252
30	378
40	504
50	630
60	756
70	882
80	1008

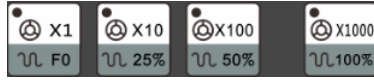
Feed rate (%)	Feed rate (mm/min).
90	1134
100	1260
110	1386
120	1512
130	1638
140	1764

150

1890

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

When moving quickly manually, the magnification of manual fast movement can be

modified by pressing the fast magnification, and  the fast magnification has F0, 25%, 50%, and 100% four gears. (F0 speed is set by data parameter No.032).

Fast magnification selection is effective in the following situations:

- 1) G00 moves fast
- 2) Fast movement in a fixed cycle
- 3) Fast movement when G28
- 4) Move quickly manually

3.2 Other manual operations

3.2.1 Turn counterclockwise, turn clockwise, stop control



: In manual operation mode, press this key to rotate the spindle in reverse time;



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



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

3.2.2 Spindle jogging



: The spindle is now jogged.



Function description: Press the  key on the panel to enter the jogging state, and the spindle jogging function needs to be in the stop state for opening and closing the spindle jogging

function. Spindle jogging state, press the  key, counterclockwise to jog; Press the  key to turn the jog clockwise. Jogging time and speed are set by data parameters No.108 and No.109 respectively.



When the spindle jog rotates, press the  key to stop the spindle jogging rotation,

and the spindle braking signal will not be output when the jogging rotation stops.

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

Parameter settings:

PLC parameter K10.4 1/0: Spindle jogging is effective in any way / manual and handwheel mode.

Data parameter No.108: spindle jogging time

Data parameter No.109: rotational speed when the spindle is jogged

3.2.3 Coolant Control



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

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

3.2.4 Lubrication control

Function description:

1. Non-automatic lubrication:

T13 = 0: Non-automatic lubrication.

When the data parameter NO.112=0, in order to lubricate the flip output, press the machine



tool operation panel key to lubricate the output, and press it repeatedly to cancel the lubrication output. When executing M32, lubricate the output, then execute M33, lubricate the output cancel.

When the data parameter NO.112>1, for the lubrication timing output, press the machine



tool operation panel key to lubricate the output, and after the time set by the data parameter 112, the lubrication output is canceled; M32, lubrication output, after the time set by the data parameter NO.112, the lubrication output is canceled. If the time set by 112 is not reached, M33 is executed at this time, and the lubrication output is canceled.

2. Automatic lubrication:

T13>0: Automatic lubrication with configuration time T13 and lubrication interval T53

After power-on, start lubricating the time set by T13, then stop the output, and after the time set by T53, repeat the output lubrication, and cycle in turn. M32, M33 codes, machine

operator panel  keys are also valid for automatic lubrication.

Parameter settings:

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

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

PLC data: T53: Automatic lubrication interval time (ms).

PLC data: T13: Automatic lubrication output time (ms).

Data parameters: No.80:M code execution duration (ms).

Data parameters: No.112: lubrication opening time (0-60000ms) (0: lubrication unlimited time).

Additional Notes:

- 1) When DT13 is not zero, automatic lubrication is effective, and the output lubrication will be output after each interval set by DT 53 (the time set by DT 13); Manual and command can also turn on or off the lubrication output, the opening time is set by DT13 (the timer will restart after this operation);
- 2) When automatic lubrication is active, if K16.2 is set to 1, the system will lubricate the output as soon as it is powered on, and the output time will be set by DT13, if K16.2 is set to 0, the lubrication will not start until the time set by DT 53 is passed after power-on.
- 3) When DT13=0, the manual lubrication is effective, if the parameter No.112=0, the output will not be automatically turned off, if it is not 0, the output time will automatically turn off the lubrication output after reaching the time set by NO.112;
- 4) When DT53=0, the lubrication output enters DT13 and turns off the output and no longer spaced the output; When DT53 is not 0, the lubrication output enters DT13 and turns off the output, and the output is output after the DT53 interval output time;
- 5) PLC parameters
K16.1==0 Insufficient lubricant warning signal is valid with +24V (0: on 1: off);
K16.6==1 Insufficient lubricant warning (0: invalid 1: valid);

- 6) Number parameters

0	7	9
---	---	---

Lubricant deficiency detection delay unit: milliseconds

3.2.5 Manual tool change



In manual/handwheel operation mode, press this button to manually change the tool in order (if the current tool is the 1st tool, after pressing this button, the tool is changed to the 2nd; If the current tool is the last tool, press this key to switch the tool to the 1st tool).

3.2.6 Tuning of spindle magnification

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



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

Chapter 4 Handwheels/Manual Operation

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

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

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

4.1 Manual feeding

See paragraph 3.1.1 of this article for details

4.1.1 Selection of increments

(This system does not have this option)

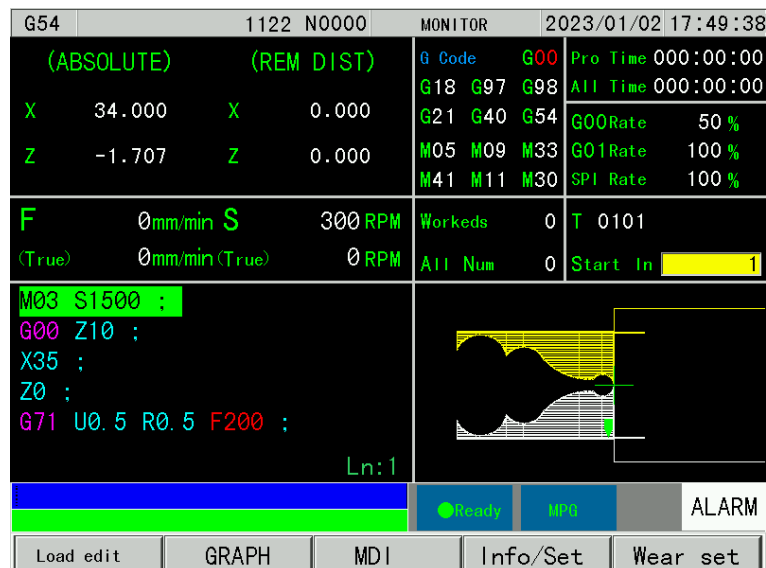
4.1.2 Movement direction selection

See paragraph 3.1.1 of this article for details

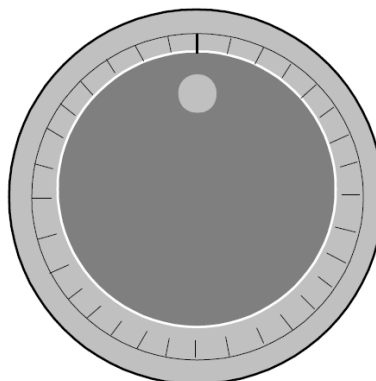
4.2 Handwheel (Hand Crank Pulse Generator) feed



Set the Bit3 bit of system parameter No.001 to 1, press the  key to enter the handwheel operation mode, and the page is displayed as follows:



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



Handwheel shape drawing

4.2.1 Selection of increments

(This system does not have this option)

4.2.2 Selection of movement axis and direction

In 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 clockwise for negative feed and counterclockwise for positive feed, the handwheel end A and B signals can be exchanged. The feed direction when the handwheel rotates can also be selected by the BIT0 ~ BIT4 position of parameter No 013.

4.2.3 Other Operations

1. Turn counterclockwise, turn clockwise, stop control



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



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



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

2. Spindle jogging



: The spindle is now jogged.



In the spindle jogging state, press the  key to turn jogging counterclockwise; Press the  key to turn the jog clockwise. The spindle jogging time and speed are set by data parameters No.108 and No.109, respectively. See section 3.2.2 of this article for details.

3. Coolant control

See paragraph 3.2.3 of this article for details

4. Lubrication control

See paragraph 3.2.4 of this article for details

5. Manual tool change



: Manual / Handwheel / Single Step mode, press this button to change the tool in order.

6. Trimming of spindle magnification

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



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

4.2.4 Descriptions

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

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

The amount of movement per tick on the handwheel

Handwheel increment 0.001 0.01 0.1 1

The coordinates specify a value of 0.001mm 0.01mm and 0.1mm 1mm

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

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

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

Chapter V: Input Operations

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

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

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

5.1 Code Word Input

Select the input operation mode, enter the program status page, enter a program segment G0 X50 Z100, the operation steps are as follows:



1. Press the  key to enter the input operation mode;
2. Press the F3 key "MDI" under the screen to enter the MDI program editing status page



3. Type the address key  and number key in turn ;

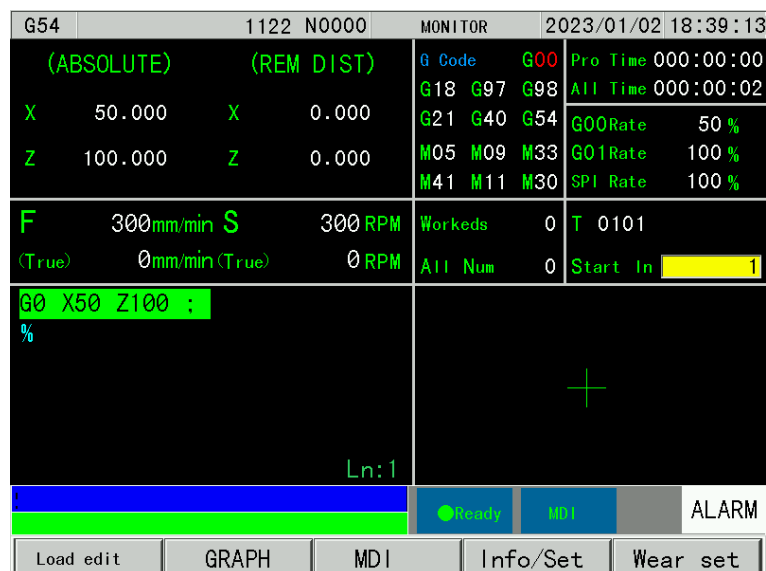
4. Type the address key , number key , , .

5. Type the address key , number key , and number key  in turn;

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

5.2 Code Word Execution

After the code word is entered, press the  key (or  key or  key) and the page is displayed as follows:



Press the "Cyclic Start" button to execute the entered program segment. During operation, the "Feed Hold" button, the "Reset" button and the "Emergency Stop" button can be pressed to stop the program segment.

Note: Subroutine call code (M98 P; etc.), composite cutting cycle codes (G70, G71, G72, G73, G74, G75, G76 etc) Execution under MDI is invalid.

5.3 Setting of Parameters

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

5.4 Modification of Data

Under the program status page, if there is an error in the field input process before executing the entered program segment, you can press the  key to cancel the inversion state and modify the program, or press the  key to clear all the content and re-enter the correct program segment.

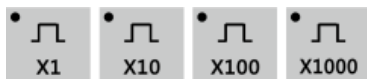
5.5 Other Operations

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



Press the  key to adjust the spindle magnification to change the spindle speed, which can realize 8 levels of real-time adjustment of spindle magnification 50%~120 %.

2. It can be repaired and quickly magnified



Press the     key to adjust the fast moving feed speed, which can achieve 4 levels of real-time adjustment of fast movement speed.

3. The feed rate can be repaired



Under the input operation mode, press the  key to change the feed rate to change the feed speed, and the actual speed can be adjusted in real time for 0 ~ 150% of the feed rate specified in the F code, a total of 16 levels.

Chapter VI: Program Editing and Management

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

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

For ease of management, the system provides 3 levels of user permission settings. Must have an operating level of 4 or more (level 4, level 3, etc.).

to open the program switch and edit the program. The actions allowed for each operation level are described in Section 10.3.

6.1 Establishment of Programs

6.1.1 Generation of program segment numbers

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

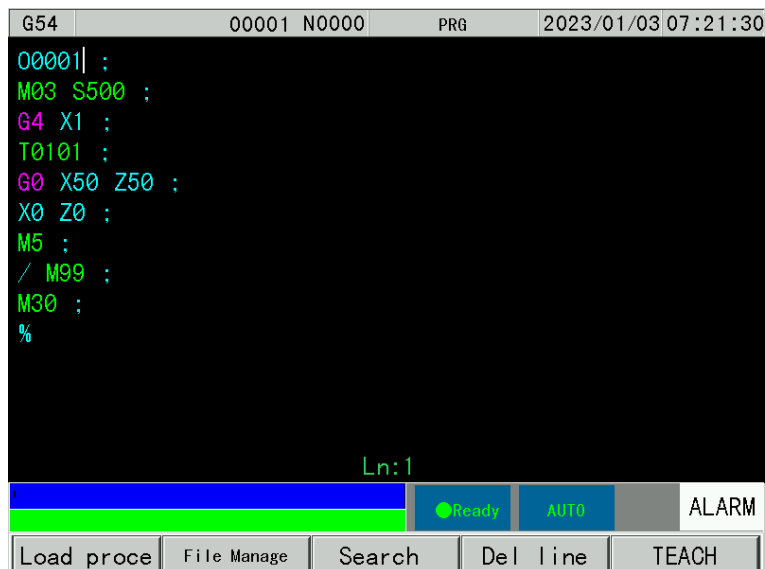
When the "Auto Serial Number" switch on the switch setting page is off, the CNC does not automatically generate the program segment number, but the program segment number can be manually programmed during programming.

When the "Automatic Serial Number" switch on the switch setting page is on, the CNC automatically generates the program segment number, and when editing, press the  key to automatically generate the program segment number of the next program segment, and the incremental value of the program segment number is set by the CNC data parameter No 042 (The settings for automatic serial numbers are described in Section 10.1.1 of this article).



