

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

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

preface

Dear Customer:

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

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

Security Warnings



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

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

Precautions

■ 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 move the product;
4. Collision and scratching of the panel and display screen are strictly prohibited;
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 the components are complete and whether there is damage to the list;
4. If there is a product model discrepancy, 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 abilities;
2. The product must be reliably grounded, the grounding resistance should be less than 0.1Ω , and the neutral wire (zero line) cannot be used instead of the ground wire;
3. The wiring must be correct and firm, so as not to lead to product failure or unintended consequences;
4. The surge absorption diode connected to the product must be connected in the prescribed direction, otherwise the product will be

damaged;

5. Before plugging in or unplugging the plug or opening the product chassis, you must cut off the power to the product.

■ Overhaul

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

2. When short circuit or overload, the fault should be checked, and the restart can be restarted after troubleshooting;

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

statement

The manual describes as many different elements as possible, but because of the possibilities involved to describe all the operations that can or cannot be performed, the contents of this manual that are not specifically described may be considered unusable.

warn

Before installing, connecting, programming and operating this product, it is necessary to read the product manual and the instruction manual of the machine tool manufacturer in detail, and strictly follow the requirements of the manual and the manual for related operations, otherwise it may lead to damage to the product, the machine tool, the 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 of this product is installed, the actual functional configuration and technical performance are determined by the design of the machine tool manufacturer, and the functional configuration and technical indicators of the CNC machine tool are subject to the instruction manual of the machine tool manufacturer;

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

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

Second operation instructions

Describes how to operate and use lathe systems.

Safety Responsibility

Manufacturer's responsibility for safety

- The manufacturer shall be responsible for the design and structural elimination and/or control of the CNC system provided and the accessories supplied with the accompanying supply.
- The manufacturer shall be responsible for the safety of the CNC system provided and the accessories supplied with the accompanying supply.
- The manufacturer shall be responsible for the information and suggestions for the use of the user.

User's Security Responsibility

- Users should learn and train the safe operation of the numerical control system, and be familiar with and master the content of safe operation.
- The user shall be responsible for the safety and danger caused by adding, changing or modifying the original NUMERICAL control system and accessories.
- The user shall be responsible for the hazards caused by the failure to operate, adjust, maintain, install and store the product in accordance with the provisions of the manual.

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

directory

Chapter 1 Operation Mode and Display Interface	12
1.1 Panel division	12
1.1.1 Status Indication.....	13
1.1.2 Edit keyboard	13
1.1.3 Display menu	15
1.1.4 Machine panels	16
1.2 Overview of how to operate	19
1.3 Display the interface	20
1.3.1 Location Interface.....	22
1.3.2 Program Interface.....	26
1.3.3 Tool offset and wear, macro variable interface.....	29
1.3.4 Alarm interface.....	31
1.3.5 Setting up the interface.....	32
1.3.6 Status parameters, data parameters, spiral complement parameter boundaries.....	36
1.3.7 CNC diagnostics, PLC signals, machine tool soft panels, help information, version information interface	38
1.3.8 Graphical Interface	40
Chapter II: Booting, Shutting Down, and Security Protection.....	42
2.1 Power on the system.....	42
2.2 Shutdown	42
2.3 Over-range protection.....	43
2.3.1 Hardware overrun protection	43
2.3.2 Over-range protection of soft parts	43
2.4 Emergency Operations.....	44
2.4.1 Reset.....	45
2.4.2 Emergency Stops	45
2.4.3 Feed Hold	45
2.4.4 Cut off the power supply.....	46
Chapter 3 Manual Operation	47
3.1 Axis movement.....	47
3.1.1 Manual feeds.....	47
3.1.2 Manual Quick Moves.....	48
3.1.3 Speed trimming.....	48
3.2 Other manual operations.....	50
3.2.1 Counterclockwise rotation, clockwise rotation, stop control.....	50
3.2.2 Spindle jog.....	50
3.2.3 Coolant control.....	50
3.2.4 Lubrication control	51
3.2.5 Manual tool change	52
3.2.6 Trimming of spindle magnification.....	52
Chapter 4 HandWheel / Single-Step Operation	54
4.1 Single-step feeding	54

4.1.1 Selection of increments.....	54
4.1.2 Movement direction selection	55
4.2 Handwheel (Hand Crank Pulse Generator) Feed.....	55
4.2.1 Selection of increments.....	56
4.2.2 Selection of moving axes and directions	57
4.2.3 Other Operations	57
4.2.4 Notes	58
Chapter 5 Input Operations	60
5.1 Input of code words	60
5.2 Execution of Code Words	61
5.3 Setting of Parameters.....	62
5.4 Modification of Data	62
5.5 Other Operations	62
Chapter VI: Program Editing and Management	63
6.1 Establishment of the Program.....	63
6.1.1 Generation of program segment numbers	63
6.1.2 Input of Program Content	64
6.1.3 Movement of the Cursor	65
6.1.4 Search for word and line numbers	66
6.1.5 Insertion of words	67
6.1.6 Deletion of words.....	68
6.1.7 Word Modifications.....	68
6.1.8 Deletion of Single Program Segments.....	68
6.1.9 Deletion of Multiple Program Segments.....	68
6.1.10 Block Deletion	69
6.1.11 Copying of Single Program Segments.....	69
6.1.12 Replication of Multiple Program Segments.....	69
6.1.13 Copying of Program Blocks.....	69
6.1.14 Pasting of Program Blocks	70
6.2 Removal of Programs	70
6.2.1 Deletion of Individual Programs.....	70
6.2.2 Removal of all programs.....	71
6.3 Selection of Programs	71
6.3.1 Search Methods	71
6.3.2 Scanning Method	71
6.3.3 Cursor Confirmation Method	72
6.4 Renaming of the Program	73
6.5 Copying of Programs	73
6.6 Program Management	73
6.6.1 Program Directory	73
6.6.2 Number of Stored Programs	74
6.6.3 Storage Capacity.....	74
Chapter 7 Tool Offset and Alignment.....	75
7.1 Fixed-point knife pairing.....	75

7.2 Try cutting the knife	76
7.3 Return to the machine tool zero point to the knife	78
7.4 Setting and modifying tool offset values.....	81
7.4.1 Setting of tool offset values.....	82
7.4.2 Modification of tool offset values	82
7.4.3 Tool offset value cleared to zero	83
7.4.4 Tool wear value setting and modification	83
7.4.5 No. 0 knife deflects the translation workpiece coordinate system	84
Chapter 8 Automatic Operation.....	85
8.1 Autorun	85
8.1.1 Selection of Running Programs	85
8.1.2 Startup of Autorun	86
8.1.3 Stop of automatic operation	86
8.1.4 Automatically run from any segment.....	87
8.1.5 Feed, fast speed adjustment	88
8.1.6 Spindle speed adjustment.....	89
8.2 Status of the Runtime	90
8.2.1 Single-Stage Operation.....	90
8.2.2 Empty Run	90
8.2.3 The machine is locked for operation	90
8.2.4 Accessibility locks up operation	91
8.2.5 Segment selection	91
8.3 Other Operations	91
Chapter 9: Back to Zero Operation.....	93
9.1 Program zeroback.....	93
9.1.1 Program Zero.....	93
9.1.2 Procedure for program zeroing	93
9.2 Machine return to zero	94
9.2.1 Machine zero	94
9.2.2 Operation steps for machine tool zeroing.....	94
9.3 Other operations in zeroback mode	96
Chapter 10 Data Setup, Backup, and Recovery	96
10.1 Settings for Data	96
10.1.1 Switch Settings	96
10.1.2 Graphics Settings.....	98
10.1.3 Setting of Parameters.....	98
10.2 Data Restore and Backup	104
10.3 Setting and Modifying Permissions	105
10.3.1 Entry of the Operation Level	107
10.3.2 Change of Operation Password	108
10.3.3 Operation Level Degradation	109
Chapter 11 USB flash drive operation function.....	111
11.1 File Directory Page	111
11.2 File Copying	112

Chapter 12 Processing Examples.....	112
12.1 Programming.....	113
12.2 Input to the Program.....	114
12.2.1 Viewing Existing Programs	114
12.2.2 Establish new procedures	115
12.3 Program Validation.....	116
12.3.1 Graphics Settings.....	116
12.3.2 Validation of Programs	117
12.4 Pairing knives and running	117

Chapter 1 Operation Mode and Display Interface

1.1 Panel division

Lathe system adopts a variety of operating panels, the composition of the panel is basically the same, can be roughly divided into 4 parts;

Part 1: Display menu, function menu keys (position, program, compensation, etc.), such keys are handled by the system software, without PLC control, for switching between different interfaces;

Part 2: MDI panel (editing keyboard), containing programming code word keys (XYZMST and other letter keys), numeric keys (0-9 keys), symbol keys (+-etc. keys), such keys are handled by the system software, not controlled by the PLC, for editing programs and modifying parameters;

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

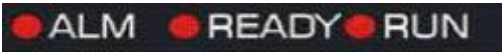
Part 4: Indicator light, containing the zero-back light for each axis and the machine status display light (tri-color lamp).



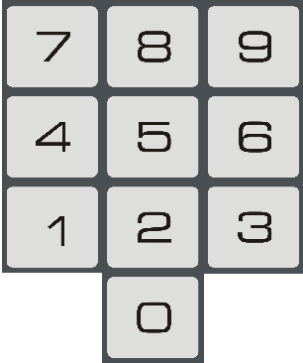
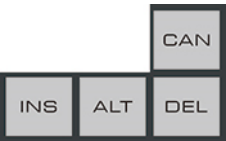
Panel division

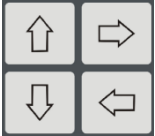
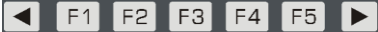
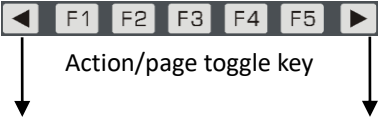
Note: Some system models have different button arrangement positions.

1.1.1 Status Indication

	Axis back-to-zero end LED
	Tri-color lights

1.1.2 Edit keyboard

keystroke	name	Function description
	Reset key	CNC reset, feed, output stop, etc
	Address key	Address entry Dual address keys, repeated keystrokes, switching between the two
	Symbol key	Three address keys, repeated keys, switch between the three
	Number keys	Numeric input
	Enter the key	Determination of data inputs such as parameters and compensation amounts
	Output key	Start the communication output
	Conversion key	Toggle of information and display
	Edit key	Insertion, modification, deletion of programs, fields, etc. during editing

	EBO key	The input of the end of the segment
	Cursor move key	Controls cursor movement
	Page turn key	Switch between pages under the same display interface
	Soft function keys	After using the function key to switch the page set, you can use the corresponding software function key to display the content of a subpage in the current page, as follows:  Go back to the parent menu and move the menu key to the right The role of soft function keys: (1) Switch subpages within the current page set; (2) As the operation input of the currently displayed subpage, such as editing, modifying data or displaying content.
The following buttons are somewhat modeled		
	Enter	Determination of data inputs such as parameters and compensation amounts; Input of the end of the segment; NOTE: Models with enter keys do not have input keys, and enter keys are used as input and line feed keys
	The first row last row key	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 up key

1.1.3 Display menu

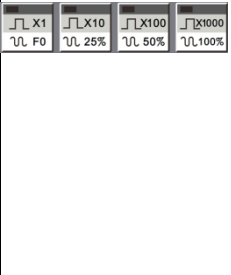
Menu key	remark
POS	Go to the location interface. The position interface has four surfaces, such as relative coordinates, absolute coordinates, comprehensive coordinates, and cheng supervisors
PRG	Go to the program interface. The program interface has four pages: program content, program status, program directory, and file directory
OFT	Enter the compensation boundary plane, macro variable boundary plane, and repeatedly press the key to switch between the three interfaces. Compensation interface shows tool bias wear; The Macro Variables interface displays CNC macro
ALM	Enter the alarm interface, alarm log, and repeated keystrokes can be switched between the two interfaces. The alarm interface has two pages: CNC alarm and PLC alarm; The alarm log shows the history of generating and dismissing alarms
SET	Enter the setting interface, graphical interface, and repeated keystrokes can be converted between the two interfaces. The setting interface includes switch setting, G54-G59, data manipulation, permission setting, and time setting; The graphical interface shows the trajectory of the feed axis
PAR	Enter the status parameters, data parameters, and screw parameters interface. Repeated keystrokes can be switched between interfaces
DGN	Enter the CNC diagnostic interface, PLC status, PLC data, machine tool soft panel, version information interface. Repeated keystrokes can be switched between interfaces. CNC diagnostic interface, PLC status, PLC data show CNC internal signal status PLC everywhere Address, status information of data; The soft panel of the machine tool can be operated by the soft keyboard of the machine tool; The version information
PLC	Enter the ladder boundary plane, PLC information, PLC ladder diagram, PLC parameter number, PLC diagnostic boundary plane, Repeated keystrokes can be switched between faces.
GRA	Enter the graphics dimension.

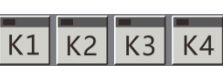
1.1.4 Machine panels

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

Lathe series standard PLC program defined in the machine panel each button function is shown in the table below:

keystroke	name	Function description	How to operate when the
	Feed hold key	Programs, MDI code runs paused	Automatic mode, input mode, DNC mode
	Loop start key	The program, MDI code runs and starts	Automatic mode, input mode, DNC mode
	Feed multiplier key	Adjustment of the feed speed	Automatic mode, input mode, editing mode, machine tool zeroing, handwheel mode, single step mode, manual mode, DNC mode
	Quick multiplier key	Adjustment of fast movement speed	Automatic mode, input mode Machine tool back to zero, manual mode, DNC mode
	Spindle magnification key	Spindle speed adjustment (spindle speed simulation control mode is effective)	Automatic mode, input mode, editing mode, machine tool zeroing, handwheel mode, single step mode, manual mode, DNC mode
	Manual tool change key	Manual tool change	Handwheel mode, single-step mode, manual mode
	Click to open the key	Spindle jog state on/off	Machine tool return to zero, handwheel mode, single-step mode, manual mode
	Lubrication opens the key	Machine lubrication on/off	Automatic mode, input mode, editing mode, machine tool zeroing, handwheel mode, single step mode, manual
	Coolant on key	Coolant on/off	Automatic mode, input mode, editing mode, machine tool zeroing, handwheel mode, single step mode, manual

	Spindle control keys	The spindle is rotating Spindle stops Spindle inversion	Automatic mode, input mode, editing mode, machine tool zeroing, handwheel mode, single step mode, manual mode, DNC mode
	Quick on and off	Fast speed/feed speed switching	Automatic mode, input mode, manual mode, DNC mode
    	X-axis feed key	Manual, one-step operation mode Of which each axis moves in the positive/negative direction	Machine return to zero, 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 multiplier selection key	Handwheel moves 1/10/100/1000 * MIN-equivalent step-by-step movement 1/10/100/1000 *MINE fast magnification F0, 25%, F50%, F100%	Automatic mode, input mode, machine tool return to zero, handwheel mode, single step mode, manual mode, DNC mode
	Select Stop	When Stop is selected active, perform the M01 pause	Automatic mode, input mode, DNC mode
	Single-stage switch	The program single-segment operation / continuous operation status switching, single-segment operation indicator light when the single segment is valid	Automatic mode, input mode, DNC mode
	Segment hop switch	Whether the program segment marked with a "/" sign at the beginning of the program segment skips the status switch, when the program segment selection switch is turned on, the jumping segment indicator is on	Automatic mode, input mode, DNC mode

	The machine locks the switch	When the machine tool is locked, the machine lock indicator light is on, and the output of the feed shaft is invalid	Automatic mode, input mode, edit mode, Machine tool zero, handwheel mode, single step mode, manual mode, DNC mode
	The accessibility function locks the switch	When the auxiliary function is locked, the auxiliary function lock indicator is on, and the M, S, T function output is invalid	Automatic mode, input mode, DNC mode
	C/S button	Spindle speed/position mode switching	Automatic mode, input mode, edit mode, Machine tool zero, handwheel mode, single step mode, manual mode, DNC mode
	Work light switch	Turn the work lights on or off	Automatic mode, input mode, edit mode, Machine tool zero, handwheel mode, single step mode, manual mode, DNC mode
	Chip conveyor switch	Turn the chip conveyor on or off	Automatic mode, input mode, edit mode, Machine tool zero, handwheel mode, single step mode, manual mode, DNC mode
	Custom output switches	Turns custom output on or off	Automatic mode, input mode, edit mode, Machine tool zero, handwheel mode, single step mode, manual mode, DNC mode
	Handwheel simulation development	Turns the handwheel simulation function on or off	Automatic mode, input mode, DNC mode
	Edit by selecting the key	Enter Edit Mode	Automatic mode, input mode, machine tool zeroing, Handwheel mode, single-step mode, manual mode, DNC mode
	Automatically select keys	Enter the automatic operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel mode, single-step mode, manual mode, DNC mode
	Input mode selection key	Enter the Input (MDI) operation mode	Automatic mode, input mode, machine tool zeroing, Handwheel mode, single-step mode, manual mode, DNC mode

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

1.2 Overview of how to operate

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 mode, you can create, delete and modify the machining program.