6.1.2 Input of program content

1. Press the  key to enter the program editing interface.



2. Press the  key to insert a new program segment line;



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

4. Follow the method of step 3 to complete the input of other program segments of the program.

Note: 1. Support entering separate letters or numbers in program editing mode.

2. When entering the program, if you find that the input instruction word is wrong, you can press the



key to cancel the input instruction one by one.

6.1.3 Cursor movement

1) Press the  key to select the program content to display the page.

2) Press the  key to move the cursor up one line; If the number of columns where the current cursor is located is greater than the total number of columns in the previous row,

press the  key, and the cursor moves to the end of the previous program segment;

3) Press the  key, the cursor moves down one row, if the number of columns where the current cursor is located 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 to move the cursor to the right column; If the cursor is at the end of the line, the cursor moves to the beginning of the next program segment;

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

6) Press the  key to page up;

7) Press the  key to page down;

8) On the program editing display page, press the  key and the cursor returns to the beginning of the program.

6.1.4 Search for words and line numbers

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

The steps for the search method are as follows:

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

2) Enter the word you want to retrieve.

3) Press the  key (determine whether to press the  key or the  key according to the position relationship between the character you want to find and the character in which the current cursor is located), and the display page is as follows:



4) Press the up/ down arrow keys again to find the word in the next position.

Note 1: If not found, the entered search word will disappear.

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

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

Search of the first and last rows: Quickly view the display of the content of the first and last lines of the program segment.

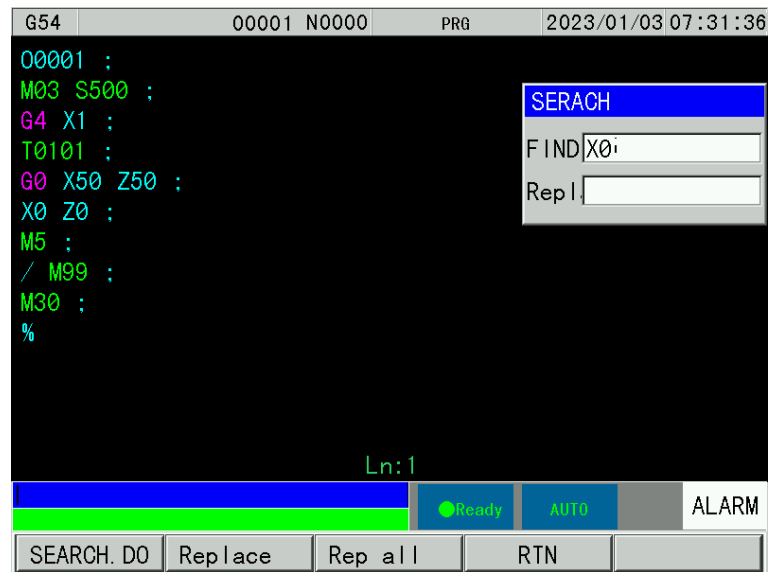
The steps for the search method are as follows:

1) Display page in the program content editing;

2) Press the  key, the program quickly switches the display back and forth between the first and last pages;

Edit word lookup:

You can press the function key F3 "Find/Replace" to enter the search word function, enter the desired search content to determine the search;



6.1.5 Insertion of words

The how-to steps are as follows:

- 1) Display page in the program content editing;
- 2) Use the up/down/left/right arrow keys to move the cursor to the position where you need to edit the input, and the system will insert the input to the left of the cursor.

6.1.6 Deletion of Words

The how-to steps are as follows:

- 1) Display page in the program content editing;
- 2) Move the cursor to the position that needs to be deleted, press the  key, and the system will delete the cursor position to the left;

6.1.7 Modification of Words

The how-to steps are as follows:

- 1) Display page in the program content editing;
- 2) Move the cursor to the place that needs to be modified, press the  key to delete the incorrect word (delete to the left of the cursor), and edit the correct words one by one after deletion (add insert to the left of the cursor).

6.1.8 Deletion of Single Program Segments

The how-to steps are as follows:

- 1) Display page in the program content editing;
- 2) Move the cursor to any place where you need to delete the line, click the F4 key "Delete line" function under the screen, and then click the F1 key "Confirm" under the screen to delete the entire program segment line (as shown below).



6.1.9 Deletion of Multiple Program Segments

The how-to steps are as follows:

(See 6.1.12 operation copy/cut below)

6.1.10 Block Deletion

(The system does not have this function)

6.1.11 Replication of Single Program Segments

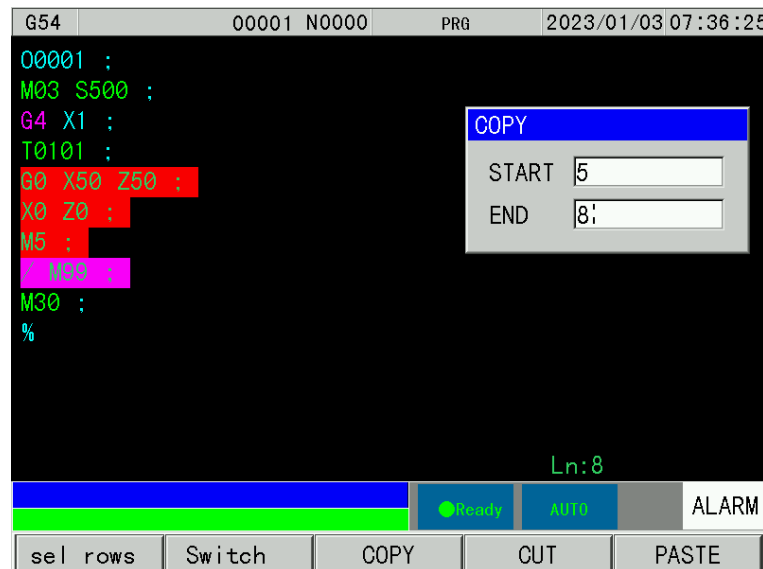
The how-to steps are as follows:

(See 6.1.12 operation copy/cut below)

6.1.12 Replication of multiple program segments

The how-to steps are as follows:

- 1) Display page in the program content editing;
 - 2) Move the cursor to the first code word of the program segment to be copied;
 - 3) Click the left/right page key under the screen to find the F1 key "Area Copy" and click to enter;
 - 4) Click the F1 key "Line Selection", the system will pop up the row selection dialog box, click F2 "Input Switch" to switch between the start row and the end row bar;
 - 5) Enter the starting and ending line numbers that need to be copied (or use left/right button to move after entering the switch) to confirm the selected range;
 - 6) Click F3 key "Copy" to record the copy area;
 - 7) Move the cursor to the position where you need to paste the program segment, and click the F5 key "Paste" function to complete the paste;
- Note: After copying and pasting, click on the bottom of the screen and use the left button to exit the "Region Copy" mode.



6.1.13 Copying of Blocks

(Refer to 6.1.12 operation copy/cut/paste above).

6.1.14 Pasting of program blocks

(Refer to 6.1.12 operation copy/cut/paste above).

6.2 Deletion of Programs

6.2.1 Deletion of Individual Programs

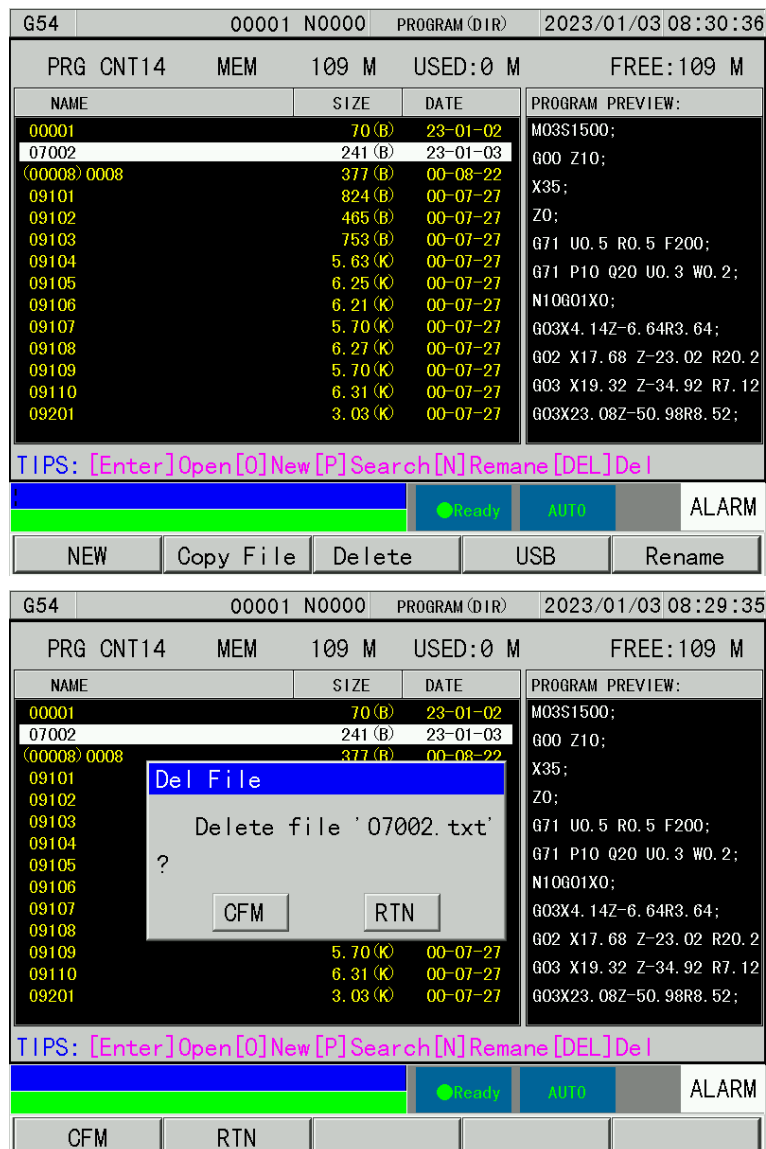
The procedure is as follows:

1) Click to enter the  program interface, click F2 key "File Management" to enter the local file directory;

2) Use  and  key movement to locate the program file;

3) Click the F3 key "Delete" under the screen, and the system prompts whether to delete the file dialog box;

4) Click the F1 key "Confirm" under the screen to delete the program;



6.2.2 Deletion of All Programs

(The system does not support all program removal)

6.3 Selection of Programs

When there are already multiple programs stored in the CNC, you can choose to open the program by the following methods.

The procedure is as follows:

1) Click to enter the  program interface, click F2 key "File Management" to enter the local file directory;

2) Use  and  key movement to locate the program file;

3) Click the  key to confirm the opening of the program, and then click the F1 key under the screen to "perform processing";

6.4 Renaming of the Program

1) Click to enter the  program interface, click F2 key "File Management" to enter the local file directory;

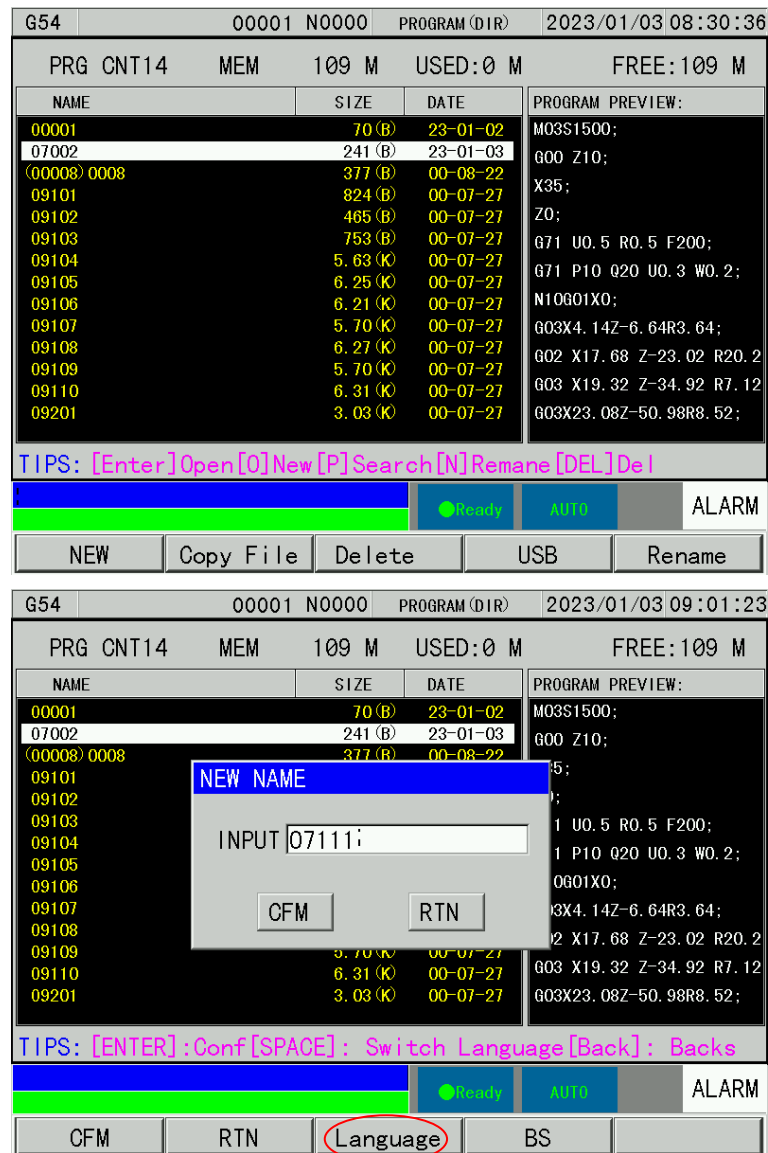
2) Use the  and  keys to move to find the program file that needs to modify the program name;

3) Click the F5 key "Rename" under the screen, and the system will pop up the rename dialog box;

4) Click the F4 key "backspace" under the screen to delete the original program name, and click the number key or letter key to enter the new program name;

5) Click the F1 key "Confirm" under the screen to complete the program name replacement;

Note: In the renaming, the system supports switching the input Chinese program name.



6.5 Reproduction of Programs

To save the current program:



1) Click to enter the  program interface, click F2 key "File Management" to enter the local file directory;



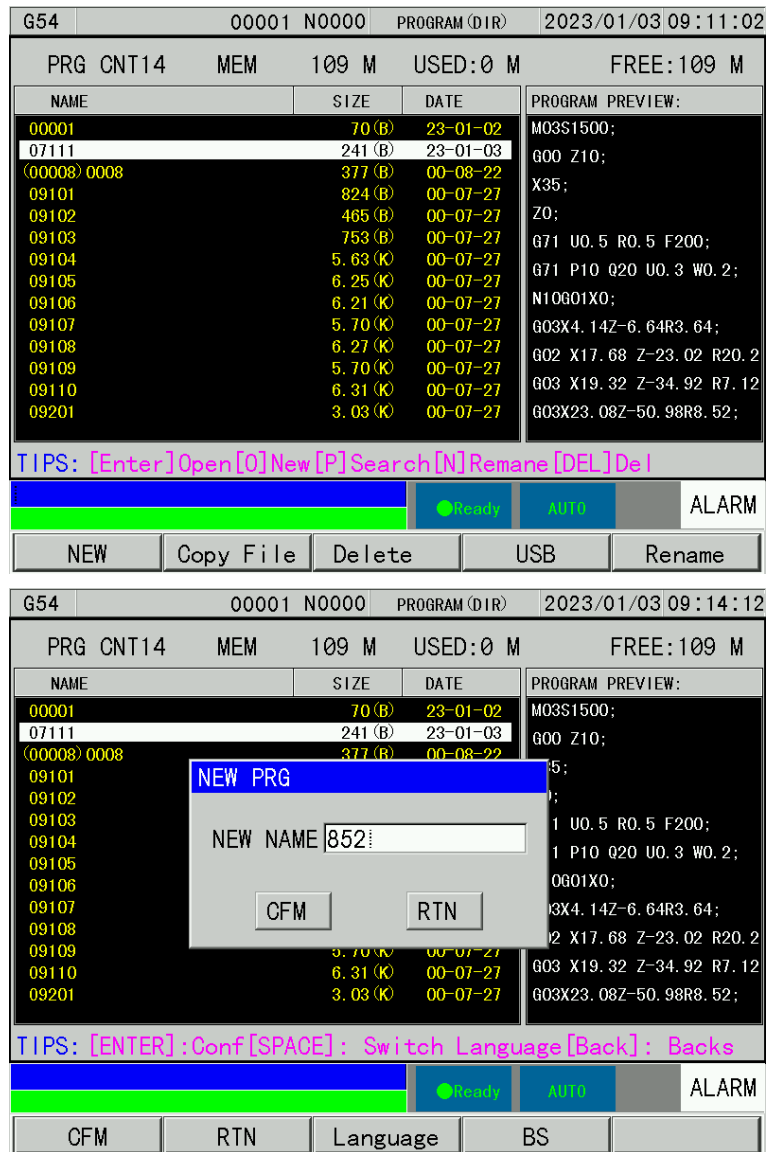
2) Use  and  key movement to find the program files that need to be copied;

3) Click the F2 key "Program Copy" under the screen, and the system will pop up the new program name input dialog box;

4) Click the number or letter keys to enter the new program name;

5) Click the F1 key "Confirm" under the screen to copy the program;

Note: In program copying, the system supports switching the input Chinese program name.



6.6 Program Management

6.6.1 Program Directory



Press the  key to enter the program directory display page. On this page, the CNC saved program names are displayed in the form of a table of contents, a page can only display up to 14 program names, when the number of saved programs exceeds 14, press up / The Page Down key displays another page of the program catalog.

G54	00001 N0000	PROGRAM (DIR)	2023/01/03 09:16:48
PRG CNT15	MEM	109 M	USED:0 M FREE:109 M
NAME	SIZE	DATE	PROGRAM PREVIEW:
00001	70 (B)	23-01-02	M03S1500;
00852	241 (B)	23-01-03	G00 Z10;
07111	241 (B)	23-01-03	X35;
(00008) 0008	377 (B)	00-08-22	Z0;
09101	824 (B)	00-07-27	G71 U0.5 R0.5 F200;
09102	465 (B)	00-07-27	G71 P10 Q20 U0.3 W0.2;
09103	753 (B)	00-07-27	N10G01X0;
09104	5.63 (K)	00-07-27	G03X4.14Z-6.64R3.64;
09105	6.25 (K)	00-07-27	G02 X17.68 Z-23.02 R20.2
09106	6.21 (K)	00-07-27	G03 X19.32 Z-34.92 R7.12
09107	5.70 (K)	00-07-27	G03X23.08Z-50.98R8.52;
09108	6.27 (K)	00-07-27	
09109	5.70 (K)	00-07-27	
09110	6.31 (K)	00-07-27	
TIPS: [Enter]Open[O]New[P]Search[N]Remane[DEL]De I			
		Ready	AUTO ALARM
NEW	Copy File	Delete	USB Rename

(Opening or deleting programs can be described in Section 6.2 or 6.3.)

6.6.2 Number of stored programs

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

6.6.3 Storage Capacity

The program storage capacity of this system (total capacity is 56M), this item shows the current occupied storage capacity and remaining storage capacity.

Chapter 7 Tool to tool and wear compensation

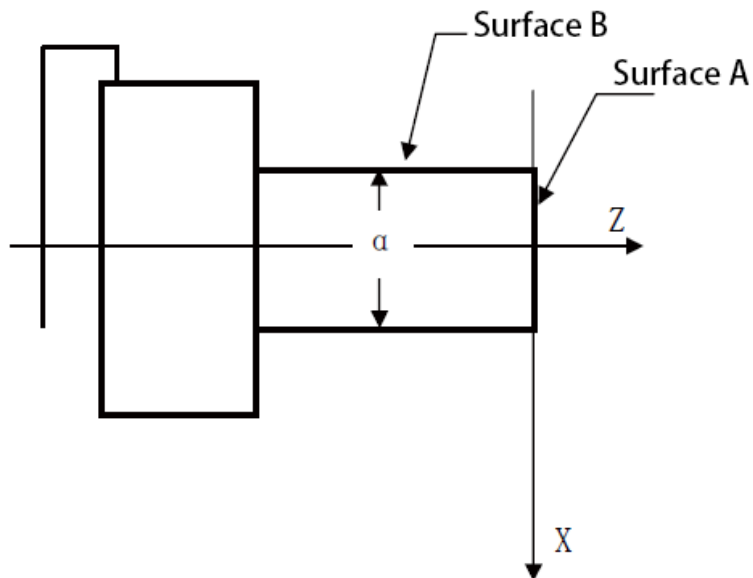
7.1 Tool to Tool

The operation steps are as follows (establish the workpiece coordinate system with the workpiece end face):

Note: The tool length input is divided into two types: absolute input and incremental input,

and the input mode SPACE can be switched by key;

Note: The knife length input must be entered with the upper axis letter;



1. Select any knife to cut along the A surface;
2. Exit the tool along the X axis without moving the Z axis, and stop the spindle rotation;
3. Press the  key to enter the compensation interface, and then click the F1 key

"knife length" under the screen to enter the knife length input interface, press the  key

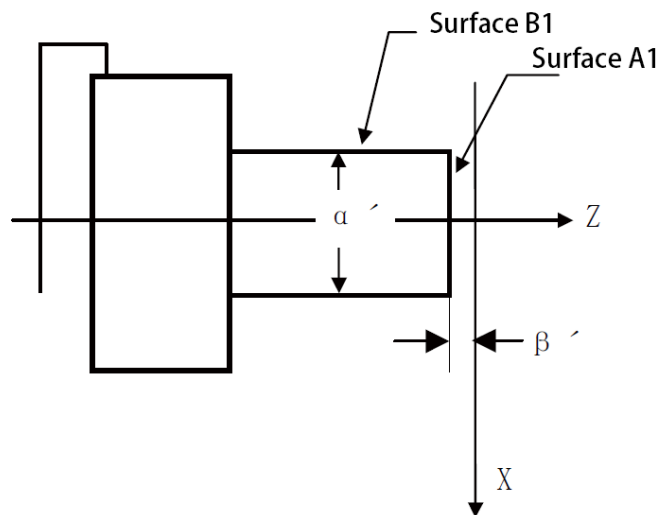
and  key to move the cursor to select the offset number corresponding to the tool;

4. Type the address key , number key  and  key in turn;

5. Make the tool cut along the B surface;
6. When the X axis is not moving, exit the tool along the Z axis and stop the spindle rotation;
7. Measuring diameter " α " (assuming $\alpha=15$);
8. Press the  key to enter the compensation interface, and then click the F1 key
- "knife length" under the screen to enter the knife length input interface, press the  key
- and  key to move the cursor to select the offset number corresponding to the tool;

9. Type the address key , number key ,  and  key in turn;

10. Move the tool to a safe tool change position and change another knife;



11. Make the tool cut along the A1 surface;
12. Exit the tool along the X axis without moving the Z axis, and stop the spindle rotation;
13. Measure the distance between the surface of A1 and the origin of the workpiece coordinate system " β " (assuming $\beta'=1$);

14. Press the  key to enter the compensation interface, and then click the F1 key
- "knife length" under the screen to enter the knife length input interface, press the  key
- and  key to move the cursor to select the offset number corresponding to the tool;

15. Press the address key , number key  and  key in turn;
16. Make the tool cut along the surface of B1 ;
17. In the case of the X axis not moving, exit the tool along the Z axis and stop the spindle rotation;
18. Measurement distance " α' " (assuming $\alpha'=10$);
19. Press the  key to enter the compensation interface, and then click the F1 key
"tool length" under the screen to enter the tool length input interface, press the  key and
 key to move the cursor to select the offset number corresponding to the tool;
20. Type the address key , number key ,  and  key in turn;
21. Repeat step 10 ~ 20 for other tool setting methods.

Note 1: The tool complement value of this pair method may be large, so the CNC must be set to perform the tool complement in a coordinate offset mode (the BIT4 bit of the CNC parameter NO.003 is set to 1), and the first program segment performs tool length compensation with T code or the first move code segment of the program contains T code that performs tool length compensation.

Note 2: Pay attention to the positive and negative directions when entering X-axis coordinates.

Note 3: The machine tool must be installed with the machine zero point switch to perform the zero point tool operation back to the machine tool.

Note 4: After returning to the zero point of the machine tool for tool set, the G50 code cannot be executed to set the workpiece coordinate system.

Note 5: The CNC must be set to perform knife compensation in a coordinate offset manner (the BIT4 bit of the CNC parameter NO.003 is set to 1), and the first program segment is used The T-code performs tool length compensation or the first move code segment of the program contains the T-code that performs the tool length compensation.

7.2 Wear compensation

In order to prevent the wrong operation when setting and modifying the tool bias value (no decimal point is entered, the decimal point position is not equal), resulting in excessive

modification of the tool offset value, resulting in knife collision and other phenomena, which is conducive to the operator to intuitively judge the wear degree of each knife, the lathe system sets the tool wear page. When the machining size is not accurate due to tool wear and other reasons and the tool complement value needs to be modified, it can be set or modified in the tool wear amount. The input range for machining tool wear values is set by the data parameter No 140. Tool wear data is saved when power is lost.

Note: The tool wear value input is divided into two types: absolute input and incremental

input, and the input mode can be switched by key  **;**



The operation steps are as follows:(taking the No. 1 tool length offset as an example: the X-axis input is 0.0 5, and the Z-axis input is -0 05)

1) Click the panel key to enter  the wear page;

2) Click the F2 key "Wear" under the screen to enter the wear input interface;

3) Use keys  and keys to  move the cursor to offset No. 1 under X-axis wear;

4) Enter the number key , character , number key ,  and  key to confirm the input;

5) Use the  keys to move the cursor to the Z-axis wear of the No. 1 bias;

6) Enter  symbols, numeric keys , characters , numeric keys ,  and  keys in turn to confirm input;

Note: Tool wear value input has positive and negative values, when entering positive values, do not add symbols, when entering negative values, you need to use the symbol key to enter "-" sign + number.



Chapter VIII Automatic Operation

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

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

8.1 Auto-Run

8.1.1 Selection of Runners

Local Files program opens:

- 1) Operate in a non-running state;



- 2) Press the  key, and enter the program content display screen;
- 3) Press the F2 key "File Management" under the screen to enter the local file directory

table interface;

4) Press the  or  key to find the desired program on the display screen (or use the "Find" program name to find it);

5) Press the  key to open the corresponding program;

6) Press the F1 key under the screen "Perform Processing" to enter the program to run state;

Note: When moving the selection in step 4, 11 lines of program content will be displayed on the right side of the screen for the operator to browse.

8.1.2 Startup of Autorun

1. Press the  key to select the automatic operation mode;

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

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

8.1.3 Stop of Auto-Run

* Code stop (M00).

1、 M00

After the program segment containing M00 is executed, it stops running automatically, and

all modal functions and states are saved. After pressing the panel  button or the external run key, the program continues.

2、 M01



Press the  key, the select stop indicator is on, and the stop function is active. After executing the program segment containing M01, the automatic operation is stopped, and the



modal functions and states are all saved. After pressing the  button or the external run key, the program continues.