- Automatic operation mode

In automatic mode, run the program automatically.

- Input operation mode

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

- Machine zero-back operation mode

In the machine machine zero operation mode, the feed shaft back to the machine zero

operation can be performed separately.

- Handwheel / single-step operation

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

- Manual operation mode

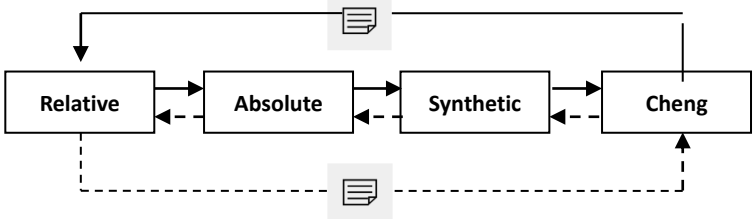
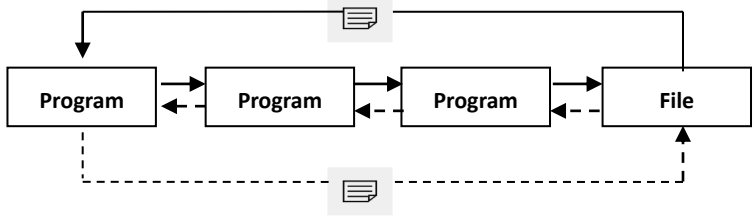
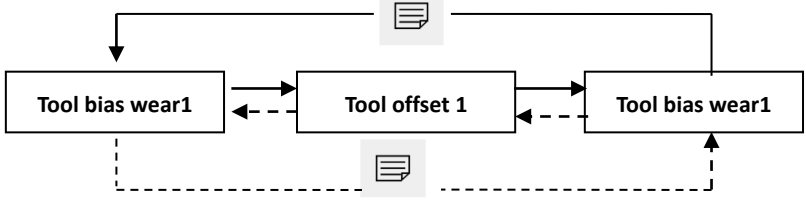
Under the manual operation mode, manual feed, manual fast, feed rate adjustment, fast magnification adjustment and spindle start and stop, coolant switch, lubrication switch, spindle jog, manual tool change and other operations can be carried out.

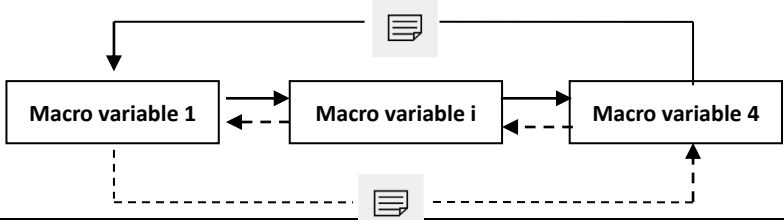
- The program zero-back operation mode

Under the program zeroback operation mode, the feed axis return program zero operation can be performed separately.

1.3 Display the 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) is independent of the operation mode. Show menus, display interfaces, and page hierarchies as shown in the following figure:

Menu	Display the	Displays the page
	Location interface	
	Program interface	
	Compensati on interface	

	Macro variable interface	
ALM	Alarm	Alarm
	Alarm log	Alarm logs
SET	Set up the interface	Each page is: Switch Settings, G54-G59, Data Operation, Permission Settings, Time Settings
	password	Operation level interface
PAR	Bit parameter	Bit parameter interface
	Number parameters	Number parameter interface
	Screw patch	Screw interface
DGN	CNC Diagnostics	CNC diagnostic interface
	PLC diagnosis	PLC signal diagnostic interface
	IO diagnostics	Input/output diagnostic interface
	Version information	The system software version is displayed
	alarm	Switch to the alarm interface
	Spindle detection	Detection spindle reaction time interface
	absolute value	Absolute value driven 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 instructions

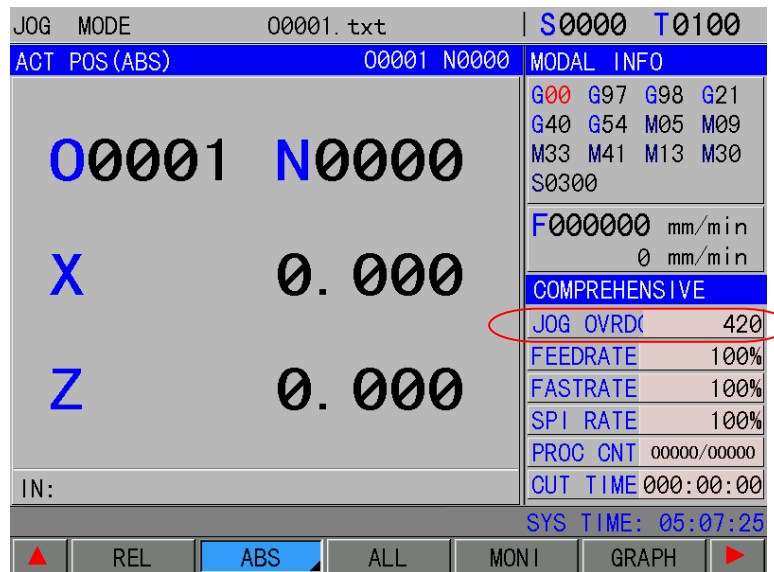
PLC	PLC information	PLC information
	Ladder diagram	Ladder diagram
	PLC parameters	PLC parameters: KEEP, TMR, DATA, CTR
	PLC signal	
	Tool magazine control	
	Extended control	
GRA	Graphical interface	

1.3.1 Location Interface

Press the  key to enter the position interface, the position interface has absolute coordinates, relative coordinates, comprehensive coordinates and cheng jian four pages, can be viewed through the  key or soft function key [F1-F5].

1) Absolute coordinates display the page

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



Note: In Editing, Auto, Entering, Displaying "Programming Speed"; Display "Manual Speed" in machine machine zero, program zero, manual mode. Display "Hand wheel increment" in handwheel mode; "Step Increments" is displayed in step-by-step mode.

Actual speed: the actual processing speed after the feed multiplier operation in actual processing;

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

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

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

Cutting time: when the automatic operation starts after the start of the timekeeping, the time unit is hours, minutes, seconds;

Fast Magnification: Displays the current fast magnification;

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

S0000: Spindle speed fed back by spindle encoder, spindle encoder must be installed to display the actual spindle speed;

T0100: Current tool number and tool offset number.

The number of processed parts and the cutting time are powered down memory, and the method of clearing zero is as follows:

Zeroing of the number of processed parts: first press and hold the  key, and then press the  key; Or go to the expansion menu in the location interface and press the corresponding function key to clear zero.

Cutting time zero: first press and hold the  key, then press the  key; Or go to the expansion menu in the location interface and press the corresponding function key to clear zero.

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. The 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 the CNC parameter No.005 of Bit1=1, with G50 set the absolute coordinate, U, W Same as the absolute coordinate value set.

Method for zeroing out U and W coordinates:

Press and hold the  key under the relative coordinate display page until the U flashes in the page, press the  key, and the U coordinate value is cleared to zero; Or go to the expansion menu in the location interface and press the corresponding function key to clear zero.

Press and hold the  key under the relative coordinate display page until the page flashes W, press the  key, and the W coordinate value is zeroed; Or go to the expansion menu in the location interface and press the corresponding function key to clear zero.

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

3) The combined coordinates display page

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

The displayed value of the machine coordinates is the coordinate value of the current position in the machine coordinate system, which is established by returning to the machine zero. The remaining amount of movement is the difference between the target position of the program segment or MDI code and the current position.

The display page is as follows:

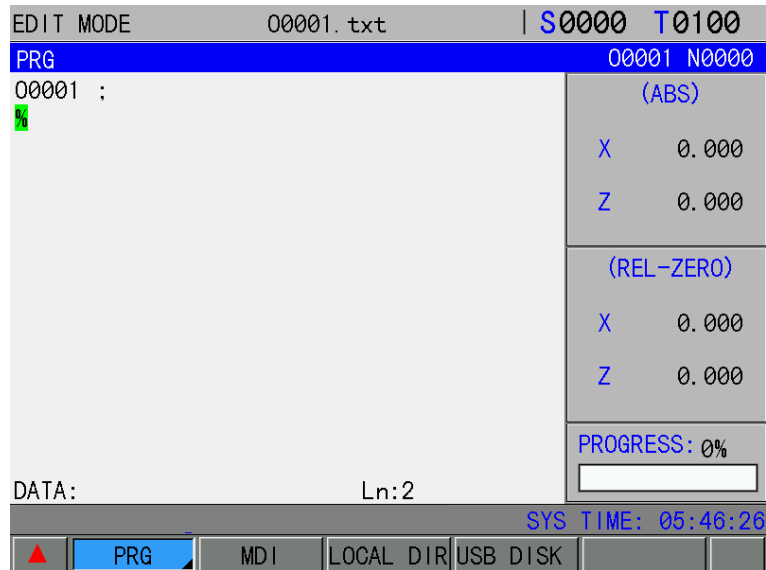
4) Cheng Jian displays the page

MDI		MODE		00001.txt		S0000		T0100	
MONITOR				00001 N0000		MODAL INFO			
(REL)		(ABS)				G00 G97 G98 G21 G40 G54 M05 M09 M33 M41 M13 M30 S0300			
U	0.000	X	0.000			F000000 mm/min 0 mm/min			
W	0.000	Z	0.000						
PRG						COMPREHENSIVE			
00001 :						PRG RATE 0			
%						FEEDRATE 100%			
						FASTRATE 100%			
						SPL RATE 100%			
						PROC CNT 00000/00000			
MDI:						CUT TIME 000:00:00			
Ln:1						SYS TIME: 05:15:44			
▲	REL	ABS	ALL	MONI	GRAPH	▶			

CHG

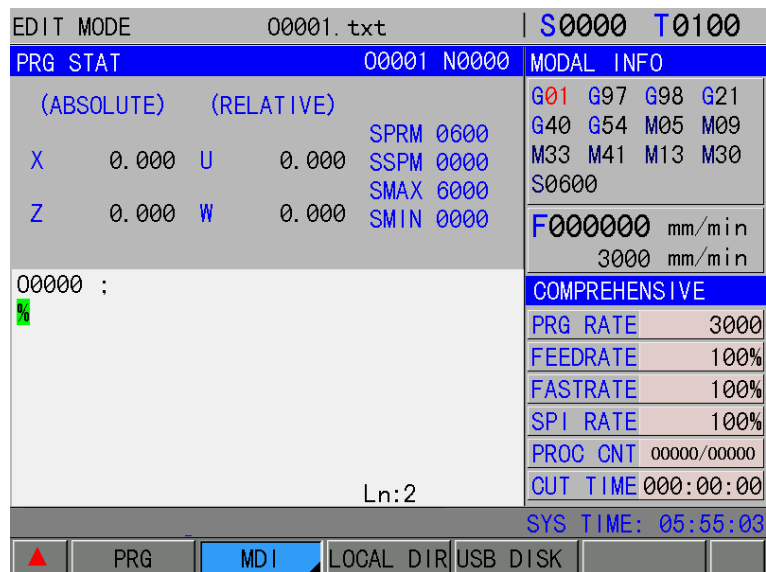
Press the key to enter the program interface, the program interface has four pages of program content, program status, program directory, file directory, by repeatedly pressing the key in each page to switch, you can also switch through the program interface function key.

On the Program Content page, program content, including the current program segment, is displayed. Press the key under Edit Mode, key forward, and backward to view the program content.



2) Program Status Page (MDI).

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



3) Program Directory page

When on the program status page, pressing the  key will take you to the program directory page. Under the interface, all the processing programs are listed, and in order to facilitate the user 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 is displayed on the program directory page:

- Number of saved programs:

Displays the number of programs (including subroutines) that have been saved in the CNC

b) Number of programs remaining:

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

c) Stored amount:

Displays the amount of storage (KB) occupied by CNC-saved part programs

d) Remaining storage:

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

e) Program Directory:

Displays the program number stored in the part program sequentially by the size of the part program name

f) Program size:

Displays the size of the storage space occupied by the CNC program

EDIT MODE		01122. txt		S0000 T0100	
PROGRAM (DIR)				01122 N0000	
PRG CNT4		MEM	103 M	USED:2 M	FREE:101 M
NAME		SIZE	DATE	PROGRAM PREVIEW:	
00001	9 (B)	00-06-17	00005;		
00005	377 (B)	00-06-17	SOM3;		
01122	3 (B)	00-06-18	G1X0Y0Z0F3000;		
07000	1.20 (K)	00-05-10	M3S600;		
			M8;		
			G1X300Y300Z300;		
			M3S1200;		
			G1X800Y800Z800;		
			M3S2400;		
			G1X1000Y1000Z1000;		
			M3S3600;		
TIPS: [EOB]Open[O]New[P]Search[N]Remane[DEL]DeI					
SYS TIME: 11:30:52					
▲	PRG	MDI	LOCAL DIP	USB DISK	

4) File Directory page

When on the Program Directory page, pressing the  key will take you to the File Directory page. The page appears as follows:



1.3.3 Tool offset and wear, macro variable interface, tool life management

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

When the status parameter No 002 has bit0 bit 1, press the  key again to enter the tool life management interface

1) Tool bias & wear interface

The Tool Bias & Wear interface has a total of 7 pages, a total of 33 biases, wear numbers (No.000 ~ No.032) for users to use, through  Keys and  keys display each page, and the display page is shown as shown in figure:

EDIT MODE		01122. txt		S0000 T0100	
OFFSET WEAR: 01122 N0000					
NO.	TYP	X	Z	R	T
00	OFT	0.000	0.000	0.000	0
	WEAR	-----	-----	-----	0
01	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
02	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
03	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
04	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0

Memory: X I Record Z J Record
 00OFT=

RELATIVE
 U 0.000
 W 0.000
 ABSOLUTE
 X 0.000
 Z 0.000

SYS TIME: 12:33:22
 [OFT] [T-Len] [WEAR] [T-NOSE] [MACRO]

2) Macro variable interface

The macro variable interface has 20 pages, which 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 variables, macro variable values can be specified by macro code or set directly by the keyboard.

EDIT MODE		01122. txt		S0000 T0100	
MACRO 01122 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
100		110		120	
101		111		121	
102		112		122	
103		113		123	
104		114		124	
105		115		125	
106		116		126	
107		117		127	
108		118		128	
109		119		129	

common variable

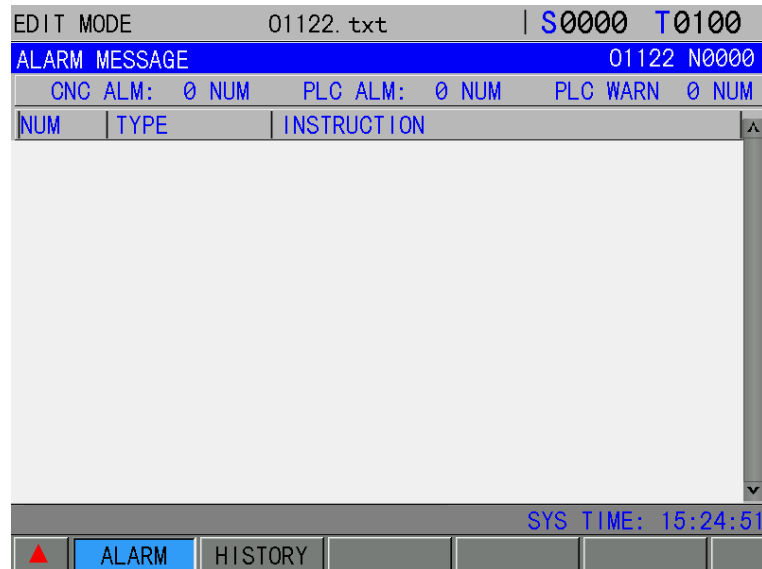
NO. 100 =

SYS TIME: 12:40:03
 [OFT] [T-Len] [WEAR] [T-NOSE] [MACRO]

1.3.4 Alarm interface

1) Alarm:

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



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

(Alarm #100 must be cleared by pressing both the  key and the  key).

2) Alarm Log:

Press the  key again to enter the alarm log interface. Keys  and keys  can view a total of 200 alarm log messages;

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

EDIT MODE		01122.txt	S0000 T0100
ALARM LOG		01122 N0000	
NUM	INSTRUCTION	ALARM TIME	
0001	Fail opening file	00-06-18 13:26	
0001	Fail opening file	00-06-18 12:58	
0001	Fail opening file	00-06-18 04:33	
0001	Fail opening file	00-06-17 23:17	
0456	Spindle driver alarm.	00-06-17 23:15	
0456	Spindle driver alarm.	00-06-17 23:15	
0456	Spindle driver alarm.	00-06-17 23:15	
0456	Spindle driver alarm.	00-06-17 23:15	
0251	Emergency stop alarm	00-06-17 23:15	
0456	Spindle driver alarm.	00-06-17 23:15	
0251	Emergency stop alarm	00-06-17 23:15	
0251	Emergency stop alarm	00-06-17 23:05	
SYS TIME: 15:29:36			
▲	ALARM	HISTORY	

Note: Manual removal of alarm logs: Press the  +  key or the corresponding function key under the level 2 password to clear all log information.

1.3.5 Setting up the interface

The key is a composite key, press the  key once from other pages to enter the setting interface, and repeatedly press  to switch the password, backup, switch, time, coordinate system interface.

1) Set up the interface

The setup interface has five pages, which can be viewed by  key or corresponding function key at the bottom of the screen.

Switch settings: Display the on and off status of parameters, programs, and automatic sequence numbers.

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

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

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

EDIT MODE		01122. txt		S0000 T0100	
ON-OFF SETTING		01122 N0000		MODAL INFO	
PAR SWITCH: OFF *ON PRG SWITCH: OFF *ON AUTO SEQ: *OFF ON		G00 G97 G98 G21 G40 G54 M05 M09 M33 M41 M13 M30 S0300		F000000 mm/min 0 mm/min	
				COMPREHENSIVE	
				PRG RATE 0	
				FEEDRATE 100%	
				FASTRATE 100%	
				SPI RATE 100%	
				PROC CNT 00000/00000	
				CUT TIME 000:00:00	
				SYS TIME: 15:44:10	
▲ PASSWORD BACKUP COORD TIME SWITCH					

Data operations: On this page, you can backup and restore CNC data (status parameters, data parameters, screw complement parameters, tool bias, etc.).

Initial Value Restore Operation: Parameters, tool complements, and screw complements can be restored to system defaults.

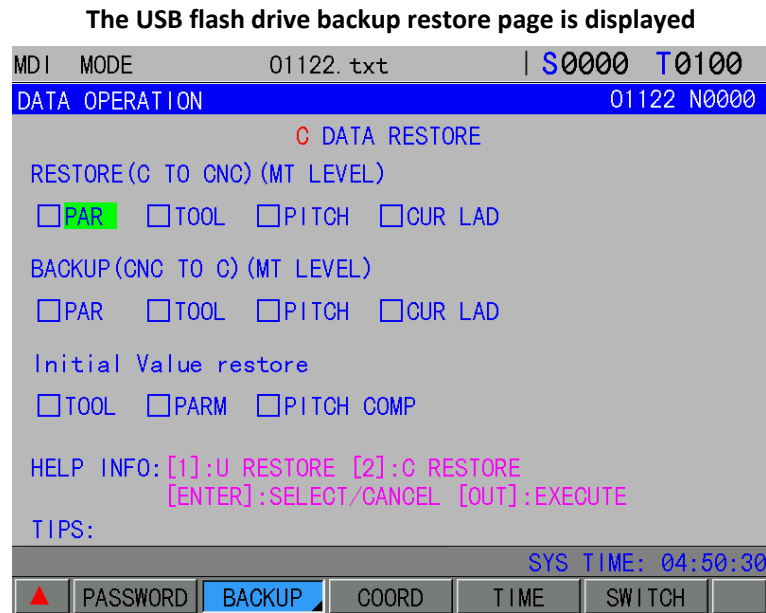
C drive data restore to CNC: Restore the data files backed up on the system disk to the system.

Restore the data of the USB drive to CNC: Return the data files backed up on the USB drive to the system.

BACKUP CNC DATA TO C DRIVE: Back up the system's current parameters, tool complement, screw patch and ladder to the system disk.

CNC data backup to USB flash drive: The system's current parameters, tool complement, screw repair and ladder diagram are backed up to the USB stick.

MDI MODE		01122. txt		S0000 T0100	
DATA OPERATION		01122 N0000			
		U DATA RESTORE			
		RESTORE(U TO CNC) (MT LEVEL)			
		☐ PAR ☐ TOOL ☐ PITCH ☐ CUR LAD ☐ ALL LAD			
		BACKUP(CNC TO U) (MT LEVEL)			
		☐ PAR ☐ TOOL ☐ PITCH ☐ CUR LAD ☐ ALL LAD			
		Initial Value restore			
		☐ TOOL ☐ PARM ☐ PITCH COMP			
		U disk path:\CNC BackUp		[U] Select folder	
		HELP INFO: [1]:U RESTORE [2]:C RESTORE			
		[ENTER]:SELECT/CANCEL [OUT]:EXECUTE			
		TIPS:			
				SYS TIME: 04:26:28	
▲ PASSWORD BACKUP COORD TIME SWITCH					



C drive backup restore displays the page

Permission settings: Display and set the user action level.

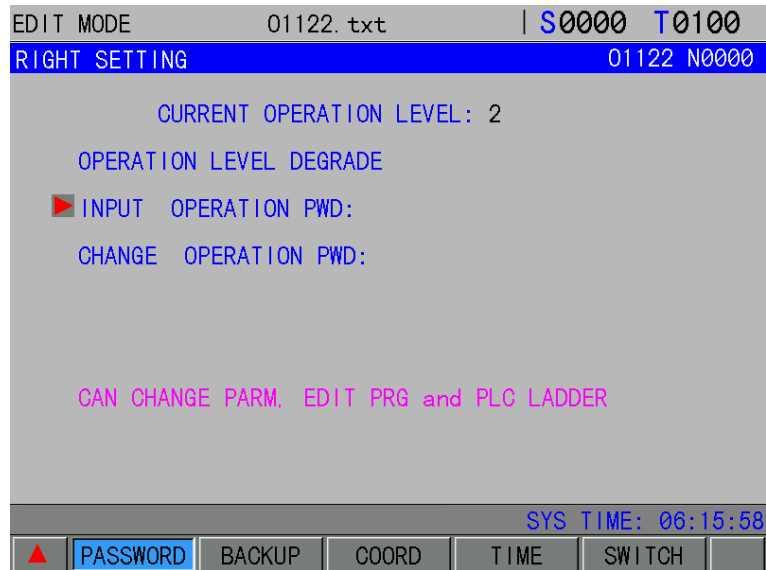
The lathe series system password level is divided into 4 levels, from high to low, the machine tool manufacturer level (level 2), the equipment management level (level 3), the craftsman level (level 4), and the machining operation level (level 5).

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

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

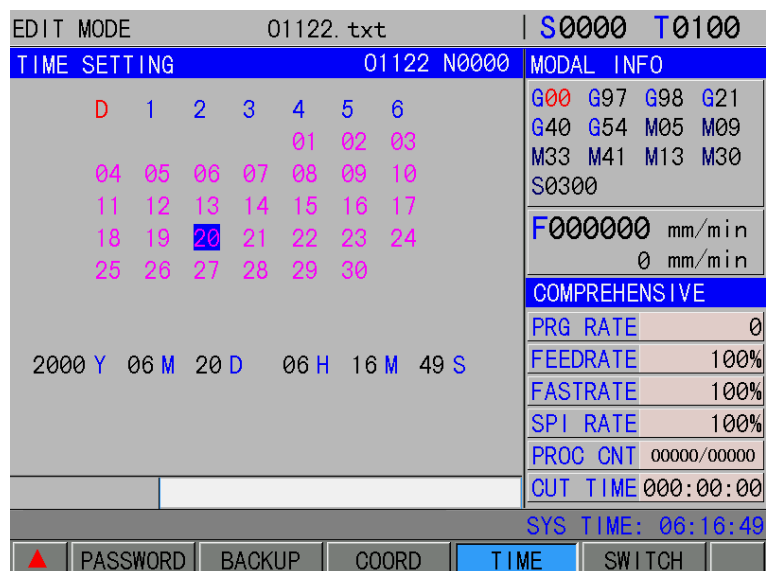
Technician level: initial password 1234, can modify the tool complement data (perform tool operation), macro variables, edit part program, can not modify the STATUS parameters, data parameters and screw complement parameters of CNC.

Machining operation level: no password level, can be operated by the machine operation panel, can not modify the tool compensation data, can not select parts program, can not edit the program, can not modify the STATUS parameters of CNC, data parameters and screw complement parameters.



Time Date: Displays the current time and date.

You can press the  key to enter the modification mode, press ,  key to switch the year / month / day / hour / minute / second of the desired modification, press , The  key modifies its value, and if you need to exit modification mode, press the  key again.



1.3.6 Status parameters, data parameters, spiral complement parameter

boundaries



The key is a multiplex key, and repeatedly pressing this key can enter several interface surfaces such as the total table of parameters, the processing technology, user parameters, status parameters, data parameters and screw complement parameters.

1) Status parameters interface



Press the key to enter the status parameter interface, a total of 48 status

parameters are displayed in two pages, you can enter each page through the key and



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

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

EDIT MODE

01122.txt

S0000 T0100

STATUS PARAMETER

01122 N0000

NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000011	007	10101111	012	00111001
003	00110111	008	00011110	013	11000000
004	01000000	009	00000000	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

NO. 001 =

SYS TIME: 06:17:25

▲

ALLPARM

PROC TE

USERPARM

BITPARM

NUMPARM

▶

2) Data parameters interface



Repeatedly press the key (such as the key on the status parameter page) to

enter the data parameter interface, you can enter each page through the key,

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

As you can see from the Data Parameters page, the middle of the page table shows the Chinese meaning of the parameters, and you can select different parameters by moving the cursor up and down.

EDIT MODE

01122. txt

| S0000 T0100

NUM PARAMETER

01122 N0000

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

MIN1MAX65536P:1/36

NO. 015 =

SYS TIME: 06:18:13

▲

ALLPARM

PROC TE

USERPARM

BITPARM

NUMPARM

▶

3) Screw complement parameter interface

Repeatedly press the  key to enter the pitch error compensation interface, the pitch error compensation is displayed in 256 sub-11 pages, and each page can be displayed by keys  and  :

EDIT MODE

01122. txt

| S0000 T0100

PITCH COMP

01122 N0000

NO.	X	Z	NO.	X	Z
000	0	0	011	0	0
001	0	0	012	0	0
002	0	0	013	0	0
003	0	0	014	0	0
004	0	0	015	0	0
005	0	0	016	0	0
006	0	0	017	0	0
007	0	0	018	0	0
008	0	0	019	0	0
009	0	0	020	0	0
010	0	0	021	0	0

NO. 000 =

SYS TIME: 06:18:51

◀

PITCH

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



The key is a composite key, and repeatedly pressing this key can enter the CNC diagnostic interface, PLC signal interface, machine tool soft panel, help information and version information interface.

1) CNC diagnostic interface

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

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

EDIT MODE		01122. txt		S0000 T0100	
CNC DGN				01122 N0000	
NO.	DATA	NO.	DATA	NO.	DATA
000	00000000	010	00000000	020	00000000
001	00000000	011	00000000	021	00000000
002	00000000	012	00000000	022	00000000
003	00000000	013	00000000	023	00000000
004	00011111	014	00100000	024	00000000
005	00000000	015	00000000	025	00000000
006	00000000	016	00000000	026	00000000
007	00000000	017	00000000	027	00000000
008	00000000	018	00000000	028	00000000
009	00000000	019	00000000	029	00000000
ESP *** ** DEC5 DEC4 DECZ DECY DECX					
BIT0:X axis machine zero return deceleration					
NO. 000					
SYS TIME: 06:43:59					
▲	CNC DGN	PLC SIG	IO DGN	VER INFO	ALM ▲

2) PLC signal interface

On the page of the PLC signal interface, a total of X0000 ~ X0063, Y0000 ~ Y0047, F0000 ~ are displayed in turn Address statuses such as F063, G0000 ~ G063, etc. Press the key

repeatedly to enter the PLC status interface. Press the  key and  key to view the signal status of each address of the PLC.

In the PLC status page, the lower part of the page has two detailed content display lines, the second line shows the Chinese meaning of a bit of the address number where the current cursor is located, you can press the left or right move key to change the displayed address bit; The first line displays the abbreviations of all the digits of the address number where the current cursor is located.

EDIT MODE

01122. txt

| S0000 T0100

PLC SIG

01122 N0000

NO.	DATA	NO.	DATA	NO.	DATA
X000	00000000	X005	00010000	X010	00000000
X001	00000000	X006	00000000	X011	00000000
X002	00000000	X007	00000000	X012	00000000
X003	00000000	X008	00000000	X013	00000000
X004	00000000	X009	00000000	X014	00000000

BIT0:X000.0 protective door detection signal

BIT1:X000.1 external Pause

BIT2:X000.2 chuck control input

BIT3:X000.3 X axis deceleration signal

BIT4:X000.4 tailstock control input

BIT5:X000.5 emergency stop signal

BIT6:X000.6 4th axis deceleration signal

BIT7:X000.7 Position T05

X000

SYS TIME: 06:44:31

CNC DGN

PLC SIG

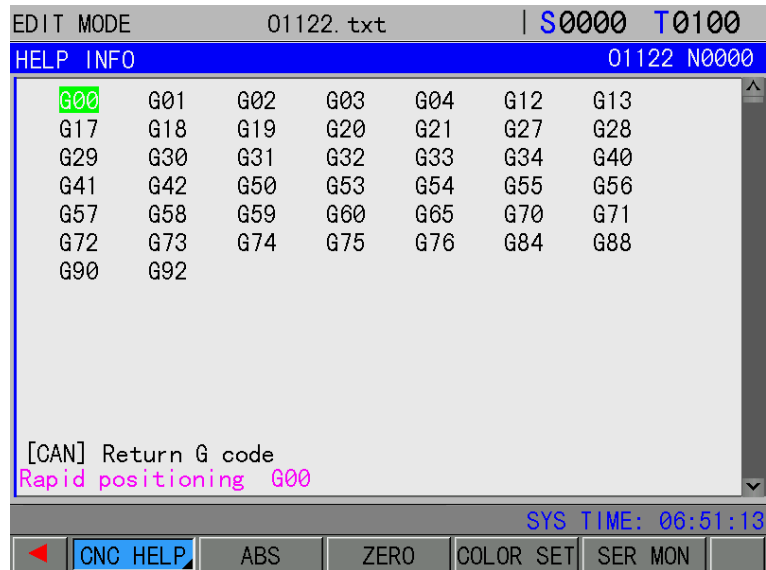
IO DGN

VER INFO

ALM

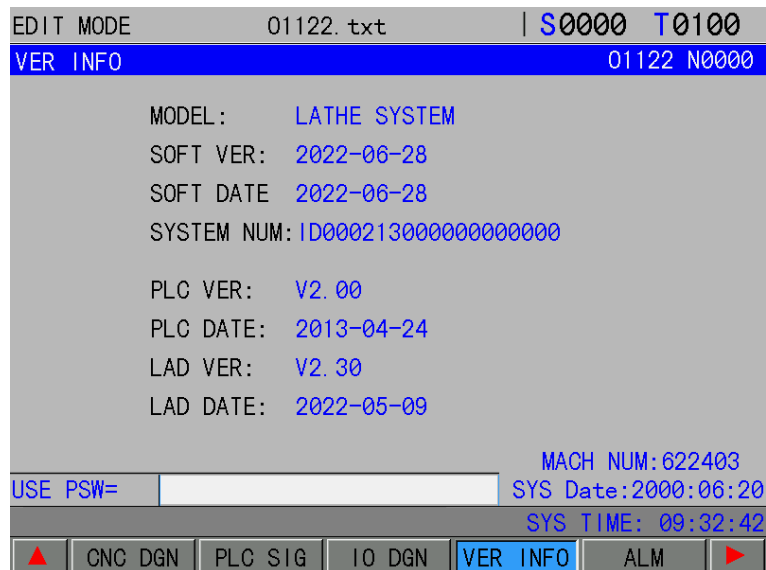
3) Help information

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



4) Version information

Press the  key repeatedly to enter the version information interface. The version information page displays the information of the current software, hardware, and system PLC version of the CNC, and the display page is shown in the figure.