* Press the relevant key to stop



1. After pressing the button or external pause button  in automatic operation, the machine tool is in the following state:

- (1) Machine tool feed deceleration stop;
- (2) Modal functions and states are saved;



(3) After pressing the  button, the program continues to execute.



2. Press the reset button

(1) All axis movements stop;



(2) The output of M and S functions is invalid (whether the  spindle can be automatically turned off after pressing the key, lubrication, cooling and other signals);

(3) The automatic operation ends, and the modal function and status are maintained.

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

4. When the conversion operation mode is converted to the machine tool zeroing, handwheel/single-step, manual, and program zeroing mode during automatic operation, the current program segment is immediately "paused"; When switching to edit and enter mode during automatic run, "Pause" is displayed only after the current program segment has finished

running.

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

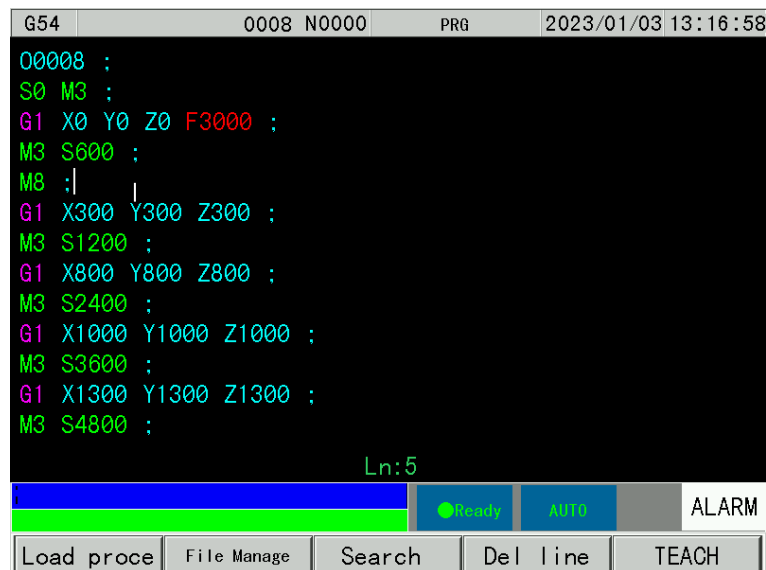
Note 2: Pressing the emergency stop button before power-up and shutdown reduces the electrical shock of the device;

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

Note 4: External emergency stop is valid only if the Bit2 bit (EALM) of the status parameter No.215 is set to 0.

8.1.4 Run automatically from any segment

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 program segment that is ready to start running (e.g. starting from the seventh line, move the cursor to the beginning of the seventh line);

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



3. Press the  key to enter the automatic operation mode, and press  the button to start the program running.



8.1.5 Feed, fast speed adjustment

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

* Adjustment of feed rate



Press the  key to achieve 16-level real-time adjustment of feed rate.

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

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

* Quick magnification adjustment



Press the  key to achieve fast rate F0, 25%, 50%, 100%

real-time adjustment.

Note 1: CNC parameters No.022 and No.023 set the fast moving rate of the X and Z axes respectively;

X-axis actual fast movement rate = No.022 set value × fast magnification

Z axis actual fast movement rate = No.023 set value × fast magnification

Note 2: When the fast magnification is F0, the minimum rate of fast movement is set by CNC parameter No.032.

8.1.6 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  key to adjust the spindle magnification to change the spindle speed, which can realize 8 levels of real-time adjustment of spindle magnification 50% ~ 120%.

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

Example:

CNC parameter NO.037 is set to 9999, execute S9999 code, select the spindle magnification to 100%, then the actual output analog voltage $\approx 10 \times 100\% = 10V$

8.2 Status at runtime

8.2.1 Single-segment operation

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

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



Press the  key to make the single-stage operation indicator in the status indication



area turn on , indicating that the single-stage operation function is selected;

When running in a single segment, the CNC stops running after the current program



segment is executed; When proceeding to the next program segment, press the  button again, and so on until the program finishes running.

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

Note 2: When executing fixed-loop G90, G92, G94, G70 ~ G76 code, the single-segment status is described in the first "Programming Description";

Note 3: The execution call subroutine (M98) and subroutine call return code (M99) single program segment are invalid. However, in the M98 and M99 program segments, except for addresses other than N, O, P, the single segment stops working.

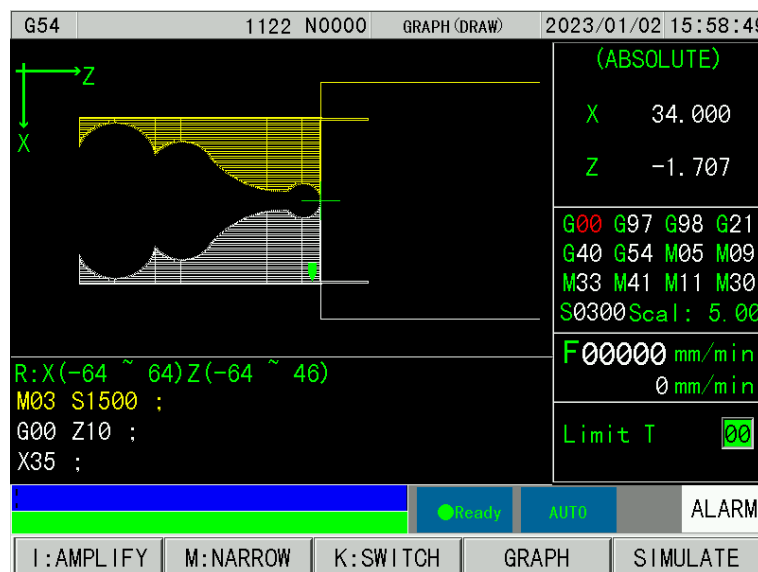
8.2.2 Empty run

Before automatically running the program, in order to prevent programming errors from unexpected or complex movements, after the program is completed, you can choose the graphical simulation to verify the program, and the operation method is as follows:

- 1) Press the  key to switch to the main interface of the program supervisor;
- 2) Press the F2 key "Graphic" function under the screen to enter the graphical display interface;
- 3) Press the F5 key "Graphic Simulation" under the screen, at this time the graphic display area will start to draw the processing path diagram according to the programmed motion trajectory;

Note: 1. After the graphical simulation starts, the system will automatically detect the grammar, and if there are unreasonable places or editing errors in the program, an alarm will be prompted;

2. The system contains the function of one-click graphics generation, and pressing the F4 key "Graphic Generation" under the screen in step 3 above will quickly generate the processing path diagram;



8.2.3 The machine is locked for operation

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

Press the  key to make the machine tool lock running indicator in the status

indication area  on, indicating that the machine tool lock running state has entered;

Machine locking operation is often used in conjunction with the auxiliary function locking

function for program calibration. Machine locked operation:

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

8.2.4 Accessibility locks operation

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

Press the  key to make the auxiliary function lock running indicator in the status indication area  turn on, indicating that the auxiliary function lock running state has been entered; At this time, the M, S, T code is not executed, and the machine drag plate moves. It is usually used in conjunction with the machine locking function for program calibration.

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

8.2.5 Program segment selection

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

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

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

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

8.3 Other Operations

1. In automatic operation mode, press the  key to switch coolant on/off;
2. Press , , , ,  or  any key in the key to realize the conversion of the operation mode;
3. Press the  key to reset the CNC.
4. Automatic lubrication function (see Chapter 3 of this article for details).

Chapter 9 Zero Return Operation

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

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

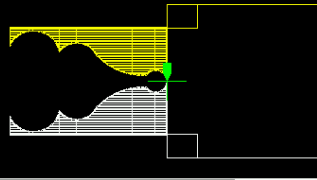
9.1 Program Returns to Zero

9.1.1 Program Zero

When the part is clamped to the machine tool, the absolute coordinate of the current position of the tool is set with G50 code according to the relative position of the tool and the workpiece, and the workpiece coordinate system is established in the CNC. The current position of the tool is called the program zero point, and it returns to this position after the program zero operation.

9.1.2 Procedure for program zeroing

1. Press the key to enter the program return to zero operation mode, the  bottom line of the display page displays the words "program back to zero", the page is shown as follows:

G50	00852 N0000		MONITOR	2023/01/03 13:47:25			
(ABSOLUTE)		(REM DIST)		G Code	G00	Pro Time 000:00:00	
X	0.000	X	0.000	G18	G97 G98	All Time 000:00:00	
Z	0.000	Z	0.000	G21	G40 G50	G00Rate 50 %	
				M05	M09 M33	G01Rate 100 %	
				M41	M11 M30	SPI Rate 100 %	
F	300mm/min	S	300 RPM	Workeds	0	T 0101	
(True)	0mm/min	(True)	0 RPM	All Num	0	Start In 1	
M03 S1500 ;							
G00 Z10 ;							
X35 ;							
Z0 ;							
G71 U0.5 R0.5 F200 ;							
Ln:1							
				●Ready PRG ZERO			
				ALARM			
Load edit		GRAPH		MDI		Info/Set	Wear set

2. Press any arrow key of the X, Z, and Y axes to return to the X, Z or Y axis program zero point;

3. The machine tool axis moves along the direction of the program zero point, and after returning to the program zero point, the axis stops moving, and the indicator light at the end of returning to zero is on.



Indicator indicating the end of return to zero

Note 1: After the zero point operation of the back program, the current tool offset state is not changed, and if the tool is offset, the position returned is the position set with G50 The position containing the tool offset is the position.

9.2 Machine tool returns to zero

9.2.1 Machine tool zero

Machine tool coordinate system is the reference coordinate system of CNC for coordinate calculation, is the inherent coordinate system of the machine tool, the origin of the machine tool coordinate system is called the machine tool zero point (or machine tool reference point), the

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

9.2.2 Operation steps of machine tool zeroing



1. Press the key to enter the machine tool zero return operation mode, and the bottom line of the display page displays the words "machine tool back to zero", which is displayed as follows:

G50		00852 N0000		MONITOR		2023/01/03 13:56:47	
(ABSOLUTE)		(REM DIST)		G Code G00		Pro Time 000:00:00	
X	0.000	X	0.000	G18 G97 G98	All Time 000:00:00		
Z	0.000	Z	0.000	G21 G40 G50	G00Rate 100 %		
				M05 M09 M33	G01Rate 100 %		
				M41 M11 M30	SPI Rate 100 %		
F	300mm/min	S	300 RPM	Workeds	0	T	0101
(True)	0mm/min	(True)	0 RPM	All Num	0	Start In	1
M03 S1500 ;							
G00 Z10 ;							
X35 ;							
Z0 ;							
G71 U0.5 R0.5 F200 ;							
Ln:1							
				●Ready		MACH	
						ALARM	
Load edit		GRAPH		MDI		Info/Set	
						Wear set	

2. Press the , ,  or  key to select back to the zero point of the X, Z, Y or 4th axis machine tool;

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



Indicator indicating the end of return to zero

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

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

- 1) Move out from the zero point;
- 2) CNC power off;

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

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

Note 5: After performing the machine zero return operation, the original workpiece coordinate system is reset and needs to be set again with G50.

9.3 Other operations in zero-back mode

1. Press the  key, the spindle turns counterclockwise;
2. Press the  key to stop the spindle;
3. Press the  key, the spindle turns clockwise;
4. Press the  key to switch coolant on/off;
5. Lubrication control (see Chapter 3 of this article for details);
6. Press the  key to manually change the knife;
7. Trimming of spindle magnification;
8. Repair of fast magnification;
9. Adjustment of feed rate.

Chapter 10 Data Setting, Backup and Recovery

10.1 Setting of Data

10.1.1 Switch Settings

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



1. Press the left/right button under the screen to find the F1 key "Settings", and click to enter the setting page;

2. Press the F5 key "Switch" under the screen to enter the switch setting interface;

3. Press the  key "*" to move left to turn off the switch, press the  key "*" to move right to open the switch;

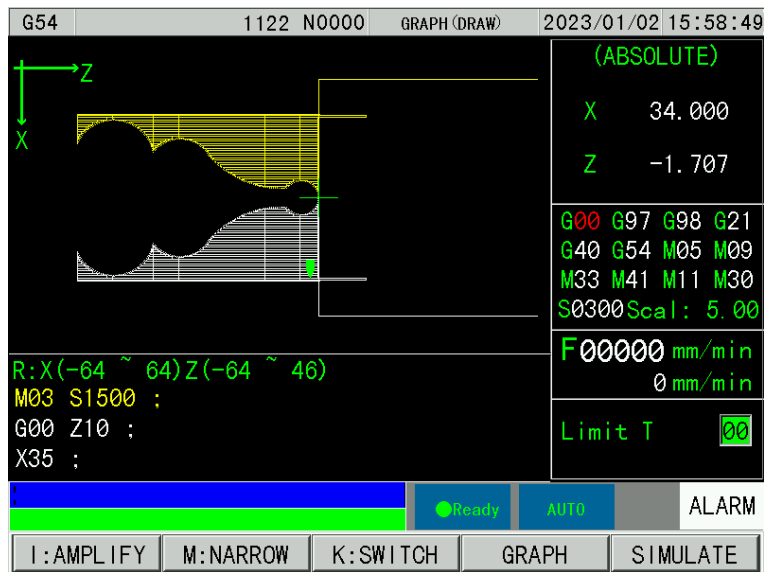
Parameters can only be modified when the parameter switch is on; Programs can only be edited when the program switch is on; The sequence number of the program segment is automatically added only when the automatic sequence number switch is turned on.

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

button  can eliminate the alarm, if the switch state of the parameter is switched again, the alarm will not be alarmed. For safety reasons, after the parameter modification is completed, be sure to set the parameter switch to "Off".