1.3.8 Graphical Interface

In the graphical interface, you can zoom in, zoom out, clear and other operations of the graph.



Chapter II: Booting, Shutting Down, and Security Protection

2.1 Power on the system

Before powering on, it should be confirmed: 1, the machine state 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 Present Location (Relative Coordinates) page is displayed.

EDIT MODE		01122. txt		S0000 T0100		
ACT POS (REL)		01122 N0000		MODAL INFO		
O1122		N0000		G00 G97 G98 G21		
U		0. 000		G40 G54 M05 M09		
W		0. 000		M33 M41 M13 M30		
				S0300		
				F000000 mm/min		
				0 mm/min		
				COMPREHENSIVE		
				PRG RATE 0		
				FEEDRATE 100%		
				FASTRATE 100%		
				SPI RATE 100%		
				PROC CNT 00000/00000		
				CUT TIME 000:00:00		
IN:				SYS TIME: 09:35:07		
▲	REL	ABS	ALL	MONI	GRAPH	▶

2.2 Shutdown

Before shutting down, you should confirm:

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

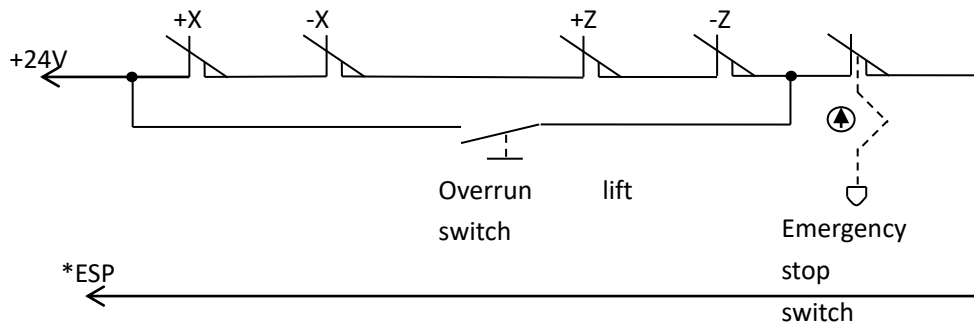
tool.

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 Over-range protection

2.3.1 Hardware overrun protection

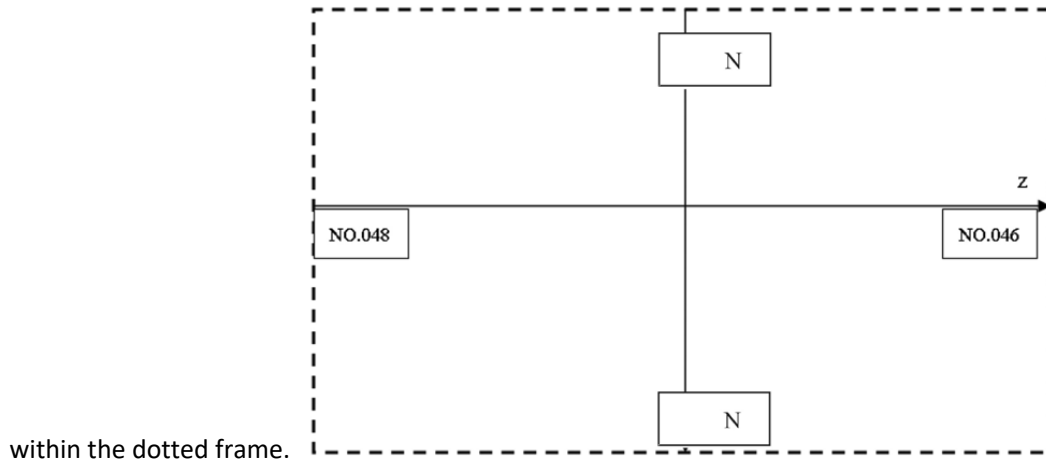
Install the stroke limit switch at the positive and negative maximum strokes of the X and Z axes of the machine tool respectively, and connect the wiring as shown in the following figure, at this time the bit2 (EALM) of the status parameter No.215 must be set to 0 。 When there is an overrun, the stroke limit switch action, stop the movement and display the emergency stop alarm; In addition, the external method X and Z stroke limit switches are connected to the X and Z corresponding hard limit overrun input points.



When a hardware overrun occurs, an "emergency stop" alarm appears. The method of eliminating the "emergency stop" alarm is: press the overrun release button without releasing, switch to the alarm information page, view the alarm information, reset the alarm after clearing the alarm, and move the workbench in the opposite direction (if the forward overrun, then move out in the negative direction; If negative overrun, move forward) off the stroke switch.

2.3.2 Over-range protection of soft parts

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



If the machine position (machine coordinates) exceeds the dotted area of the figure above, an overdraft alarm appears. The method of lifting the overrange alarm is: press the reset key, clear the alarm display, and move in the opposite direction (if the forward overrun, then move out in the negative direction; If the negative direction is overrun, it is moved out in the positive direction).

Additional axes

The Y-axis software stroke range 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 stroke range is set by the data parameters NO.193 and NO.196, with the machine coordinate values as the reference values. NO.193 is the 4th axis forward maximum stroke and NO.196 is the 4th axis negative maximum stroke.

The 5th axis software stroke range is set by the data parameters NO.194 and NO.197, with the machine coordinate values as the reference values. NO.194 is the 5th axis forward maximum stroke and NO.197 is the 5th axis negative maximum stroke.

2.4 Emergency Operations

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

2.4.1 Reset

When the system abnormal output and axis abnormal action, press the  key to reset the system:

1. There is axial motion stop;
2. M, S function output is invalid (can be set by the parameter to set the button  after whether to automatically close the spindle counterclockwise / clockwise rotation, lubrication, cooling and other signals, PLC ladder diagram definition);
3. Automatic operation ends, modal function, state retention.

2.4.2 Emergency Stops

During the operation of the machine tool, press the emergency stop button (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 closed. Release the E-STOP button to lift the E-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 lifting the emergency stop alarm;

NOTE 2: Pressing the E-STOP button before power-on and shutdown reduces the electrical shock of the device;

Note 3: After the emergency stop alarm is lifted, the operation of returning 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 operation of returning to the zero point of the machine tool shall not be carried out);

Note 4: An external emergency stop is only valid if **bit2 (EALM)** of the status parameter **NO.215** is set to 0.

2.4.3 Feed Hold

The operation can be paused by pressing a  key during the operation of the machine. It is important to note that this function does not stop the operation immediately during thread cutting, tapping, and looping code runs.

2.4.4 Cut off the power supply

During the operation of the machine, 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 supply, the CNC display coordinates may deviate greatly from the actual position, and operations such as re-alignment of the knife must be carried out.

Chapter 3 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 can be found in the manual of the machine tool manufacturer.

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

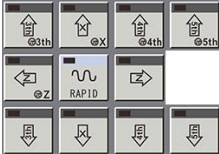
Press the  key to enter the manual operation mode, manual feeding, spindle control, magnification repair, tool change and other operations can be carried out under the manual operation mode.

3.1 Axis movement

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

3.1.1 Manual feeds

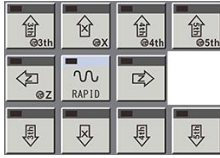
Press and hold  or  in the feed axis and direction selection key

. The X-axis arrow key can make the X-axis negative or positive feed, and the axis movement stops when the key is released; Press and hold  or , the Z-axis arrow key can make the Z-axis negative or positive feed, and the axis movement stops when the key is released; Press and hold  or , the Y-axis arrow key can make the Y-axis negative or positive feed, and the axis movement stops when the key is released; Press and hold  or , the 4th-axis arrow key can make the 4th-axis negative or positive feed, and the axis movement stops when the key is released. 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 illuminate and enter the manual fast movement state.

3.1.2 Manual Quick Moves

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 negatively or forward quickly, and the axis movement stops when the key is released; Pressing the  or  key can make the Z axis move quickly in the negative or positive direction, and the axis movement stops when the key is released; Press and hold the  or  Y axis arrow keys to make the Y axis negative or forward feed, and the axis movement stops when the key is released; Press and hold the  or  4th axis arrow keys to make the 4th axis negative or forward feed, and the axis movement stops when the key is released; 5th axis operation is consistent with the above axis operation; Fast multiplier real-time trimming is effective.

When performing a manual quick move, press the  key to make the indicator light go off, the fast move is ineffective, and feed at manual speed.

NOTE 1: After power is turned on, if the reference point is not returned, when the fast move switch is turned on (the fast move indicator is on), whether the fast move speed is manual feed speed or fast move speed is determined by bit0 bit (ISOT) of the system status parameter NO.012 Select;

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

3.1.3 Speed trimming

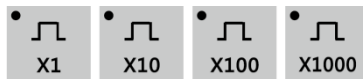
In manual feed, the manual feed rate can be modified by , a total of 16 levels. When the parameter NO.031 is set to 1260, the relationship between the feed rate and the feed rate is shown in the following table

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

Feed multiplier	Feed speed (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 percent .

In the case of manual fast movement, the multiplier of manual fast movement can be



modified by pressing . The multiplier has four gears: Fo, 25%, 50% and 100%. (Fo speed is set by data parameter No.032).

The fast magnification selection is valid in the following situations:

- 1) G00 moves fast
- 2) Fast movement in a fixed loop
- 3) Fast movement when G28
- 4) Manual quick movement

3.2 Other manual operations

3.2.1 Counterclockwise rotation, clockwise rotation, stop control



: Under the manual operation mode, press this key, the spindle rotates in reverse;



: Under manual operation mode, press this key, the spindle stops;



: Under manual operation, press this key to turn the spindle clockwise.

3.2.2 Spindle jog



: The spindle is in jog state.

Function description: Press the  key on the panel to enter the jog state, and the spindle jog function should be in the stopped state when the spindle is turned on and off. Spindle jogging state, pressing the  key, counterclockwise turning jog; Press the  key and turn the jog clockwise. The jog time and speed are set by the data parameters No.108 and No.109, respectively.

When the spindle jogs and rotates, press the  key to stop the spindle jog rotation, and the spindle braking signal is not output when the jog rotation stops.

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

Parameter settings:

PLC parameter K10.4 1/0: spindle jog is effective in any way / manual, handwheel mode.

Data parameter No.108: Spindle jogging time

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

3.2.3 Coolant control



: Press this key 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, for lubrication flip output, press the machine

operation panel  key, lubrication output, repeated press then lubrication output canceled. When M32 is executed, the lubrication output is performed, and then M33 is executed, and the lubrication output is canceled.

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

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

2. Automatic lubrication:

T13>0: Automatic lubrication, lubrication time T13 and lubrication interval T53 can be set

After power-up, lubrication begins at the time set by T13, then stops the output, and after the time set by T53, the output lubrication is repeated and cycled in turn. When lubricating

automatically, M32, M33 codes, and machine operation panel  keys are also valid.

Parameter settings:

PLC parameters: K10.1 1/0: Spindle lubrication cooling output maintained/closed when reset

PLC parameters: K16.2 1/0: automatic lubrication is effective when the boot is yes / no output lubrication

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

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

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

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

Additional Notes:

- 1) Automatic lubrication is effective when DT13 is not zero, at which point the output lubrication will be output after the time set by DT 53 (the time is set by DT 13); The lubrication output can also be turned on or off by hand and by command, and the opening time is set for DT1 3 (the timing will start again after this operation);
- 2) When the automatic lubrication is effective, if K16.2 is set to 1, the lubrication output will be output as soon as the system is powered on, and the output time is set by DT13, and if K16.2 is set to 0, the lubrication will be started after the power-on time set by DT53.
- 3) When DT13=0, manual lubrication is effective, if the parameter No.112=0, the output will not be automatically turned off, if not 0, the output time will automatically turn off after reaching the time set by NO.112;
- 4) When DT53=0, the lubrication output is turned off after entering DT13, and the output is no longer spaced; When the DT53 is not 0, the lubrication output is turned off after entering the DT13, and the output is in the output after the DT53 interval output time;
- 5) PLC parameters
 K16.1==0 Insufficient lubrication warning signal is valid with +24V (0: on 1: disconnected);
 K16.6==1 Insufficient lubrication warning prompt (0: invalid 1: effective);
- 6) Number parameters

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

Lubrication deficiency detection delay Unit: millisecond
--

3.2.5 Manual tool change

T. CHANGE: Under the manual/handwheel operation mode, press this key, manually change the tool in order (if it is currently the 1st tool, after pressing this key, the tool is changed to the 2nd; If the current tool is the last one, press this key and the tool is changed to the first one).

3.2.6 Trimming of spindle magnification

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



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

Chapter 4 HandWheel / Single-Step Operation

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

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

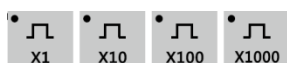
Please note that the functions related to the operator panel buttons below in this chapter are described for the system's standard PLC program.

4.1 Single-step feeding

Set the Bit3 bit of system parameter No.001 to 0, press the key to enter the single-step operation mode, and the page is displayed as follows:



4.1.1 Selection of increments

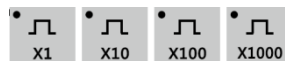


press the key to select the move increment, which will be

displayed on the page. When the BIT7 bit (SINC) of PLC status parameter K016 is 1, the



step value is invalid; When BIT7 is 0,



it is valid. If you press a



key, the page appears as follows:

MPG	MODE	01122. txt	S0000 T0100
ACT POS(REL)		01122 N0000	MODAL INFO
01122 N0000 U 0.000 W 0.000		G00 G97 G98 G21	
		G40 G54 M05 M09	
		M33 M41 M13 M30	
		S0300	
		F000000 mm/min	
		0 mm/min	
		COMPREHENSIVE	
		WHEEL IN(0.001	
		FEEDRATE 100%	
		FASTRATE 100%	
		SPI RATE 100%	
		PROC CNT 00000/00000	
		CUT TIME 000:00:00	
IN:		SYS TIME: 10:46:23	
▲	REL	ABS	ALL MONI GRAPH ▶

4.1.2 Movement direction selection

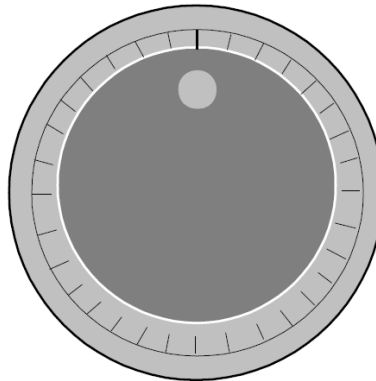
Press the  key once or  once to make the X axis negative or positive in a single step increment; Press the  key once or the  key to make the Z axis negative or positive in a single step increment. Press the key once or once to make the Y axis negative or positive in one step increment.

4.2 Handwheel (Hand Crank Pulse Generator) Feed

Set the Bit3 bit of the 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 following figure:



Handwheel shape drawing

4.2.1 Selection of increments

Press the     key to select move increments, which are displayed on the page. When the BIT7 bit (SINC) of PLC parameter K016 is 1, the  step value is invalid; When BIT7 is 0,     it is valid. If you press a  key, the page appears as follows:



4.2.2 Selection of moving axes and directions

Under Handwheel Operation, press the , , ,  key to select the appropriate shaft.

The handwheel feed direction is determined by the direction of handwheel rotation. In general, the handwheel is forward-feeding clockwise and negative-clockwise. If sometimes the handwheel is clockwise negative and counterclockwise is forward, the A and B signals at the end of the handwheel can be exchanged. The feed direction when the handwheel rotates can also be selected by bit0 to BIT4 of the parameter No 013.

4.2.3 Other Operations

1. Counterclockwise rotation, clockwise rotation, stop control



: Handwheel / one-step mode, press this key, the spindle rotates counterclockwise;



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



: Handwheel / Step, press this key and the spindle rotates clockwise.

2. Spindle jog



: The spindle is in jog state.

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

3. Coolant control

See paragraph 3.2.3 of this Part for details

4. Lubrication control

See section 3.2.4 for details

5. Manual tool change



T. CHANGE: Manual/Handwheel/ Step, press this key to change the knife in order.

6. Trimming of spindle magnification

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



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

4.2.4 Notes

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

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

The amount of movement per tick on the hand wheel

Handwheel increments 0.001 0.01 0.1 1

Coordinates specify a value of 0.001mm 0.01mm 0.1mm 1mm

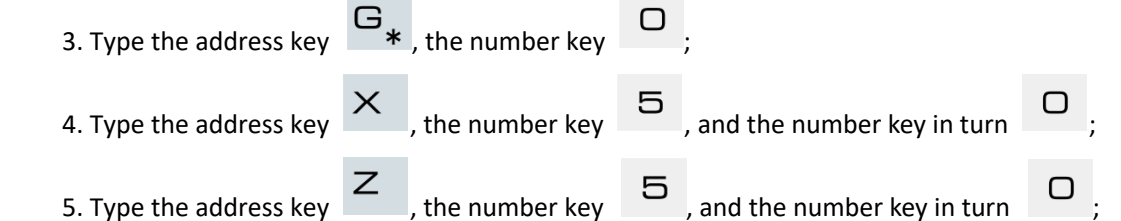
(Minimum input increment 0.001mm for example).

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

Note 2: The handwheel must not rotate faster than 5r/s, if it exceeds 5r/s, it may cause the scale value and the amount of movement to match.

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

- PRG



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

MDI	MODE	01122. txt		S0000	T0100
PRG	STAT	01122 N0000	MODAL	INFO	
(ABSOLUTE)	(RELATIVE)		G00	G97	G98 G21
X	0.000	U 0.000	G40	G54	M05 M09
		SPRM 0300	M33	M41	M13 M30
Z	0.000	W 0.000	S0300		
		SSPM 0000	F000000	mm/min	
		SMAX 6000	0	mm/min	
		SMIN 0000			
00000 ;			COMPREHENSIVE		
%			PRG RATE	0	
			FEEDRATE	100%	
			FASTRATE	100%	
			SPI RATE	100%	
			PROC CNT	00000/00000	
DATA: G0X50Z50	Ln:2		CUT TIME	000:00:00	
			SYS TIME	11:33:56	
▲	PRG	MDI	LOCAL DIR	USB DISK	

5.2 Execution of Code Words

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

MDI	MODE	01122. txt		S0000	T0100
PRG	STAT	01122 N0000	MODAL	INFO	
(ABSOLUTE)	(RELATIVE)		G00	G97	G98 G21
X	0.000	U 0.000	G40	G54	M05 M09
		SPRM 0300	M33	M41	M13 M30
Z	0.000	W 0.000	S0300		
		SSPM 0000	F000000	mm/min	
		SMAX 6000	0	mm/min	
		SMIN 0000			
00000 ;			COMPREHENSIVE		
G0 X50 Z50 ;			PRG RATE	0	
%			FEEDRATE	100%	
			FASTRATE	100%	
			SPI RATE	100%	
			PROC CNT	00000/00000	
DATA:	Ln:2		CUT TIME	000:00:00	
			SYS TIME	11:38:23	
▲	PRG	MDI	LOCAL DIR	USB DISK	

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

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

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, before executing the input program segment, if there is an error in the field input process, you can press the  key to cancel the reflection state and modify the program, or you can press the  key to clear everything and re-enter the correct program segment.

5.5 Other Operations

1. Adjustable spindle magnification Under the handwheel / single-step mode, when the analog voltage output is selected to control the spindle speed, the spindle speed can be modified.



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

2. Repairable fast magnification

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

3. Repairable feed rate



Under the input operation mode, press the  key to change the feed rate of the feed rate, and realize the real-time adjustment of 16 steps of the actual speed of 0~150% of the feed rate specified by the F code.

Chapter VI: Program Editing and Management

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

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

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

in order to open the program switch and edit the program. The operations allowed by operation level are described in Section 10.3 .

6.1 Establishment of the Program

6.1.1 Generation of program segment numbers

Programs can and cannot be programmed in program segment numbers, and programs are executed in the order in which program segments are programmed (except when invoked).

When the Auto Sequence switch on the switch settings page is off, the CNC does not automatically generate program segment numbers, but can manually program segment numbers when programming.

When the switch setting page "Auto Sequence" switch is on, the CNC automatically generates the program segment number, when editing, press the  key (or  key, or  key) to automatically generate the program segment number of the next program segment, and the increment value of the program segment number is determined by the CNC data parameter No 042 Settings (For details of the setting of automatic serial numbers, see Section 10.1.1 of this article).

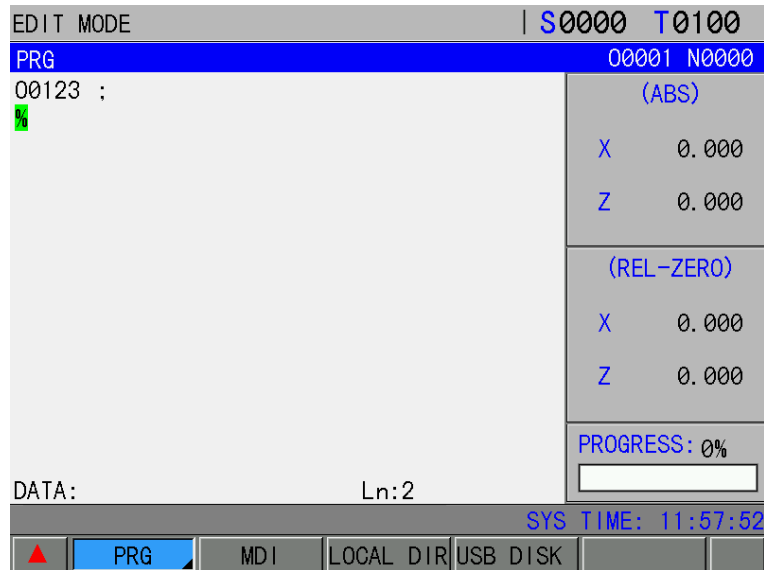
EDIT MODE		01122. txt		S0000 T0100	
ON-OFF SETTING		01122 N0000		MODAL INFO	
PAR SWITCH: OFF *ON				G00 G97 G98 G21	
PRG SWITCH: OFF *ON				G40 G54 M05 M09	
AUTO SEQ: *OFF ON				M33 M41 M13 M30	
				S0300	
				F000000 mm/min	
				0 mm/min	
				COMPREHENSIVE	
				PRG RATE 0	
				FEEDRATE 100%	
				FASTRATE 100%	
				SPI RATE 100%	
				PROC CNT 00000/00000	
				CUT TIME 000:00:00	
				SYS TIME: 11:48:26	
▲		PASSWORD	BACKUP	COORD	TIME
		SWITCH			

6.1.2 Input of Program Content

1. Press the  key to enter the editing operation mode; Press the  key to enter the program interface.

EDIT MODE		00001. txt		S0000 T0100	
PRG		00001 N0000			
00001 ;				(ABS)	
%				X 0.000	
				Z 0.000	
				(REL-ZERO)	
				X 0.000	
				Z 0.000	
				PROGRESS: 0%	
DATA:		Ln:2			
				SYS TIME: 11:56:18	
▲		PRG	MDI	LOCAL DIR	USB DISK

2. Type the address key , the number key , the , ,  (take the example of setting up the O 0123 program).
3. Press the  or  or  key to create a new program



4, according to the prepared parts program one by one input, each character entered, immediately on the screen to display the input of characters (the processing of the composite key is to repeatedly press this composite key to achieve alternating input), after the completion of the need to switch to other ways of working to store the program.

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

Note: 1. Under the editing mode, only the complete command word can be entered, and the system does not support individual letters or numbers.

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

6.1.3 Movement of the Cursor

1) Press the  key to enter the edit operation mode, press the  key to select the program content to display the page.

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

3) Press the  key, the cursor moves down one row, if the number of columns in which 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 and move the cursor to the right one 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 and move the cursor to the left one 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 and turn the page up;

7) Press the  key and turn the page down;

8) In the edit operation mode, program display page, press the  key, the cursor back to the beginning of the program.

6.1.4 Search for word and line numbers

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

The steps to find the method are as follows:

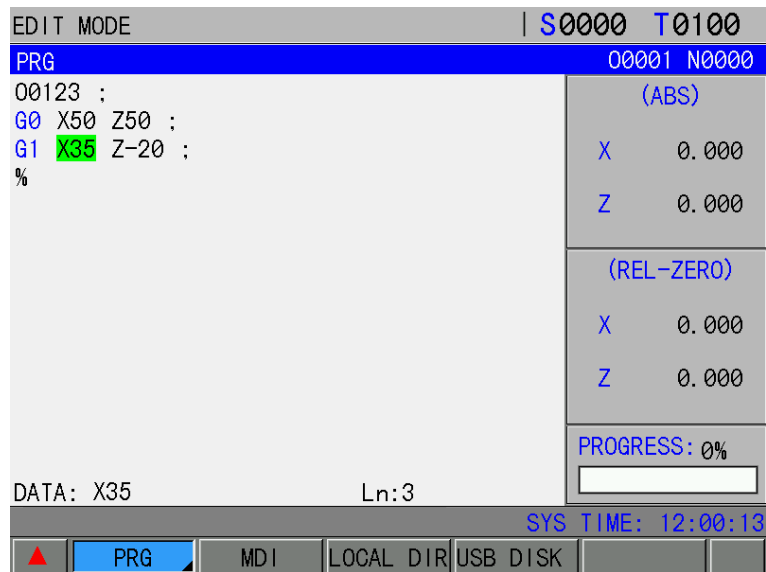
1) Press  to select the editing mode;

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

3) Enter the word you want to retrieve.

4) Press the  key (according to the position relationship between the character you want to find and the character where the current cursor is located, determine whether to press

the  key or  key), and the display page is as follows:



5) Press the Up/ Down Arrow key again to find the word in the next position.

Note 1: If it is not found, the words entered for retrieval will disappear.

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

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

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

The steps to find the method are as follows:

1) Select the editing operation mode, and the program content displays the page;

2) Press the  key (the address bar data should be empty);

3) Enter the program line number;

4) Press the  key (or  key or  key) and the cursor will jump to the entered line number of the program.

You can also press the function key F1 to enter the lower menu of the program editing page, click the find word function key, enter the desired search content to determine the search; Row search you can click jump and enter the desired line number to confirm the search.

6.1.5 Insertion of words

The how-to steps are as follows:

1) Select the editing operation mode, and the program content displays the page;

2) Enter the word you want to insert, press the  key, and the system will insert the input content on the left side of the cursor.

6.1.6 Deletion of words

The how-to steps are as follows:

1) Select the editing operation mode, and the program content displays the page;

2) Position the cursor to the location that needs to be deleted, press the  key, the system will delete the content where the cursor is located, if you press the  key continuously, you will continuously delete the program content on the right side of the cursor.

6.1.7 Word Modifications

The how-to steps are as follows:

1) Select the editing operation mode, and the program content displays the page;

2) Move the cursor to the place where you need to modify, enter the modified word, and

then press the  key, the system will replace the content of the cursor positioning with the input content.

6.1.8 Deletion of Single Program Segments

The how-to steps are as follows:

1) Select the editing operation mode, and the program content displays the page;

2) Move the cursor to the beginning of the deleted section (column 1), press the  key, and then press the  key to delete the segment where the cursor is located.

6.1.9 Deletion of Multiple Program Segments

The how-to steps are as follows:

1) Select the editing operation mode, and the program content displays the page;

2) Move the cursor to the beginning of the deleted program segment;

2) Enter the sequence number of the last program segment of the part you want to delete;

3) Press the  key to delete the program segment between the cursor and the marked address.

6.1.10 Block Deletion

The how-to steps are as follows:

- 1) Select the editing operation mode, and the program content displays the page;
- 2) Move the cursor over the first code word of the block to be deleted;
- 3) Enter the last code word of the program block you want to delete;
- 4) Press the  key to delete the block between the cursor and the marked address.

Note 1: If the deletion is successful, the code word entered will disappear, otherwise the deletion will fail.

If there are multiple specified characters that are the same down, the default is the one closest to the current cursor.

6.1.11 Copying of Single Program Segments

The how-to steps are as follows:

- 1) Select the editing operation mode, and the program content displays the page;
- 2) Move the cursor to the beginning of the copied program segment;
- 3) Press the  key, press the  key again, and copy the program section where the cursor is located.

6.1.12 Replication of Multiple Program Segments

The how-to steps are as follows:

- 1) Select the editing operation mode, and the program content displays the page;
- 2) Move the cursor to the first code word of the program segment to be copied;
- 3) Enter the sequence number of the last paragraph of the program segment to be copied;
- 4) Press the  key and copy the program segment between the cursor and the input characters to complete.

6.1.13 Copying of Program Blocks

The how-to steps are as follows:

- 1) Select the editing operation mode, and the program content displays the page;
- 2) Move the cursor over the first code word of the program block to be copied;
- 3) Enter the last code word of the program block you want to copy;

4) Press the  key and copy the block between the cursor and the input characters to complete.

You can also press the function key F1 to enter the lower menu of the program editing page,

click the copy button, press  or  select the program segment that needs to be copied, and you can also enter the program line number that needs to be copied in the copy box to copy.

6.1.14 Pasting of Program Blocks

The how-to steps are as follows:

- 1) Select the editing operation mode, and the program content displays the page;
- 2) Move the cursor to the position of the pasted program;

3) Press the  key to insert the last copied program content before the cursor to complete the paste operation.

You can also press the function key F1 to enter the lower menu of the program editing page, click the Paste key, and click the Paste key

6.2 Removal of Programs

6.2.1 Deletion of Individual Programs

The steps are as follows:

- 1) Select the editing operation mode and enter the program display page;

2) Type the address key in turn , the number key , , ,  (take the O0001 program as an example);

- 3) Press the  key and the O 0001 program is deleted.

6.2.2 Removal of all programs

The steps are as follows:

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

2) Type the address key , symbol key , number key , , ,  in turn;

3) Press the  key and all programs are deleted.

6.3 Selection of Programs

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

6.3.1 Search Methods

1) Select editing or automatic operation mode;

2) Press the  key, and enter the program content display screen;

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

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

Note: In step 4, under Edit Operation Mode, if the program does not exist, press the  key (or

 key or  key) and the CNC creates a new program.