10.1.2 Graphics Settings

(See Section 8.2.2 above)



The meaning of the graphical parameters

Coordinate system settings: According to the different coordinate systems of the front and rear tool holders, the lathe system can display 8 kinds of graphic trajectory A: figures

The zoom in and out of the shape trajectory can be zoomed in and out in real time through the F1 key "zoom in" and the F2 key "zoom out" under the screen; Or "toggle" the coordinate display via the function key F3.

B: Graphic generation, by pressing the F4 key "Graph Generation" can generate the program trajectory diagram with one click;

C: The movement of the graphical trajectory display

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

10.1.3 Setting Parameters

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

Press the left/right page key under the screen to find "Parameter Settings", and click the F2 key "Parameter Settings" under the screen to enter the parameter setting interface, as shown in the following figure:

A. Modify the status parameters

1. Modify by bit:

1) Turn on the parameter switch (see section 10.1.1);

2) Press the left/right page turning button under the screen to find "Parameter Settings", and click the F2 key "Parameter Settings" under the screen to enter the parameter setting interface;

3) Press the F4 key "bit parameter" under the screen to enter the status parameter setting

interface;

Method 1: Press  or  key to the page where the parameter to be set is located, and press

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

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

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

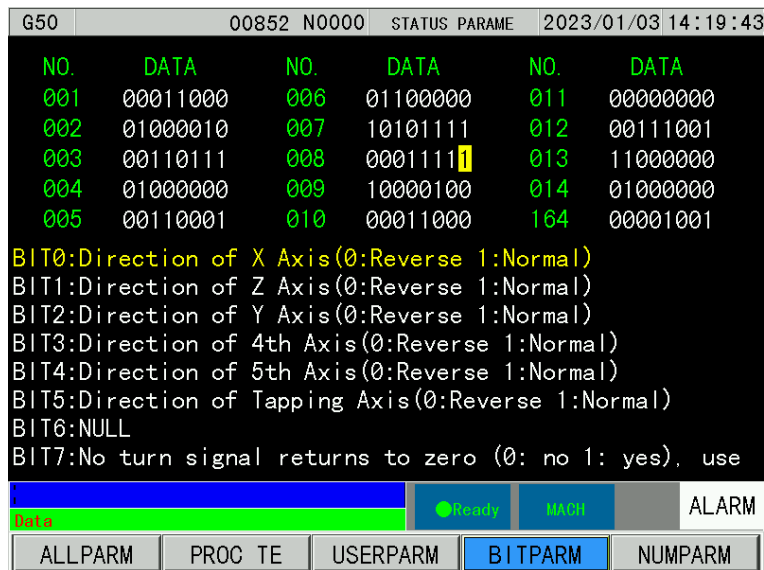
5) For safety reasons, after all parameters are set, the parameter switch needs to be turned off.

Example:

Set Bit0 (DECI) of the status parameter No 008 to 1, leaving the rest unchanged. Follow the above steps to move the cursor to No 008, and then use Move to the right Bit0 bit, as shown in the following figure:

G50	00852	N0000	STATUS	PARAME	2023/01/03 14:18:15
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000010	007	10101111	012	00111001
003	00110111	008	00011110	013	11000000
004	01000000	009	10000100	014	01000000
005	00110001	010	00011000	164	00001001
BIT0:Direction of X Axis(0:Reverse 1:Normal)					
BIT1:Direction of Z Axis(0:Reverse 1:Normal)					
BIT2:Direction of Y Axis(0:Reverse 1:Normal)					
BIT3:Direction of 4th Axis(0:Reverse 1:Normal)					
BIT4:Direction of 5th Axis(0:Reverse 1:Normal)					
BIT5:Direction of Tapping Axis(0:Reverse 1:Normal)					
BIT6:NULL					
BIT7:No turn signal returns to zero (0: no 1: yes), use					
data		Ready		MACH	
ALLPARM		PROC TE		USERPARM	
		BITPARM		NUMPARM	
				ALARM	

The parameters are modified. The page is displayed as follows:



B. Modification settings of data parameters and screw supplementary data

- 1) Turn on the parameter switch (see section 10.1.1);
- 2) Press the left/right page turning button under the screen to find "Parameter Settings", and click the F2 key "Parameter Settings" under the screen to enter the parameter setting interface;
- 3) Press the F5 key "Number Parameters" under the screen to enter the status parameter setting interface;

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

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

- 4) Enter the new parameter value, the parameter value is entered and displayed;
- 5) For safety reasons, after all parameters are set, the parameter switch needs to be turned off.

Example 1:

Set the data parameter No 022 to 80 00. Move the cursor over No 022 as described above, and type 80 00 in the prompt line, as shown in the following figure

G50	00852 N0000	NUM PARAMETER	2023/01/03 14:25:48
NO.	MEANING	DATA	
015	X multiplication coefficient (CMR)	10	
016	Z multiplication coefficient (CMR)	10	
017	X command frequency division coefficient (CMD)	10	
018	Z command frequency division coefficient (CMD)	10	
019	Back tail length TCH when thread cutting (back width=TCH)	5	
020	Null	0	
021	Offset comp. of spindle analog voltage output voltage	0.0000	
022	X-Axis rapid movement speed(radius)	5000	
023	Z-Axis rapid movement speed	5000	
024	S front acc/dec time setting when X rapid movement	100	
025	S front acc/dec time setting when Z rapid movement	100	
026	Acc/dec time constant of short axis when thread back ta	60	
MIN	0	MAX	99999
		P:1/33	
Data		●Ready	MACH
		ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM
		NUMPARM	

The parameters are modified. The page is displayed as follows:

G50	00852 N0000	NUM PARAMETER	2023/01/03 14:27:06
NO.	MEANING	DATA	
015	X multiplication coefficient (CMR)	10	
016	Z multiplication coefficient (CMR)	10	
017	X command frequency division coefficient (CMD)	10	
018	Z command frequency division coefficient (CMD)	10	
019	Back tail length TCH when thread cutting (back width=TCH)	5	
020	Null	0	
021	Offset comp. of spindle analog voltage output voltage	0.0000	
022	X-Axis rapid movement speed (radius)	8000	
023	Z-Axis rapid movement speed	5000	
024	S front acc/dec time setting when X rapid movement	100	
025	S front acc/dec time setting when Z rapid movement	100	
026	Acc/dec time constant of short axis when thread back ta	60	
MIN	0	MAX	99999
		P:1/33	
Data		●Ready	MACH
		ALARM	
ALLPARM		PROC TE	USERPARM
		BITPARM	NUMPARM

Note: Under the level 1 operation password, press the panel key



key to switch to modify the maximum and minimum value range of the corresponding parameter value;

C. Processing process interface

- 1) Turn on the parameter switch (see section 10.1.1);
- 2) Press the left/right page turning button under the screen to find "Parameter Settings", and click the F2 key "Parameter Settings" under the screen to enter the parameter setting interface;

3) Press the F2 key "Processing Technology" under the screen to enter the processing process setting interface;

G54	0008 N0000	PROC TE	2023/01/02 14:18:24
NO.	MEANING	DATA	
6100	Alter Offset during processing (0:Invalid 1:Valid)	1	
6110	(0:Diameter 1:Radius) Programming	0	
6130	Value of Tool Offset (0:Diameter 1:Radius)	0	
6140	Processing Mode Between blocks (0:Continuous 1:Exact)	1	
6150	Back tail length TCH when thread cutting (back width=TCH	5	
6160	Max.Value of Tool offset/Wear	50.0000	
6170	G71/G72 cycle turing single tool feeding value	0.0010	
6180	G71/G72 cycle turing sigle tool retraction value	0.0000	
6190	Tool retraction value of X when G73 cycle turing	0.0000	
6200	Tool retraction value of Z when G73 cycle turing	0.0000	
6210	Cutting count of G73 cycle turning	1	
6220	Tool retraction value of Z/X of G74/G75 cycle turing	0.0000	
P: 1 / 2			
		●Ready	JOG
		ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM
		NUMPARM	

Note: The processing process has two independent interfaces, which can be switched by

pressing the  and  key.

D. User parameter interface

- 1) Turn on the parameter switch (see section 10.1.1);
- 2) Press the left/right page turning button under the screen to find "Parameter Settings", and click the F2 key "Parameter Settings" under the screen to enter the parameter setting interface;
- 3) Press the F3 key "User Parameters" under the screen to enter the user parameter setting interface;

G54	0008 N0000	USERPARM	2023/01/02 14:21:16
NO.	MEANING	DATA	
7110	Processed total workpieces	0	
7111	Required process total workpieces	0	
7120	1/0:Yes/No alarm after reaching process numbe	1	
7130	Counting Of Pieces with M99 (0:NO 1:YES)	1	
7131	Counting Of Pieces with M31 (0:NO 1:YES)	0	
7140	Memory Pieces Number When Power Off (0:YES 1:NO)	0	
7141	Clear finished Count of Pieces (0:NO 1:YES)	0	
7210	Auto Clear Processing Time of Each piece (0:NO 1:YES)	1	
7220	1/0:reseting spindle lubricant and coolant ou	0	
7230	Lubrication open time(0 shows that Lubrication is not r	0	
7231	Auto lubricant interval time	3600	
7240	Cursor Return to Beginning after M30 (0:NO 1:YES)	1	
MIN	0	MAX	99999
		P: 1 / 2	
		●Ready	JOG
		ALARM	
ALLPARM	PROC TE	USERPARM	BITPARM
		NUMPARM	

Note: User parameters have two separate interfaces, which can be switched by pressing the



E. Parameter summary table interface

- 1) Turn on the parameter switch (see section 10.1.1);
- 2) Press the left/right page turning button under the screen to find "Parameter Settings", and click the F2 key "Parameter Settings" under the screen to enter the parameter setting interface;
- 3) Press the F1 key "Parameter General Table" under the screen to enter the parameter summary table setting interface;

Note: The parameter summary table is divided into 9 sub-pages, which are machine tool, speed, spindle, feed axis, limit, zero return, absolute value, tool, tapping/positioning, which can be switched by the function key at the bottom of the screen

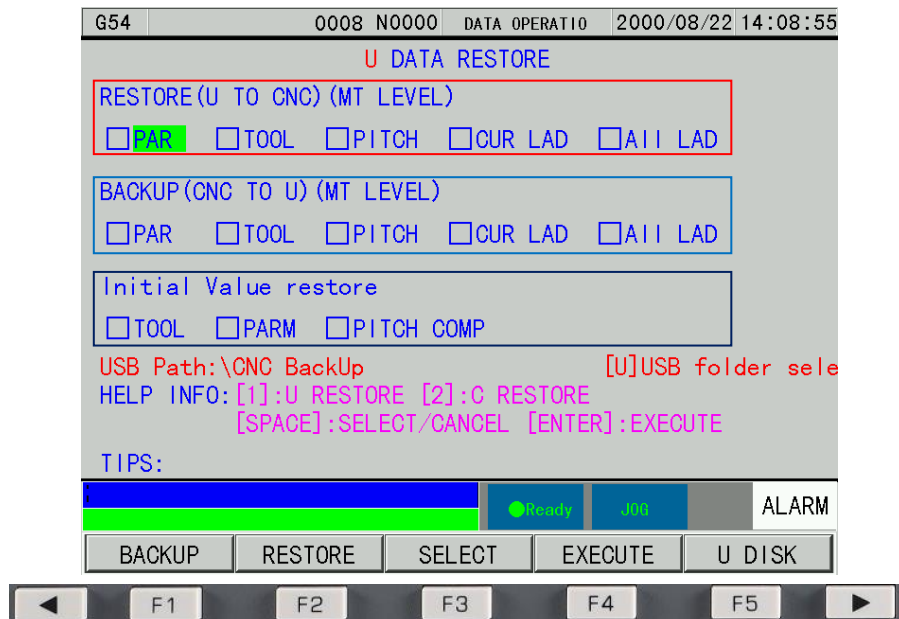


G50	00852 N0000	ALLPARM	2023/01/03 14:58:27
NO.	MEANING	DATA	
0010	Check E-Stop Alarm Signal (0:YES 1:NO)	1	
0020	Alarm Level of Spindle Driver (0:High 1:Low)	1	
0021	Alarm Level of X-axis Driver (0:High 1:Low)	0	
0022	Alarm Level of Z-axis Driver (0:High 1:Low)	0	
0023	Alarm Level of Y-axis Driver (0:High 1:Low)	1	
0024	Alarm Level of 4th-axis Driver (0:High 1:Low)	0	
0030	3-bits switch function valid/invalid	0	
0040	1/0:Each axis overtravel valid/invalid	0	
0050	Soft-Limit Before Switch Homing (0:Invalid 1:Valid)	1	
0060	Run G28 without homing (0:No Alarm 1:Alarm)	0	
0070	External Cycle Running (0:Valid 1:Invalid)	0	
0071	External Pause (0:Valid 1:Invalid)	1	
P: 1 /27			
		●Ready	MACH
		ALARM	
MACHINE	SPEED	SPINDLE	FEEDAXIS
LIMIT			

G54	00001 N0000	ALLPARM	2023/01/03 15:01:24	
NO.	MEANING	DATA		
0810	Number of controlled axis	2		
0811	Definition of 4th axis' name(3:A 4:B 5:C)	3		
0812	Definition of 5th axis' name(3:A 4:B 5:C)	5		
0830	Alarm Level of Spindle Driver (0:High 1:Low)	1		
0831	Alarm Level of X-axis Driver (0:High 1:Low)	0		
0832	Alarm Level of Z-axis Driver (0:High 1:Low)	0		
0833	Alarm Level of Y-axis Driver (0:High 1:Low)	0		
0834	Alarm Level of 4th-axis Driver (0:High 1:Low)	0		
0860	X multiplication coefficient (CMR)	10		
0861	X command frequency division coefficient (CMD)	10		
0862	Z multiplication coefficient (CMR)	10		
0863	Z command frequency division coefficient (CMD)	10		
MIN 2 MAX 5		P:13 /27		
		●Ready MDI ALARM		
MACHINE	SPEED	SPINDLE	FEEDAXIS	LIMIT

10.2 Data Restoration and Backup

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



1. Turn on the parameter switch;
2. Press the left and right switch keys at the bottom of the screen to find the F1 key "Settings", and click the F1 key to enter the setting interface;
3. Press the F2 key "Backup" under the screen to enter the system parameter backup and restore interface;

4. Press the  key or  key to switch between "U disk data restore to CNC", "CNC data backup to U disk", "initial value restore operation";

5. Press the  key or  key to move the cursor left and right to the options;
6. Press the F3 key under the screen to "select" the required data items;
7. Press the F4 key "Perform Operation" under the screen to start restoring or backing up;

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

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

10.3 Setting and Modifying Permissions

In order to prevent the processing program and CNC parameters from being maliciously

modified, the lathe system has arched the permission setting function, and the password level is divided into 4 levels, from high to low, they are 2 levels (machine tool manufacturer level), 3 levels (set H Management level), level 4 (technician level), level 5 (machining operation level), the current operation level of the CNC is displayed by the current operation level: " on the permission setting page .