6.3.2 Scanning Method

1) Select editing or automatic operation mode;

2) Press the  key and enter the program display screen;

3) Press the address key ;

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

PRG

EDIT MODE		00123.txt		S0000 T0100	
PROGRAM(DIR) 00123 N0000					
PRG CNT5		MEM 103 M		USED:2 M	
				FREE:101 M	
NAME		SIZE	DATE	PROGRAM PREVIEW:	
00001	9 (B)	00-06-17	00001;		
00005	377 (B)	00-06-17	%		
00123	32 (B)	00-06-23			
01122	3 (B)	00-06-18			
07000	1.20 (K)	00-05-10			
TIPS: [EOB] Open [O] New [P] Search [N] Rename [DEL] Del					
SYS TIME: 12:20:30					
▲	PRG	MDI	LOCAL DIP	USB DISK	



EDIT MODE		00123.txt		S0000 T0100	
PROGRAM (DIR)				00123 N0000	
PRG CNT5		MEM	103 M	USED: 2 M	FREE: 101 M
NAME		SIZE	DATE	PROGRAM PREVIEW:	
00001		9 (B)	00-06-17	00005;	
00005		377 (B)	00-06-17	SOM3;	
00123		32 (B)	00-06-23	G1X0Y0Z0F3000;	
01122		3 (B)	00-06-18	M3S600;	
07000		1.20 (K)	00-05-10	M8;	
				G1X300Y300Z300;	
				M3S1200;	
				G1X800Y800Z800;	
				M3S2400;	
				G1X1000Y1000Z1000;	
				M3S3600;	

TIPS: [EOB] Open [O] New [P] Search [N] Rename [DEL] Del

SYS TIME: 12:22:50

EOB

IN

Enter



6.4 Renaming of the Program

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

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

4) Press the  key.

You can also click Rename in the lower menu of the program directory.

6.5 Copying of Programs

Save the current program:

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

2) Address  key, type the new program number;

3) Press the  key.

6.6 Program Management

6.6.1 Program Directory

Press the  key to enter the program directory display page. In this page, the existing program names of CNC are displayed as a catalog table, and only a maximum of 10 program names can be displayed on a page, and when the number of existing programs exceeds 10, you can press the up/down page turning key to display another page of the program directory.



1) Open the program

Opens the specified program: + Ordinal + Key (or Key or Key)

or Ordinal + Key (or Key or Key).

In Edit Mode, if the ordinal number entered does not exist, the program is created.

2) Deletion of the program:

1. Edit by specifying the program by the Delete cursor.

2. Edit by + ordinal + key or Ordinal + key .

6.6.2 Number of Stored Programs

The maximum number of stored programs in this system is 400, and this item shows the number of part programs that the CNC can currently store and remain.

6.6.3 Storage Capacity

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

Chapter 7 Tool Offset and Alignment

In order to simplify programming and allow programming without considering the actual position of the tool, the lathe system provides three pairing methods of fixed-point tool pairing, test cutting tool pairing and return machine zero point tooling, and obtain tool bias data through tool operation.

7.1 Fixed-point knife pairing

The procedure is as follows:

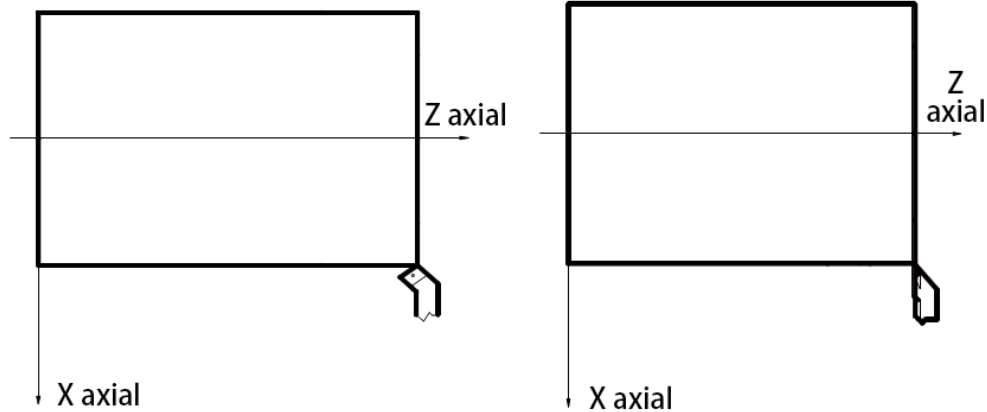


Figure A

Figure B

1. First determine whether the X and Z knife complement values are zero, if not, the knife complement values of all tool numbers must be cleared to zero;
2. Make the offset number in the tool 00 (such as T0100, T0300);
3. Choose any knife (generally the first knife in processing, this knife will be used as the reference knife);
4. Locate the tip of the reference knife to a certain point (to the knife point), as shown in Figure A;
5. Use G50 X Z code to set the coordinate system of the workpiece under the input operation mode and program status page;
6. Clear the coordinate value of the relative coordinate (U, W), and the zeroing method is detailed in Appendix 10 "List of Common Operations";
7. After moving the tool to a safe position, select another tool and move to the opposite knife point, as shown in Figure B;

8. Press the  key, press the  key, move the cursor by the  key to select the tool offset number corresponding to the knife;

9. Press the address key  , and then press the  key (or  key or  key), and the X-direction tool bias value is set to the corresponding offset number;

10. Press the address key  , press the  key again (or  key or  key), and the Z-direction tool bias value is set to the corresponding offset number;

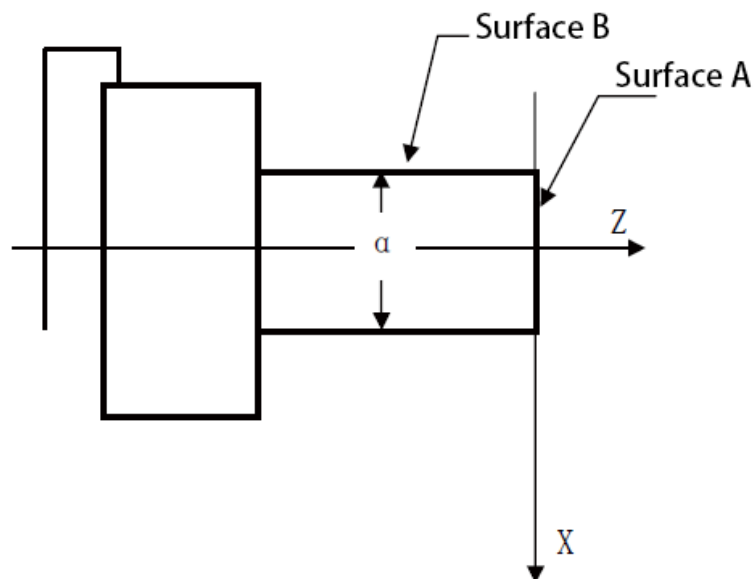
11. Repeat steps 7 ~ 10 to align other tools.

Note: When the knife is fixed, the original knife bias in the system must be cleared first, and the new knife bias value cannot be repeated multiple times when entering the new knife bias value by pressing U and W, and the method of zeroing the knife value can be found in section 7.4.4 of this article.

7.2 Try cutting the knife

The effectiveness of the test cut-to-knife method depends on the setting of bit5 bits of the CNC parameter No.012.

The operation steps are as follows (establishing the workpiece coordinate system from the workpiece end face):



1. Select any knife to make the tool 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 bias interface, select the tool bias page, press the  key,  key to move the cursor to select the corresponding offset number of the tool;

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

5. Make the tool cut along the B surface;

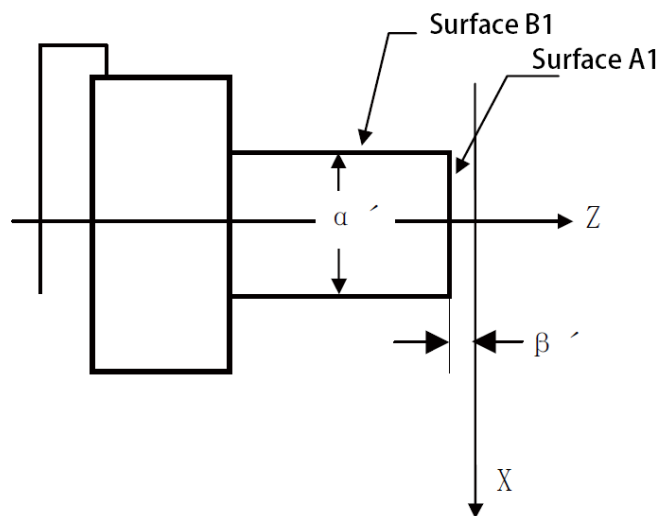
6. In the case of the X axis not moving, exit the tool along the Z axis and stop the spindle rotation;

7. the measurement diameter of " α " (assuming that the $\alpha = 15$);

8. Press the  key to enter the bias interface, select the tool bias page, press the  key,  key to move the cursor to select the corresponding bias number of the tool;

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

10. Move the knife 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 when the Z axis is not moving, 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 bias interface, select the tool bias page, press the  key and  key move the cursor to select the corresponding bias number of the tool;

15. Press the address key , symbol key , number key  and  key (or  key or  key) in turn;

16. Make the tool cut along the B1 surface;

17. In the case of the X axis not moving, exit the tool along the Z axis and stop the spindle rotation;

18, the measurement distance " α " (assuming $\alpha' = 10$);

19. Press the  key to enter the bias interface, select the tool bias page, press the  key,  key to move the cursor to select the offset number corresponding to the tool;

20. Type the address key , the number key ,  and the  key (or  key or  key) in turn;

;

21. Repeat steps 10 ~ 20 for other tool-to-knife methods.

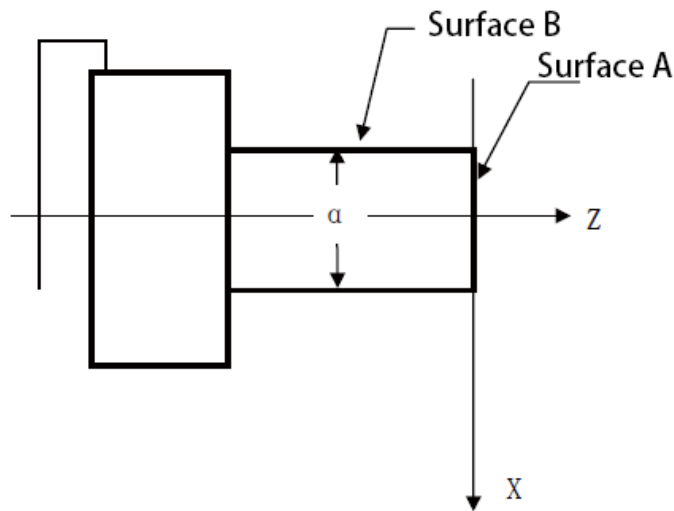
Note: The tool complement value for this pair method can be large, so the CNC must be set to perform the tool complement at coordinate offset (the BIT4 bit of the CNC parameter NO.003 is set to 1), and the first program segment is used The T code performs tool length compensation or the first move code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

7.3 Return to the machine tool zero point to the knife

There is no reference knife non-reference knife problem with this tool pairing method, and

when the tool wears out or adjusts any one of the knives, it is enough to re-align the knife. Return the machine machine to zero points before the knife. After the power is cut off, as long as the machine tool is returned to zero point once, the processing can continue, and the operation is simple and convenient.

The operation steps are as follows (establishing the workpiece coordinate system from the workpiece end face):



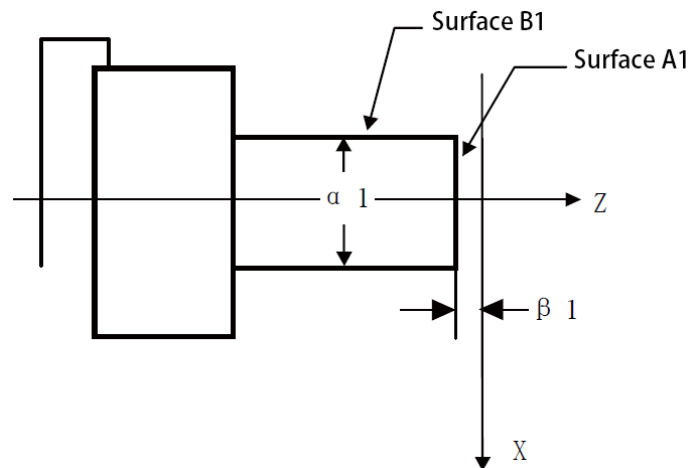
1. Press the  key to enter the machine tool back to zero operation mode, so that the two axes return to the machine tool zero;
2. Select any knife so that the offset number in the tool is 00 (such as T0100, T0300);
3. Make the tool cut along the A surface;
4. In the case of the Z axis not moving, exit the tool along X and stop the spindle rotation;
5. Press  to enter the bias interface, select the tool offset page, press the  key,  key to move the cursor to select a certain bias number;
6. Press the address key , number key  and  key ((or  key or  key) in turn, and the Z axis bias value is set;
7. Make the tool cut along the B surface;
8. In the case of the X axis not moving, exit the tool along Z and stop the spindle rotation;
- 9, the measurement distance " α " (assuming that the $\alpha = 15$);
10. Press  to enter the bias interface, select the tool bias page, press the  key

key or  key move the cursor to select the bias number;

11. Type the address key  , the number key  ,  and  the key  (or  key) in turn, and the X-axis tool offset value is set;

12. Move the tool to a safe tool changing position;

13. Change another knife so that the offset number in the tool is 00 (such as T0100, T0300);



14. Make the tool cut along the surface of A1;

15. Exit the tool along the X axis without moving the Z axis, and stop the spindle rotation;

Measure the distance " $\beta 1$ " between the A1 surface and the origin of the workpiece coordinate system (assuming $\beta 1 = 1$);

16. Press  to enter the bias interface, select the tool bias page, press the  key,  key to move the cursor to select a certain bias number;

17. Press the address key  , symbol key  , number key  and  key  (or  key) in turn, and the Z axis tool offset value is set;

18. Make the tool cut along the B1 surface;

19. In the case of the X axis not moving, exit the tool along Z and stop the spindle rotation;

20. the measurement distance " $\alpha 1$ " (assuming that the $\alpha 1 = 10$);

21. Press  to enter the bias interface, select the tool bias page, press the  key,  key to move the cursor to select the offset number;

22. Type the address key , the number key ,  and the  key
(or  key or  key) in turn, and the X-axis tool offset value is set;
23. Move the tool to a safe tool change position;
24. Repeat steps 12 ~ 23 to complete the alignment of all knives.

Note 1: The machine tool must be equipped with a machine zero switch to perform the return machine zero point tool operation.

Note 2: After returning to the machine with a zero point, the G50 code cannot be performed to set the workpiece coordinate system.

Note 3 to entry: The CNC must be set to perform tooling at coordinate offset (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 tool length compensation.

7.4 Setting and modifying tool offset values

Press the  key to enter the bias interface , and display the No.000 ~ No.032 bias number through the  key and  key respectively.

EDIT MODE

01122. txt

| S0000 T0100

OFFSET WEAR:

01122 N0000

NO.	TYP	X	Z	R	T
00	OFT	0.000	0.000	0.000	0
	WEAR	-----	-----	-----	
01	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	
02	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	
03	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	
04	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	

RELATIVE

U 0.000

W 0.000

ABSOLUTE

X 0.000

Z 0.000

Memory: X I Record Z J Record

00OFT=

SYS TIME: 12:33:22

▲

OFT

T-Len

WEAR

T-NOSE

MACRO

▶

Tool wear (two axes)

7.4.1 Setting of tool offset values

1. Press the  key to enter the tool bias page, press the  key and  key to select the required page;

2. Move the cursor to the position of the tool bias and wear number to be entered.

Scanning method: Press  and  keys to move the cursor.

Search Method: Use the key sequence described below to move the cursor directly to the position you typed.

 + Offset sign +  (or  key or  key);

3. Press the address key  or  after entering the number (you can enter the decimal point);

4. After pressing the  key (or  key or  key), the CNC automatically calculates the amount of tool bias and displays it on the page.

7.4.2 Modification of tool offset values

1. Move the cursor to the position of the tool offset number to be changed according to the method described in Section 7.4.1 of this Chapter;

2. If you want to change the tool offset value of the X axis, type U; For Z-axis, type W;

3. Type the increment value;

4. Press the  key (or  key or  key) to add the current tool offset value with the typed increment value, and the result is displayed as a new tool offset value.