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

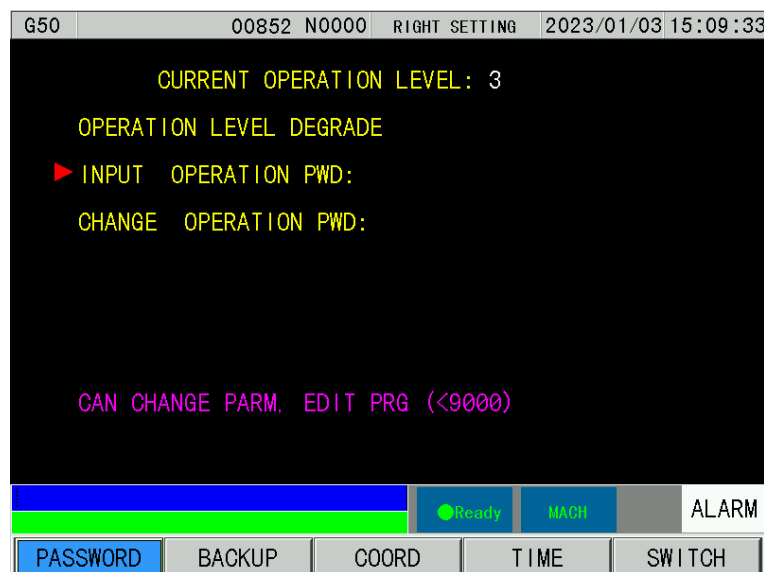
Level 3: The initial password is 12345, which allows modification of CNC status parameters, data parameters, knife complement data, and editing part programs.

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

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

Steps to go to the page:

1. Press the left and right switch keys at the bottom of the screen to find the F1 key "Settings", and click the F1 key to enter the setting interface;
2. Press the F1 key "Password" under the screen to enter the system password setting interface;



After entering the permission setting page, the cursor first stops on the line "Enter

operation password". You can press the  key or  move the cursor to the corresponding action for "Action Level Demotion", "Enter Operation Password", "Change Operation Password".

10.3.1 Entry to the Operation Level

1. After entering the permission setting page, move the cursor to the "Enter operation password" line;
2. Enter the operation password (for each number entered, the display will add a "*" sign);

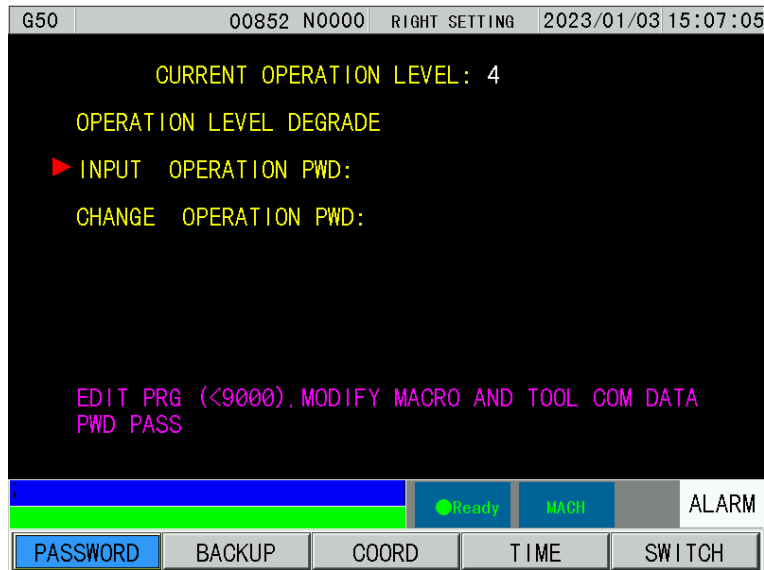
3. After entering and pressing the  key, you can enter the operation level corresponding to the password.

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

The details are as follows:

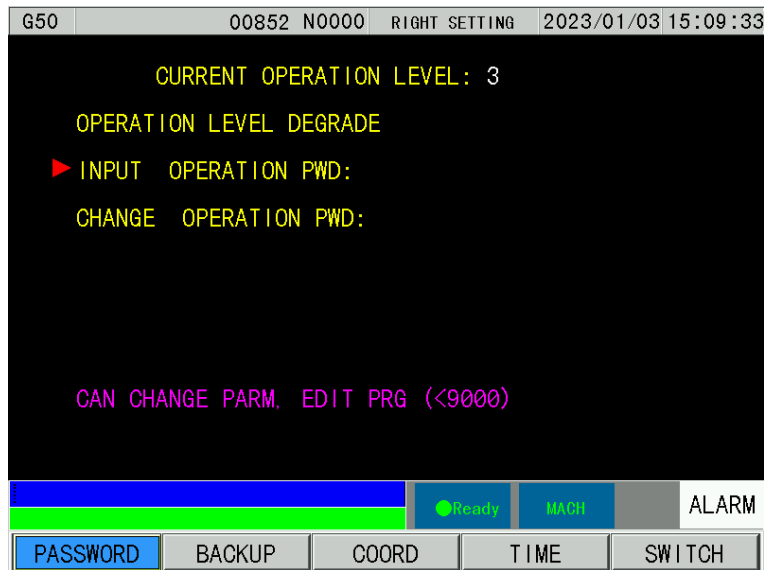
The level of	Password data length	Initial
3 level	5 digits	12345
4 level	4 digits	1234
5 level	not	not

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



Move the cursor to the "Enter operation password" line, enter 12345, press the key, CNC prompt "Parameters can be modified, programs can be edited." "

Password passed, the current operation level is changed to level 3 . The display page is as follows:



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

10.3.2 Change of Operation Password

To change your password, follow these steps:

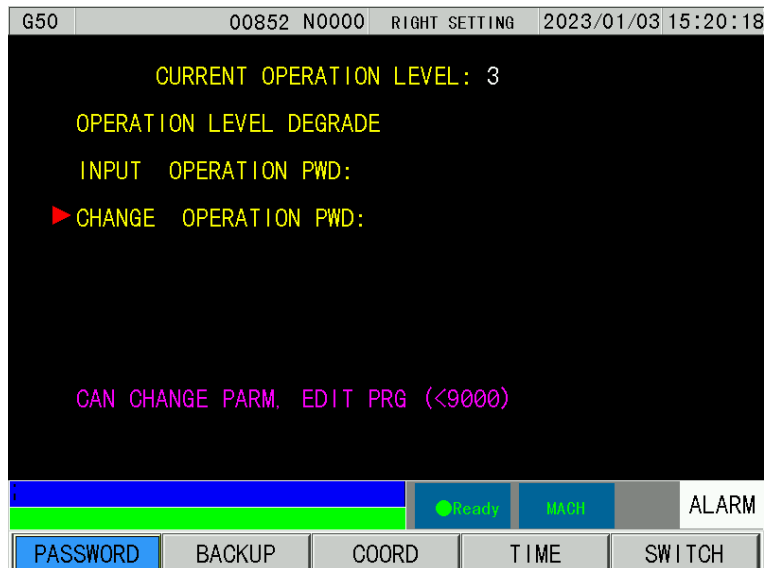
1. After entering the permission setting page, enter the password according to the method

steps described in Section 10.3.1;

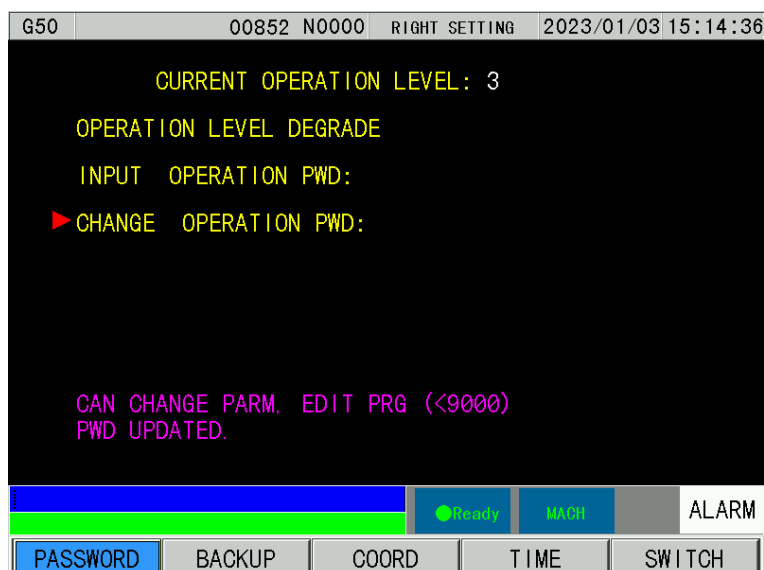
2. Move the cursor to the "Change Operation Password" line;

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

4. The CNC prompts "Please enter the new password again", and the display page is as follows:



5. After entering the operation password again, press the 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 password entered twice is different, the CNC prompts "Change password

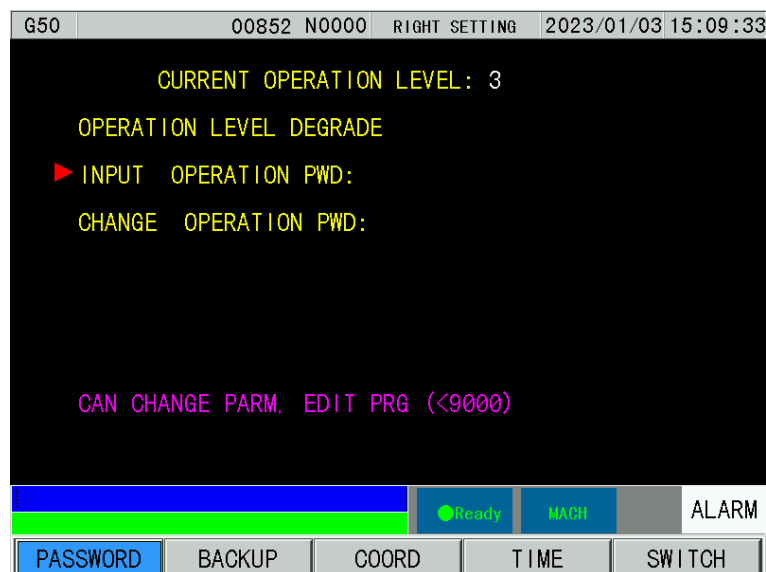
verification does not match, please re-enter".

10.3.3 Downgrading the Operation Level

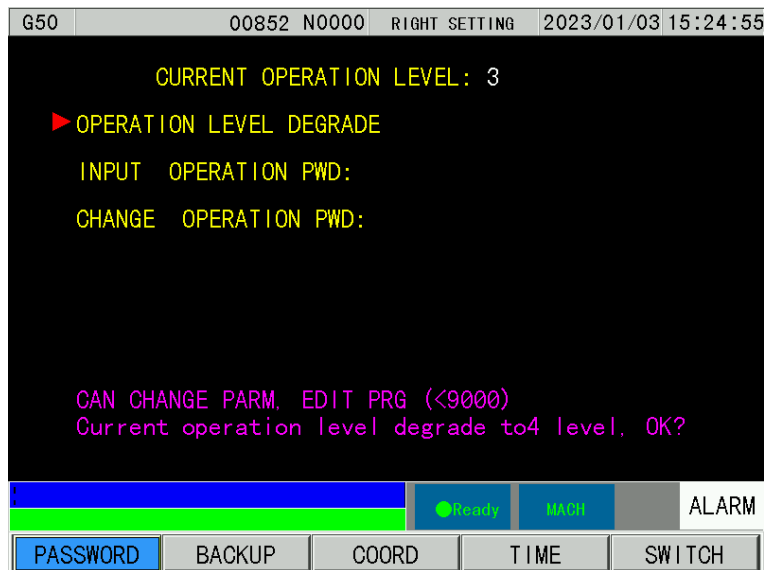
Action level downgrading makes it easy to downgrade from a higher level of action to a lower level of operation as follows:

1. After entering the permission setting page, enter the password according to the method steps described in Section 10.3.1;

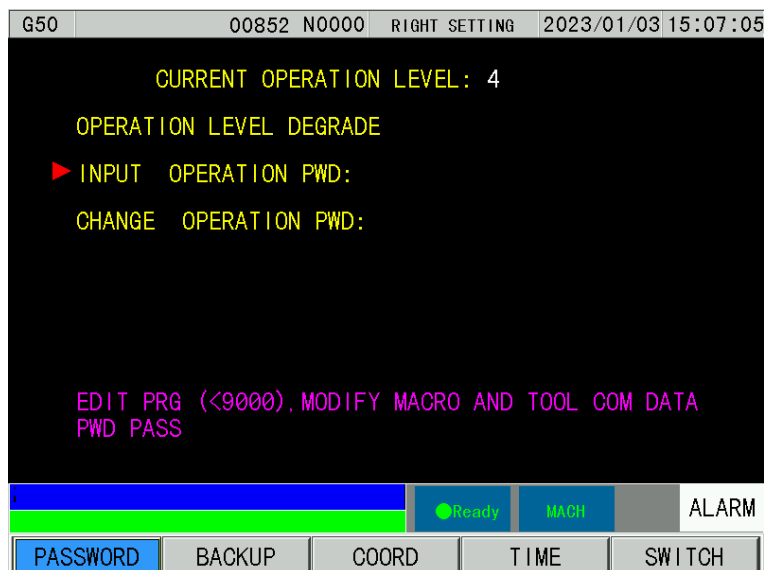
2. Move the cursor to the "Operation Level Demotion" line, if the current operation level of the CNC is level 3, the display page is as follows:



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



4. Press the  key again (the operation level is downgraded successfully, and the display page is as follows).



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

Chapter 11 U disk operation functions

11.1 File Directory Page

To enter the U disk directory, follow these steps:

1. Press the  key to enter the program page;
2. Press the F2 key under the screen to enter the [File Management] interface;
3. Press F4 "U disk directory" under the screen to enter the U disk directory interface, and

insert the U disk as shown in the figure:



The top of the page displays the CNC disk directory information, the bottom shows the USB disk directory information, if the USB disk is not detected, the lower display bar does not display the content.

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

Press the  key or the F2 key under the screen to "switch" the cursor will switch from

the CNC drive to the USB drive, press the  key (F3 key "previous line" under the screen)

or  key (F4 key "next line" under the screen). You can move the cursor.

11.2 File Copying

Move the cursor to the file to be copied, and press the F1 key under the screen to copy the "Save to U Disk" key.

Note: When the CNC disk and USB disk are switched, the function corresponding to the F1 key under the screen will be changed accordingly;

Chapter 12 Processing Examples

To process the workpiece shown in the figure below, the bar stock size is $\Phi 136\text{mm} \times 180\text{mm}$.

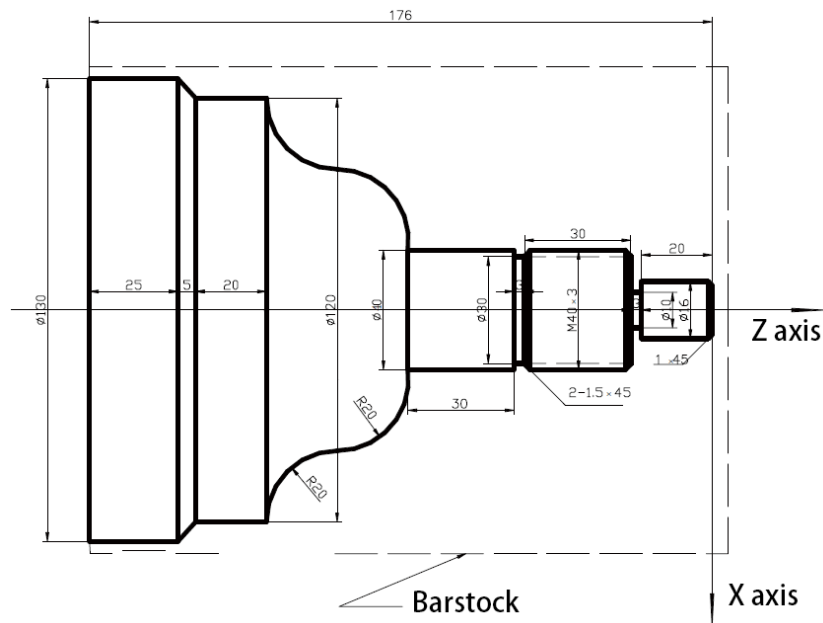


Figure 14-1

Machining with 4 knives, as follows:

Cutter number	Tool Type	Explain
1 knife		Outer round rough turning tool

Cutter number	Tool Type	Explain
2 knife		External round fine turning tool
3 knife		Slotting knife, knife width 3mm
4 knife		Thread turning tool, knife Angle 60°

12.1 Programming

According to the machining process and the code interpretation of this manual, the workpiece coordinate system shown in Figure 14-1 is established, and the editing program is as follows:

```

O 0 0 0 1 ; //Part program name
N 0 0 0 0 G0 X150 Z50; // Position to a safe position for tool change
N 0 0 0 5 M12; // Clamp chuck
N 0 0 1 0 M3 S800; // Open spindle, speed 800
N 0 0 2 0 M8; // Turn on the coolant
N 0 0 3 0 T0101; // Change the first knife
N 0 0 4 0 G0 X136 Z2; //Depth of cut close to workpiece 1mm, retreat 1mm
N 0 0 5 0 G71 U0.5 R0.5 F200; // 0.5mm margin for X-axis and 0.5mm for Z-axis
N 0 0 5 5 G71 P0060 Q0150 U0.25 W0.5; // Close to the workpiece end face
N 0 0 6 0 G0 X16; // Car Φ16 outer circle
N 0 0 7 0 G1 Z-23; // Car end face
N 0 0 8 0 X39.98; // Car Φ39.98 outer circle
N 0 0 9 0 W-33; // Car end face
N 0 1 0 0 X40; // Car Φ40 outer circle
N 0 1 0 5 W-30; Car convex arc
N 0 1 1 0 G3 X80 W-20 R20; // Car concave arc
N 0 1 2 0 G2 X120 W-20 R20; // Car Φ120 outer circle
N 0 1 3 0 G1 W-20; // Car taper
N 0 1 4 0 G1 X130 W-5; // Car Φ130 outer circle
N 0 1 5 0 G1 W-25; // Rough turning is completed and the tool change point is returned
N 0 1 6 0 G0 X150 Z185;
N 0 1 7 0 T0202; // Change the No. 2 knife and perform the No. 2 knife deflection
N 0 1 8 0 G70 P0060 Q0150; // Fine car cycle
N 0 1 9 0 G0 X150 Z185; // Rough turning is completed and the tool change point is

```

returned

```

N 0 2 0 0    T0303;    // Change knife No. 3 and perform knife No. 3 deflection
N 0 2 1 0    G0 Z-56 X42;    // Close to the workpiece
N 0 2 2 0    G1 X30 F100;    // Cut Φ30 slots
N 0 2 3 0    G1 X37 F300;
N 0 2 4 0    G1 X40 W1.5;    // Return
N 0 2 5 0    G0 X42 W30;    // Chamfer
N 0 2 6 0    G1 X40;    // Make way the grooving knife width
N 0 2 6 2    G1 X37 W1.5;    // Chamfer
N 0 2 6 4    G1 X10;    // Cut Φ10 slots
N 0 2 6 6    G0 X17 Z-1;
N 0 2 6 8    G1 X16;
N 0 2 7 0    G1 X14 Z0 F200;    // Chamfer
N 0 2 8 0    G0 X150 Z50;    // Return to the tool change point
N 0 2 9 0    T0404 S100;    // Change the size 4 knife and set the spindle at 100 speed
N 0 3 0 0    G0 X42 Z-20;    // Close to the workpiece
N 0 3 1 0    G92 X39 W-34 F3;    // Thread cutting cycle
N 0 3 2 0    X38;    // Feed 1mm to cut the second knife
N 0 3 2 0    X37;    // Feed 1mm and cut the third cut
N 0 3 3 0    X36.4;    // Feed 0.6mm Cut the fourth knife
N 0 3 3 2    X36;    // Feed 0.4mm Cut the fifth knife
N 0 3 4 0    G0 X150 Z50;    // Return tool point
N 0 3 5 0    T0100 U0 W0;    // Swap back to knife No. 1 and execute the knife offset
N 0 3 6 0    M5;    // Off spindle
N 0 3 7 0    M9;    // Turn off the coolant
N 0 3 8 0    M13;    // Release the chuck
N 0 3 9 0    M30;    // End of program

```

12.2 Program Inputs

12.2.1 Viewing Existing Programs

System file directory into steps:



- 1、 Press the  key to enter the program interface;
- 2、 Press the F2 key "File Management" under the screen to enter the program directory page, which is displayed as follows:

G54	00001 N0000	PROGRAM (DIR)	2023/01/03 08:30:36
PRG CNT14	MEM 109 M	USED:0 M	FREE:109 M
NAME	SIZE	DATE	PROGRAM PREVIEW:
00001	70 (B)	23-01-02	M03S1500;
07002	241 (B)	23-01-03	G00 Z10;
(00008) 0008	377 (B)	00-08-22	X35;
09101	824 (B)	00-07-27	Z0;
09102	465 (B)	00-07-27	G71 U0.5 R0.5 F200;
09103	753 (B)	00-07-27	G71 P10 Q20 U0.3 W0.2;
09104	5.63 (K)	00-07-27	N10G01X0;
09105	6.25 (K)	00-07-27	G03X4.14Z-6.64R3.64;
09106	6.21 (K)	00-07-27	G02 X17.68 Z-23.02 R20.2
09107	5.70 (K)	00-07-27	G03 X19.32 Z-34.92 R7.12
09108	6.27 (K)	00-07-27	G03X23.08Z-50.98R8.52;
09109	5.70 (K)	00-07-27	
09110	6.31 (K)	00-07-27	
09201	3.03 (K)	00-07-27	
TIPS: [Enter]Open[0]New[P]Search[N]Remane[DEL]DeI			
		Ready	AUTO
NEW	Copy File	Delete	USB
Rename			

On the upper page, you can view the program name of the stored program in the CNC to prepare for the confirmation of the new program name, and the 11 paragraphs of program content are displayed on the right for easy browsing and confirmation.

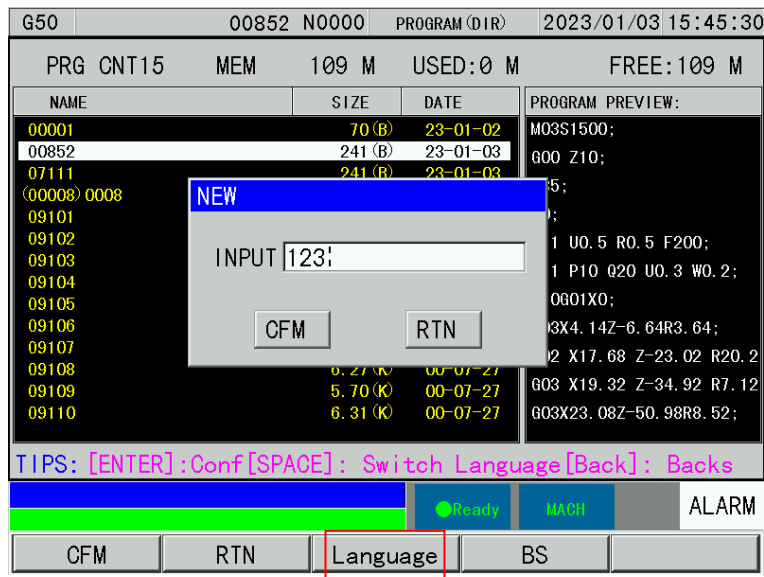
12.2.2 Establish new procedures

The steps to create a new program are as follows:



1. Press the  key to enter the program interface;
2. Press the F2 key "File Management" under the screen to enter the program directory page;
3. Press the F1 key "New" under the screen to pop up the new program name input dialog box;
4. Enter the file name (e.g. 123);
5. Press the F1 key "Confirm" under the screen to create a new program;

G54	00001 N0000	PROGRAM (DIR)	2023/01/03 08:30:36
PRG CNT14	MEM 109 M	USED:0 M	FREE:109 M
NAME	SIZE	DATE	PROGRAM PREVIEW:
00001	70 (B)	23-01-02	M03S1500;
07002	241 (B)	23-01-03	G00 Z10;
(00008) 0008	377 (B)	00-08-22	X35;
09101	824 (B)	00-07-27	Z0;
09102	465 (B)	00-07-27	G71 U0.5 R0.5 F200;
09103	753 (B)	00-07-27	G71 P10 Q20 U0.3 W0.2;
09104	5.63 (K)	00-07-27	N10G01X0;
09105	6.25 (K)	00-07-27	G03X4.14Z-6.64R3.64;
09106	6.21 (K)	00-07-27	G02 X17.68 Z-23.02 R20.2
09107	5.70 (K)	00-07-27	G03 X19.32 Z-34.92 R7.12
09108	6.27 (K)	00-07-27	G03X23.08Z-50.98R8.52;
09109	5.70 (K)	00-07-27	
09110	6.31 (K)	00-07-27	
09201	3.03 (K)	00-07-27	
TIPS: [Enter]Open[0]New[P]Search[N]Remane[DEL]DeI			
		Ready	AUTO
NEW	Copy File	Delete	USB
Rename			



Note: 1. The file name can be numbers, letters or Chinese (switch between Chinese and English mode by the F3 key "Chinese-English switch" under the screen).