Example: The X axis has been set to a tool offset value of 5.678, and the keyboard is used to enter the increment U1.5

The new X-axis has a tool offset value of 7.178 (=5.678+1.5).

7.4.3 Tool offset value cleared to zero

1. Move the cursor to the position of the compensation number to be cleared.

2. Method 1:

If you want to zero the tool offset value of the X axis, press the  key, press the  key again (or  key or  key), and the tool offset value of the X axis is cleared;

If you want to zero the tool offset value of the Z axis, press the  key, press the  key (or  key or  key), and the tool offset value of the Z axis is cleared;

Method 2:

If the X-to-current tool offset value is α , enter U- α , press the  key (or  key or  key), and the tool bias value for the X axis is zero;

If the Z-to-current tool offset value is β , enter W- β , press the  key (or  key or  key), and the tool offset value for the Z axis is zero;

7.4.4 Tool wear value setting and modification

In order to prevent the tool bias value setting, mistaken operation when modifying (no decimal point is entered, the decimal point position is not equal), resulting in too large a modification of the tool bias value, resulting in phenomena such as colliding knives, which is conducive to the operator's intuitive judgment of the wear degree of each knife, the lathe system sets the tool wear page. When the machining size is not allowed to modify the tool compensation value due to tool wear and other reasons, it can be set or modified in the tool wear amount. The input range of the wear value of the machined tool is set by the data parameter No 140. Tool wear data is lost to power saving.

The setting and modification method of the tool wear value is basically the same as the setting and modification method of the tool offset value, and the input of wear amount is carried out with U (X axis), W (Z axis), V (Y axis).

7.4.5 No. 0 knife deflects the translation workpiece coordinate system

When the BIT6 bit of the CNC parameter No.012 is set to 1, the 0-gauge tool deflection workpiece coordinate system is valid.

When a value is entered in 0-gauge cutter bias, the workpiece coordinate system is offset by the input value.

EDIT MODE		01122. txt		S0000 T0100	
OFFSET WEAR:		01122 N0000			
NO.	TYP	X	Z	R	T
00	OFT	0.000	0.000	0.000	0
	WEAR	-----	-----	-----	0
01	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
02	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
03	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
04	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0

Memory: X I Record Z J Record

00OFT=

▲ ►

SYS TIME: 12:33:22

Enter the number 0 knife forward

MDI MODE		00001. txt		S0000 T0100	
OFFSET WEAR:		00001 N0000			
NO.	TYP	X	Z	R	T
00	OFT	100.000	100.000	0.000	0
	WEAR	-----	-----	-----	0
01	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
02	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
03	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0
04	OFT	0.000	0.000	0.000	0
	WEAR	0.000	0.000	0.000	0

Memory: X I Record Z J Record

00OFT=

▲ ►

SYS TIME: 14:49:45

Enter a number 0 knife deflected backward

As shown in the figure above, after entering X100, Z100 with a tool bias of 0, the workpiece coordinate system is offset by X100, Z100.

Chapter 8 Automatic Operation

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

Please note that the functions related to the operator panel buttons in this chapter are described in the standard PLC procedure for lathe systems.

8.1 Autorun

8.1.1 Selection of Running Programs

1. Search method

- 1) Select editing or automatic operation mode;

- 2) Press the  key, and enter the program content display screen;

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

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

2. Scanning method

- 1) Select editing or automatic operation mode;

- 2) Press the  key, and enter the program display screen;

- 3) Press the address key ;

- 4) Press the  or  key to display the next or previous program;
- 5) Repeat steps 3) and 4) to show the programs that are deposited one by one.

3. Cursor confirmation method

- a) Select the automatic operation mode (must be in a non-running state).

- b) Press the  key to enter the program directory to display the page (if necessary,

- press the  key,  key);
- c) Press the , , ,  keys to move the cursor to the program name to be selected.
- d) press the  key (or  key or  key);

8.1.2 Startup of Autorun

1. Press the  key to select the automatic operation mode;
2. Press the  key to start the program, and the program runs automatically.

Note: The program starts from the line where the cursor is located, so you should check whether the

cursor is on the program segment that needs to be run before pressing the  key to run.

8.1.3 Stop of automatic operation

* Code stop (M00).

1、 M00

After the program segment containing M00 is executed, the automatic operation is

stopped, and the modal functions and states are all saved. After pressing the Panel  key or the External Run key, the program continues.

2、 M01

Press the  key, select Stop Light on, select Stop function is valid. After executing the program segment containing M01, the automatic operation is stopped, and the modal functions

and states are all saved. After pressing the Panel  key or the External Run key, the program continues.

* Press phase key to stop

1. After pressing the  key or external pause key during automatic operation, the machine tool shows the following states:

- (1) Machine tool feed deceleration stop;
- (2) The modal function and state are saved;

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

2. Press the Reset key



(1) All axis movement stops;

(2) M, S function output is invalid (can be set by the parameter setting after pressing the



key whether to automatically close the spindle counterclockwise rotation / clockwise rotation, lubrication, cooling and other signals);

(3) Automatic operation end, modal function, state maintenance.

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

4. When the conversion operation mode is converted to machine tool zeroing, handwheel / single step, manual, program zeroing during automatic operation, the current program segment is immediately "suspended"; When you convert to editing and entering mode during automatic operation, "Pause" is displayed only after the current program segment is run.

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

NOTE 2: Pressing the E-STOP button before power-on and shutdown reduces the electrical shock of the device;

Note 3: After the emergency stop alarm is lifted, the operation of returning 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 operation of returning to the zero point of the machine tool shall not be carried out);

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

8.1.4 Automatically run from any segment

Press the



key to enter the editing operation mode, press the



key to enter

the program interface, press the



key or



key to select the program content

page;



1. Move the cursor to the program segment that is ready to start running (such as starting from the second line, moving the cursor to the beginning of the second line);

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

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

8.1.5 Feed, fast speed adjustment

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

* Adjustment of feed rate



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

Note 1: The value specified by F in the feed multiplier adjustment procedure;

Note 2: Actual Feed Rate = F The specified value × Feed rate.

* Fast magnification adjustment

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

four real-time adjustment.

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

The actual fast moving rate of the X axis = The value set by No.022 × fast magnification

The Z-axis actual fast moving rate = No.023 sets the value × fast multiplier

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

8.1.6 Spindle speed adjustment

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



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

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

Example:

The CNC parameter NO.037 is set to 9999, the S9999 code is executed, and the spindle magnification is selected at 100%, the actual output analog voltage $\approx 10 \times 100\% = 10V$

8.2 Status of the Runtime

8.2.1 Single-Stage Operation

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

In the automatic operation mode, the single-segment program switch is opened as follows:

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

When running a single segment, the CNC stops running after the current program segment is executed; To continue with the next program segment, you need to press the  key again, and so on until the program finishes running.

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

Note 2: When executing the fixed loop G90, G92, G94, G70 ~ G76 code, the single-segment state can be found in the first "Programming Instructions";

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

8.2.2 Empty Run

8.2.3 The machine is locked for operation

Under the automatic operation mode, the machine lock switch is opened as follows:

Press the  key to make the machine lock in the status indicator area  to light up, indicating that the machine tool is locked in the operation state;

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

1. The machine tool drag plate does not move, the "machine tool coordinates" in the

comprehensive coordinate page under the position interface do not change, and the relative coordinates, absolute coordinates and residual movement are constantly refreshed, the same as when the machine tool lock switch is in the off state;

2, M, S, T code can be executed normally.

8.2.4 Accessibility locks up operation

Under the automatic operation mode, the machine lock switch is opened as follows:

Press the  key to make the auxiliary function lock in the status indicator area to illuminate the operation light , indicating that the auxiliary function lock operation state is entered; At this time, the M, S, T code is not executed, and the machine tool drags the plate to move. It is usually used in conjunction with the machine lock function for program calibration.

Note: When the auxiliary function lock is valid, it does not affect the execution of M00, M30, M98, M99.

8.2.5 Segment selection

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

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

Press the  key to make the program segment in the status indicator area select jump light  on;

Note: When the program segment jump switch is not turned on, the program segment with a "/" 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. Under the automatic operation mode, press the  key, and the coolant is switched

on/off;

2. Press any key of , , , ,  or  to realize the conversion of operation mode;

3. Press the  key to reset the CNC.

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

Chapter 9: Back to Zero Operation

The function of the keys 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 tool manufacturer's manual.

Please note that the functions related to the operator panel buttons below in this chapter are described in the standard PLC procedure for lathe systems.

9.1 Program zeroback

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 is returned to this position after the program zero operation is performed.

9.1.2 Procedure for program zeroing



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

PRG	ZERO	00005.txt		S0000	T0100
ACT POS(REL)			00005 N0000	MODAL INFO	
00005 N0000			G00 G97 G98 G21		
			G40 G54 M05 M09		
			M33 M41 M13 M30		
			S0300		
			F000000 mm/min		
			300 mm/min		
			COMPREHENSIVE		
			PRG RATE 300		
			FEEDRATE 100%		
			FASTRATE 100%		
			SPI RATE 100%		
			PROC CNT 00000/00000		
			CUT TIME 000:00:00		
IN:			SYS TIME: 15:45:02		
▲	REL	ABS	ALL	MON I	GRAPH

2. Press any arrow key of the X, Z, Y axis to return to the zero point of the X, Z or Y axis program;
3. The machine tool axis moves along the zero point of the program, and after returning to the zero point of the program, the axis stops moving, and the end indicator light is on.



Indicator indicating the end of return to zero

Note 1: After the zero operation of the program, the current tool bias state is not changed, if there is a tool bias, the position returned is the position set with G50 is the position with the tool bias.

9.2 Machine return to zero

9.2.1 Machine zero

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

9.2.2 Operation steps for machine tool zeroing



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

PRG		ZERO		00005.txt		S0000		T0100	
MONITOR				00005 N0000		MODAL INFO			
(REL)				(ABS)		G00 G97 G98 G21			
U -22.600		X		77.400		G40 G54 M05 M09			
W 24.920		Z		124.920		M33 M41 M13 M30			
						S0300			
						F000000 mm/min			
						300 mm/min			
PRG						COMPREHENSIVE			
G1 X0 Y0 Z0 F3000 ;						PRG RATE 300			
M3 S600 ;						FEEDRATE 100%			
M8 ;						FASTRATE 100%			
G1 X300 Y300 Z300 ;						SPI RATE 100%			
M3 S1200 ;						PROC CNT 00000/00000			
G1 X800 Y800 Z800 ;						CUT TIME 000:00:00			
MDI:				Ln:6		SYS TIME: 15:49:51			
▲		REL		ABS		ALL		MON I	
								GRAPH	
								▶	

2. Press , ,  or  keys, select back X, Z, Y or 4th axis machine zero;

3. The machine tool moves along the zero point direction of the machine tool, returns to the zero point of the machine tool after the deceleration signal and zero signal detection, at this time the axis stops moving, and the return to zero ends The indicator light is on.



Indicator indicating the end of return to zero

Note 1: If the CNC machine tool is not installed with the zero point of the machine tool, the machine tool back to zero operation shall not be used;

NOTE 2: The zero-end LED is off when:

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

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

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

Note 5: After performing the machine tool zeroing operation, the original workpiece coordinate system is reset and needs to be reset with the G50.

9.3 Other operations in zeroback mode

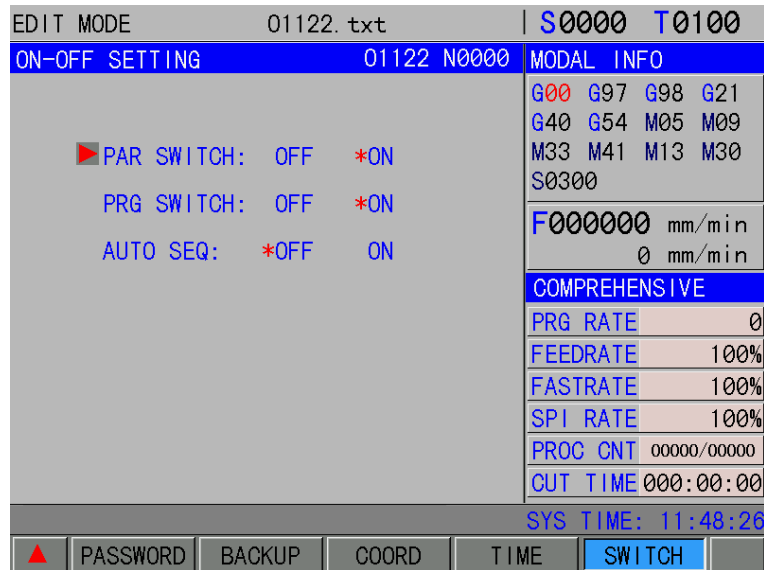
1. Press the  key, the spindle rotates counterclockwise;
2. Press the  key, the spindle stops;
3. Press the  key, the spindle rotates clockwise;
4. Press the  key to switch between coolant on/off;
5. Lubrication control (see Chapter 3 of this article for details);
6. Press the  key and manually change the knife relatively;
7. Trimming of spindle magnification;
8. Repair of rapid magnification;
9. Modification of feed rate.

Chapter 10 Data Setup, Backup, and Recovery

10.1 Settings for 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 sequence numbers, and the page shows the following figure:



1. Press the  key to enter the setting interface, press the  or  key to enter the switch setting page;

2. Press  or  key to move the cursor to the item to be set;

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

Parameters can be modified only when the parameter switch is on; Programs can be edited only when the program switch is on; The program sequence number is automatically added when the program edits only when the auto sequence switch is on.

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



and hold the



key and then hold the key to eliminate the alarm, if you switch the switch

state of the parameter, then do not alarm. For safety reasons, after the parameter modification is complete, be sure to set the parameter switch to Off.

10.1.2 Graphics Settings

Press the  key twice to enter the track page



The meaning of graphical parameters

Coordinate system settings: According to the coordinate systems of the front and rear tool holders, etc., the lathe system can display 8 kinds of graphical trajectories A: figures

Zoom in and out of shape track on the graphic display page, you can zoom in and out of

shape track in real time by editing keys  and  on the keyboard; Or switch via function keys.

B: Start, stop and clear the graph track display on the graph track display page, press the

 key once to start drawing; Press the  key once to stop drawing; Press the  key once to clear the current graph track; Or switch via function keys.

C: The movement of the graph track display

On the graphic track display page, you can press the arrow keys to achieve the movement of the graphic track.

10.1.3 Setting of Parameters

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

Press the  key to enter the parameter interface, press the  or  key

to switch the parameters page, as shown in the following figure:

A. Modify the settings of the status parameters

1. Byte modification:

- 1) Turn on the parameter switch;
- 2) Select the input method;
- 3) Move the cursor over the parameter number you want to set:

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

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

4) Enter the new parameter value;

5) Press the  key (or  key or  key), the parameter value is entered and displayed;

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

Example:

Set bit5 (DECI) for status parameter No 004 to 1, and the rest remain the same. Follow the steps above to move the cursor over No 004, type 011000000 in the prompt line in turn, as shown in the following figure:

EDIT MODE		00005. txt		S0000 T0100	
STATUS PARAMETER 00005 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000011	007	10101111	012	01111001
003	00110111	008	00011110	013	11000000
004	01000000	009	00000000	014	01000000
005	00110001	010	00011000	164	00001001

BIT0: (0:Metric 1:Inch)Output
 BIT1:NULL
 BIT2:NULL
 BIT3:NULL
 BIT4:Value of Tool Offset(0:Diameter 1:Radius)
 BIT5:Tool change,tool supplement number changes with too
 BIT6:Speed of G00 when MPG Trail Cut(0:Manual 1:Rapid)
 BIT7:Only Write-in moving axes when Teachin 0:NO 1:YES

NO.004 = 01100000

SYS TIME: 10:52:22

▲ ALLPARM PROC TE USERPARM **BITPARM** NUMPARM ►

Press the  key (or  key or  key) and the parameter modification is complete. The display page is as follows:

MDI MODE		00005. txt		S0000 T0100	
STATUS PARAMETER 00005 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000011	007	10101111	012	01111001
003	00110111	008	00011110	013	11000000
004	01000000	009	00000000	014	01000000
005	00110001	010	00011000	164	00001001

BIT0: (0:Metric 1:Inch)Output
 BIT1:NULL
 BIT2:NULL
 BIT3:NULL
 BIT4:Value of Tool Offset(0:Diameter 1:Radius)
 BIT5:Tool change,tool supplement number changes with too
 BIT6:Speed of G00 when MPG Trail Cut(0:Manual 1:Rapid)
 BIT7:Only Write-in moving axes when Teachin 0:NO 1:YES

NO.004 =

SYS TIME: 12:22:05

▲ ALLPARM PROC TE USERPARM **BITPARM** NUMPARM ►

2. Bitwise modification:

- 1) Turn on the parameter switch;
- 2) Select the input method;
- 3) Move the cursor to the parameter number you want to set;

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

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



key).