2. The right side of the screen can display 11 paragraphs of program content for easy operation viewing.

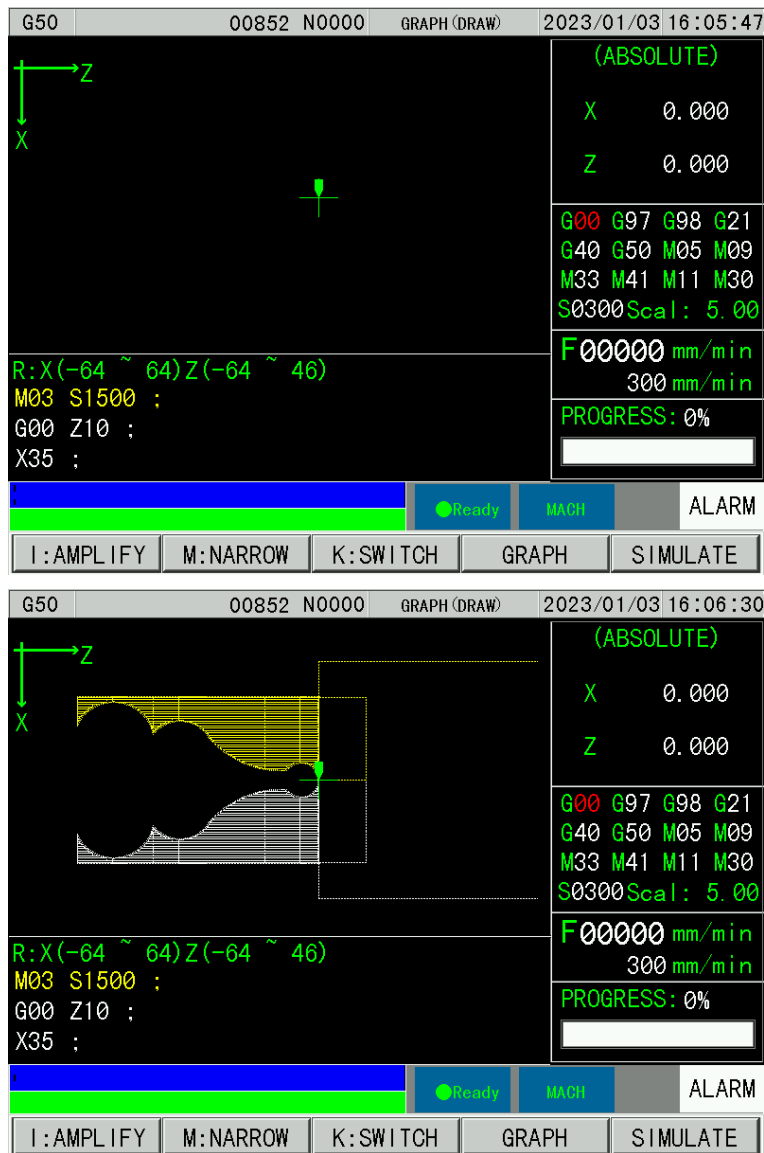
3. File management also includes F2 key "Program Copy", F3 key "Program Delete", F5 key "duplicate name", page down right F1 key "Size Sort", F2 key "Descending", F3 key "Search".

12.3 Program Verification

12.3.1 Graphics Settings



1. Press the  key to enter the graphical interface;
2. Press the F2 key under the screen The "Graphics" page is displayed as follows:



Note: 1. The graphical interface corresponds to the F key function under the screen, F1 key "zoom in", F2 key "zoom out", F3 key "switch", F4 key "graphic generation", F5 key "graphic simulation", F1 key "mirror display" after turning down to the right, F2 key "origin center", F3 key "clear", F4 key "pre-collision".



2. Use the movement arrow keys to move the graphic.

Zoom in: The graph is scaled up, and the equivalent scale is displayed in the right modal "Zoom:";

Zoom out: The graph is scaled down, and the equivalent scale is displayed in the right modal "Zoom:";

Switching: Observe the displayed path for the non-coaxial change of the graph;

Graph generation: one-click generation of the current program processing path (check whether the program syntax editing is reasonable when generating the path);

Graphical simulation: display the processing path diagram according to the programmed trajectory, and observe whether the motion path is correct in real time (check whether the

program syntax editing is reasonable at this time);

Mirror display: graphic X-axis symmetrical mirror display;

Center Origin: Quickly place the graphic in the middle of the display area;

Clear: clear the formed trajectory pattern with one click;

Pre-collision: After enabling this function, the graphical display area will display a two-axis soft limit limit prompt box;

12.3.2 Program Validation

Press the  key to enter the program supervision interface, and then press the F2 key under the screen to enter the graphic display page, press the  key to enter the automatic operation mode, press the  key and  key to make the auxiliary function in the status indication area lock the light , the machine tool lock light and the empty running indicator light on , enter the auxiliary function lock, machine tool lock and empty running state. By running  the program automatically, you can check the correctness of the program by displaying the trajectory of the tool movement, and the operation is completed.

If the displayed program trajectory is wrong, analyze the error in the program and modify the part program, and check again according to the above method until it is correct. Or press the F4 key "graphic generation" on the screen on the graphical interface, one-click shape or press the F5 key "graphic simulation" graphic gradually form the motion trajectory verification view;

12.4 Tooling, Running and Wear

The knife length interface enters:

- 1、 Press the panel  key to enter the knife length offset compensation page;
- 2、 Press the F1 key "Tool Length" under the screen to enter the tool length input interface;

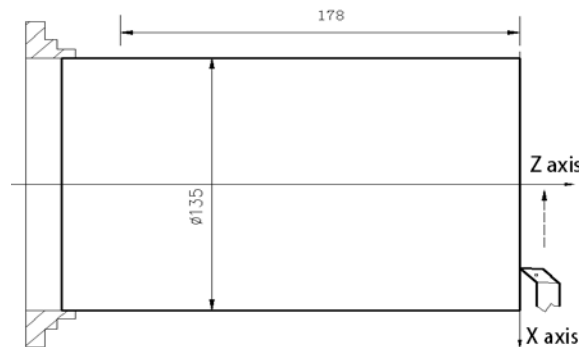
Note: The button  can switch the absolute value or incremental input mode of the

knife length;



1. Move the tool to a safe position, execute T0100 U0 W0 on the input operation mode and program status page, and cancel the tool offset;

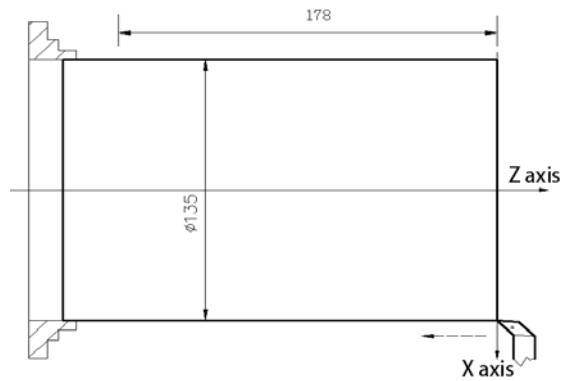
2. Move the tool and make the tool cut along the end face of the workpiece;



3. Release the tool along the X axis without moving in the Z axis, stop the spindle rotation, execute G50 Z0 on the input operation mode and program status page, and set the coordinates of the Z axis;

4. Switch to the knife length page and enter Z0 at offset at No. 001;

5. Move the tool and cut the tool along the outer circle of the workpiece;



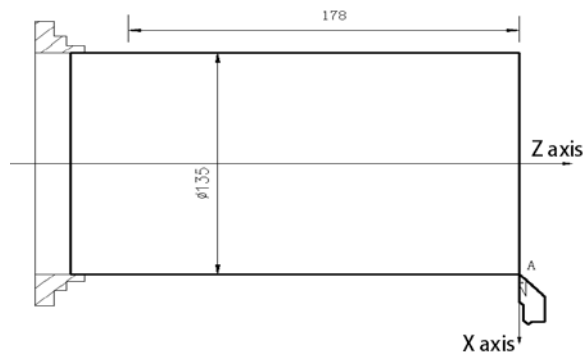
6. Release the tool along the Z axis without moving in the X axis, stop the spindle rotation, and measure the outer circle size of the workpiece (if the measured value is 135mm);

7. Run G50 X135 on the input operation mode and program status page, and set the coordinates of the X axis;

8. Switch to the knife length page and enter X135 at No. 001;

9. Move the tool to a safe position, and press the tool change key in manual operation mode to execute the second knife;

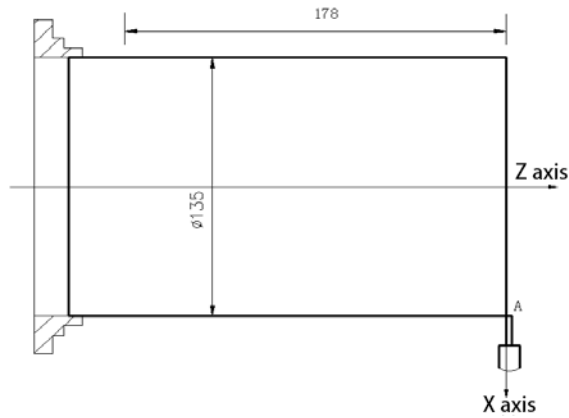
10. Start the spindle and move the tool to the tool point, as shown in point A of the figure below;



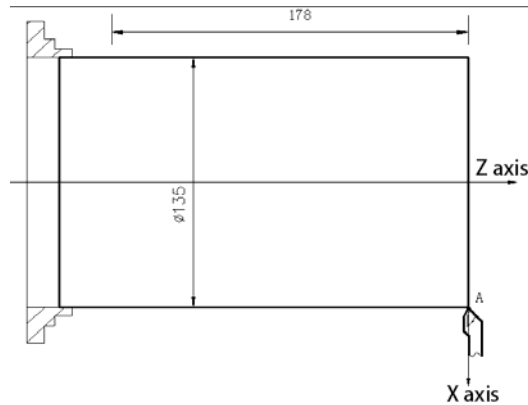
11. Switch to the knife length page, move the cursor to the 002 bias, and enter X135, Z0;

12. Move the tool to a safe position, and press the tool change key in manual operation mode to execute the No. 3 knife;

13. Start the spindle and move the tool to the tool point, as shown in point A as shown in the figure below;



14. Switch to the knife length page, move the cursor to the offset of 003, and enter X135, Z0;
15. Move the tool to a safe position, and press the tool change key in manual operation mode to execute the fourth knife;
16. Move the tool to the tool point, as shown in the figure A;



17. Switch to the tool offset page, move the cursor to the 004 bias, and enter X135, Z0;
18. After the tool is set, remove the tool to a safe position;



19. In the automatic operation mode, press Automatic to start processing;

20. Measure the size of the workpiece, if there is a deviation from the actual part size, the knife wear value can be modified until the part size is within the tolerance range.



Note: If you need to pause in the middle, press to pause the automatic run. If there is an



emergency in the middle, you can press the button, emergency stop button, and cut off the power to terminate.

Wear compensation interface enters:

- 1、 Press the panel  key to enter the knife length offset compensation page;
- 2、 Press the F2 key "Wear" under the screen to enter the tool wear input interface;

Note: The button  can switch the absolute value of wear or incremental input mode;

**The tip compensation interface enters:**

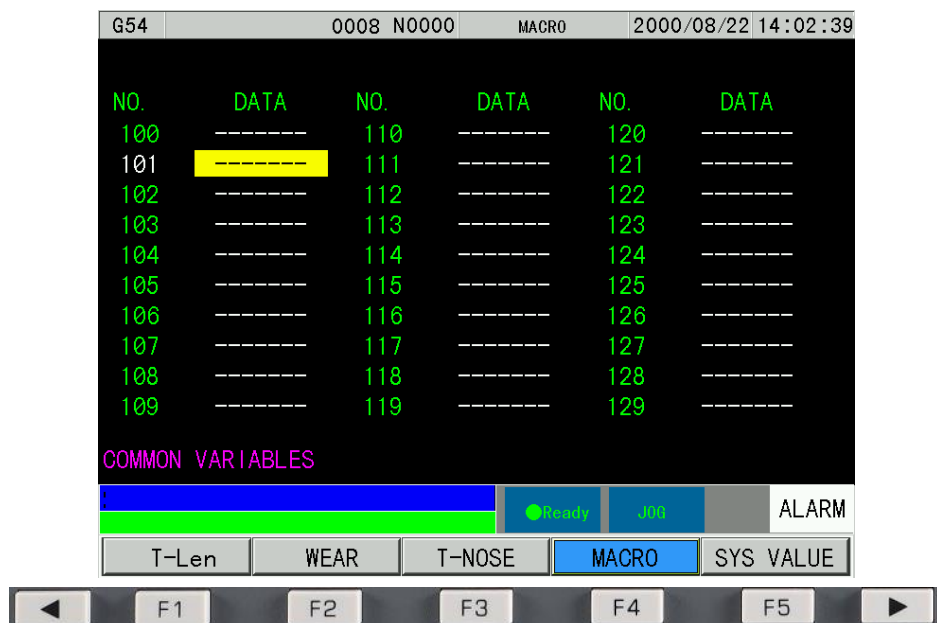
- 1、 Press the panel  key to enter the compensation page;
- 2、 Press the F3 key "Knife Tip" under the screen to enter the Knife Tip Wear Input Interface;

Note: The button  can switch the absolute value or incremental input mode of the tool tip compensation;



The macro variables interface goes to:

- 1、 Press the panel  key to enter the compensation page;
- 2、 Press the F4 key "Macro Variable" under the screen to enter the macro variable input interface;



The system variables interface goes to:

- 1、 Press the panel  key to enter the compensation page;

- 2、 Press the F5 key "System Variables" under the screen to enter the system variable input interface;



Second edition finalized on 22 July 2022