4) Press to jump into a certain bit of the parameter, at this time the bit is inverted.

Press or keys to move the cursor over the bit to be modified, and type 0 or 1 as desired;

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

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

Example:

Set the Bit5 bit (DECI) of status parameter No 004 to 1, and the rest remain the same.

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

MDI MODE		00005. txt		S0000 T0100	
STATUS PARAMETER				00005 N0000	
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000011	007	10101111	012	01111001
003	00110111	008	00011110	013	11000000
004	01000000	009	00000000	014	01000000
005	00110001	010	00011000	164	00001001

BIT0: (0:Metric 1:Inch)Output
 BIT1:NULL
 BIT2:NULL
 BIT3:NULL
 BIT4:Value of Tool Offset(0:Diameter 1:Radius)
 BIT5:Tool change,tool supplement number changes with too
 BIT6:Speed of G00 when MPG Trail Cut(0:Manual 1:Rapid)
 BIT7:Only Write-in moving axes when Teachin 0:NO 1:YES

NO. 004 =

SYS TIME: 10:58:25

▲ ALLPARM
 PROC TE
USERPARM
BITPARM
NUMPARM
▶

Press or keys to move the cursor over the Bit5 bit, as shown in the following figure:

MDI MODE		00005.txt		S0000 T0100	
STATUS PARAMETER				00005 N0000	
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000011	007	10101111	012	01111001
003	00110111	008	00011110	013	11000000
004	01000000	009	00000000	014	01000000
005	00110001	010	00011000	164	00001001

BIT0: (0:Metric 1:Inch)Output
 BIT1:NULL
 BIT2:NULL
 BIT3:NULL
 BIT4:Value of Tool Offset(0:Diameter 1:Radius)
 BIT5:Tool change,tool supplement number changes with too
 BIT6:Speed of G00 when MPG Trail Cut(0:Manual 1:Rapid)
 BIT7:Only Write-in moving axes when Teachin 0:NO 1:YES

NO.004 =

SYS TIME: 10:59:56

Enter 1 and the parameters are modified.

MDI MODE		00005.txt		S0000 T0100	
STATUS PARAMETER				00005 N0000	
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	006	01100000	011	00000000
002	01000011	007	10101111	012	01111001
003	00110111	008	00011110	013	11000000
004	01100000	009	00000000	014	01000000
005	00110001	010	00011000	164	00001001

BIT0: (0:Metric 1:Inch)Output
 BIT1:NULL
 BIT2:NULL
 BIT3:NULL
 BIT4:Value of Tool Offset(0:Diameter 1:Radius)
 BIT5:Tool change,tool supplement number changes with too
 BIT6:Speed of G00 when MPG Trail Cut(0:Manual 1:Rapid)
 BIT7:Only Write-in moving axes when Teachin 0:NO 1:YES

NO.004 =

SYS TIME: 11:00:57

B. Modification settings of data parameters and screw complement data

- 1) Turn on the parameter switch;
- 2) Select the input method;
- 3) Move the cursor to the parameter number you want to set;
- 4) Enter the new parameter value;

5) Press the  key (or  key or  key), the parameter value is entered and displayed;

6) For safety reasons, it is recommended to turn off the parameter switch after all parameters are set.

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

Example 1:

Set the data parameter No 022 to 3800. Follow the above steps to move the cursor over No

022 and type 3800 in the prompt line in turn, as shown in the following figure:

MDI	MODE	00005. txt	S0000 T0100
NUM PARAMETER			00005 N0000
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	
MIN	0	MAX	99999
		P:1/36	
NO. 022 =		3800	
SYS TIME: 13:13:43			
	ALLPARM	PROC TE	USERPARM
		BITPARM	NUMPARM
			

Press the  key (or  key or  key) and the parameter modification is complete. The display page is as follows:

MDI MODE		00005.txt	S0000 T0100
NUM PARAMETER			00005 N0000
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)	3800	
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	
MIN		0	MAX 99999 P:1/36
NO.022 =			
SYS TIME: 13:14:18			
▲	ALLPARM	PROC TE	USERPARM BITPARM NUMPARM ▶

Example 2:

Set the value of the X axis of the spiral data No 000 to 12 and the value of the Z axis to 30.

Follow the steps above to move the cursor over the spiral data No 000 and type X12 in the prompt line.

Press the  key (or  key or  key) and the data modification is complete. The display page is as follows:

MDI	MODE	00005. txt			S0000	T0100
PITCH COMP			00005 N0000			
NO.	X	Z	NO.	X	Z	
000	12	0	011	0	0	
001	0	0	012	0	0	
002	0	0	013	0	0	
003	0	0	014	0	0	
004	0	0	015	0	0	
005	0	0	016	0	0	
006	0	0	017	0	0	
007	0	0	018	0	0	
008	0	0	019	0	0	
009	0	0	020	0	0	
010	0	0	021	0	0	
NO. 000 =						
SYS TIME: 13:15:22						
PITCH						

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

MDI	MODE	00005. txt			S0000	T0100
PITCH COMP			00005 N0000			
NO.	X	Z	NO.	X	Z	
000	12	30	011	0	0	
001	0	0	012	0	0	
002	0	0	013	0	0	
003	0	0	014	0	0	
004	0	0	015	0	0	
005	0	0	016	0	0	
006	0	0	017	0	0	
007	0	0	018	0	0	
008	0	0	019	0	0	
009	0	0	020	0	0	
010	0	0	021	0	0	
NO. 000 =						
SYS TIME: 13:17:17						
PITCH						

10.2 Data Restore 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 is backed up and restored without affecting part programs stored in CNC. The Data Actions page appears as follows:



1. Open the parameter switch;
2. Press the  key to enter the input operation mode, press the  key (press the  or  key if necessary) to enter the data operation page;
3. Press the  key to enter the data backup page, and press the  key to enter the data restore page.
4. Move the cursor to the option to be operated, press the  key to select/ cancel the operation option;

Note 1: When performing data backup and restore operations, do not power off, and recommend that you do not perform other operations until the operation is completed;

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

10.3 Setting and Modifying Permissions

In order to prevent the machining program and CNC parameters from being maliciously modified, the lathe system has raised the privilege setting function, and the password level is divided into 4 levels, from high to low, it is 2 (machine tool manufacturer level) and 3 levels (set H). Ready 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 of the permission

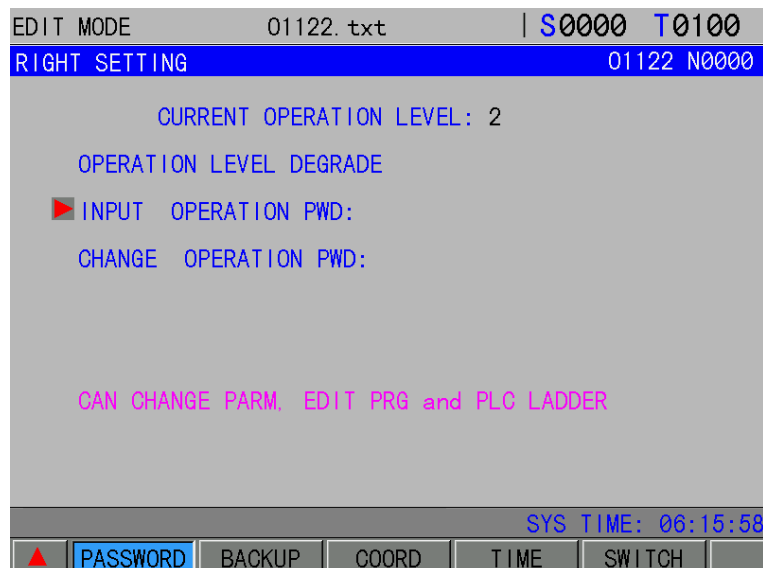
settings page: ".

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

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

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

Level 5: No password level, can be operated by the machine operation panel, can not modify the tool compensation data, can not select parts program, can not edit the program, can not modify the STATUS parameters, data parameters and screw complement data of the CNC.



After entering the permission setting page, the cursor first stays on the "Enter Operation

Password" line. You can press  or  key move the cursor over the corresponding action.

a) Press the  key once and move the cursor up one line. If the current cursor is on the "Operation Level Degraded" row (first row), after pressing the  key once, the cursor moves to the "Change Operation Password" line (the last line);

b) Press the  key once and move the cursor up one line. If the current cursor is on the trailing line, after pressing the  key once, the cursor moves to the first line.

10.3.1 Entry of 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, a "*" sign is added to the display);

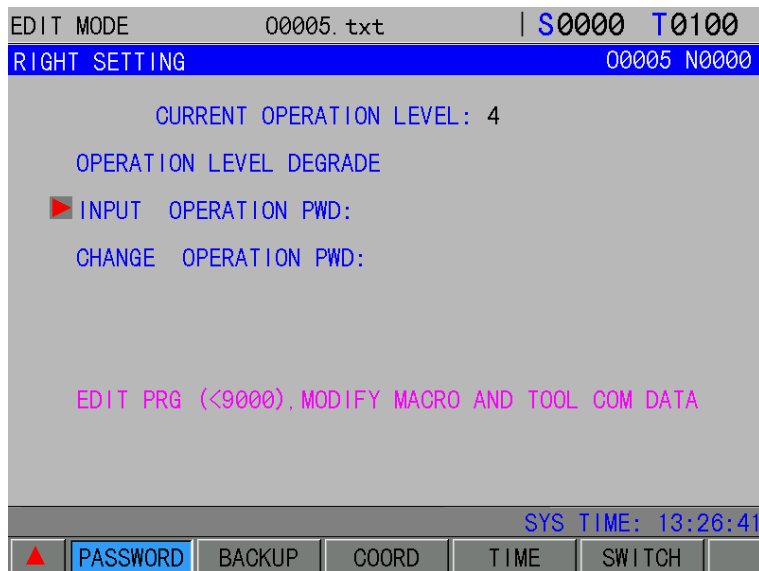
3. Enter and press the  key (or  key or  key) to enter the operation level corresponding to the password.

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

The details are as follows:

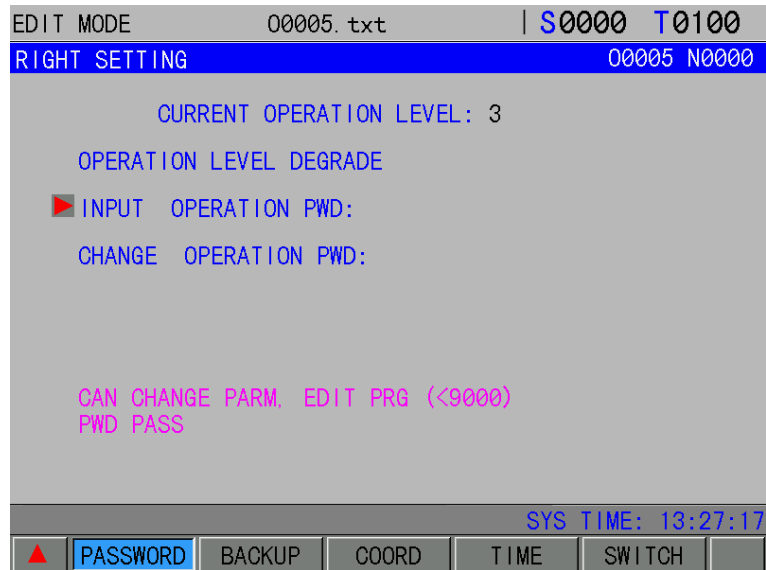
The level of operation		Password data length	Initial password
3	level	5 digits	12345
4	level	4 digits	1234
5	level	not	not

Example: The current operation level of the CNC is level 4, and the display page is as follows.
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 "To modify the parameters, to edit the program." "

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



Note: If the current operation permission is less than or equal to level 3 (level 3, level 4 or level 5), the operation permission remains unchanged when power-on again. If the last operation permission is level 2, the operation permission defaults to level 3 when powering on again.

10.3.2 Change of Operation Password

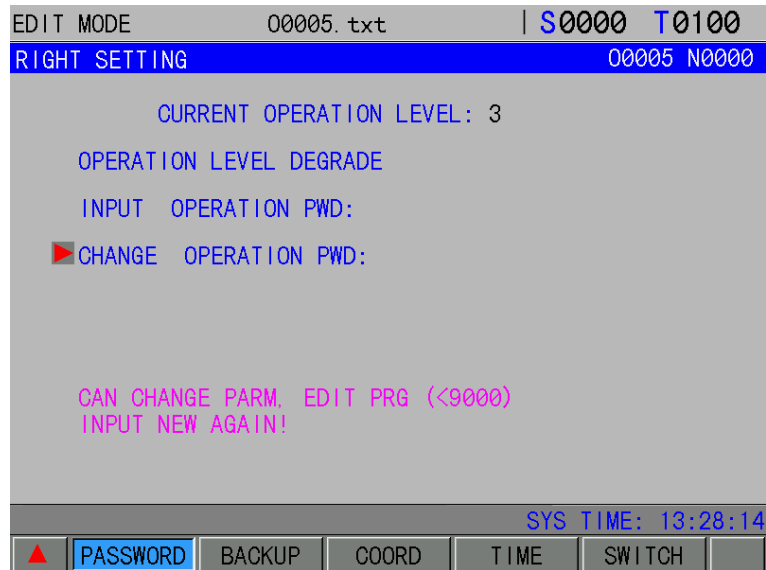
To change your password, follow these steps:

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

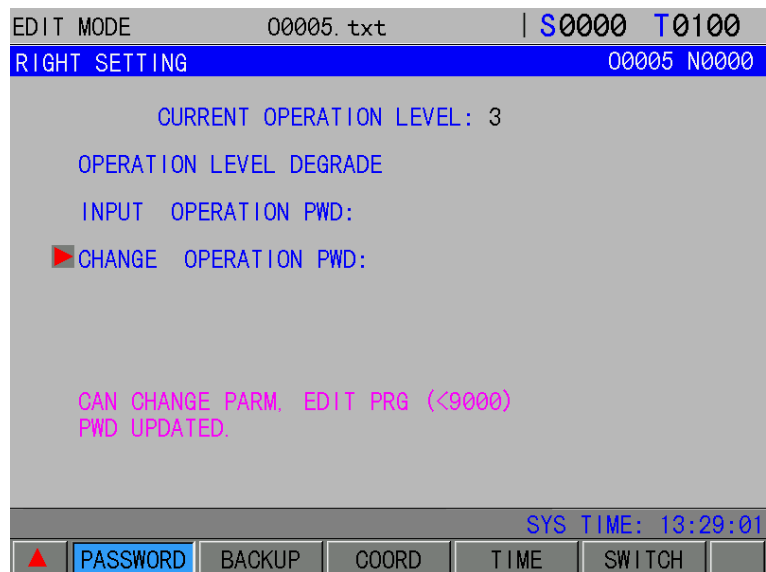
2. Move the cursor to the "Change Operation Password" line;

3. Enter the new operation password and press the  key (or  key or  key);

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



5. Press the key after entering the operation password again, if the password entered twice is the same, the CNC prompts "Password updated", and the operation password is changed successfully.



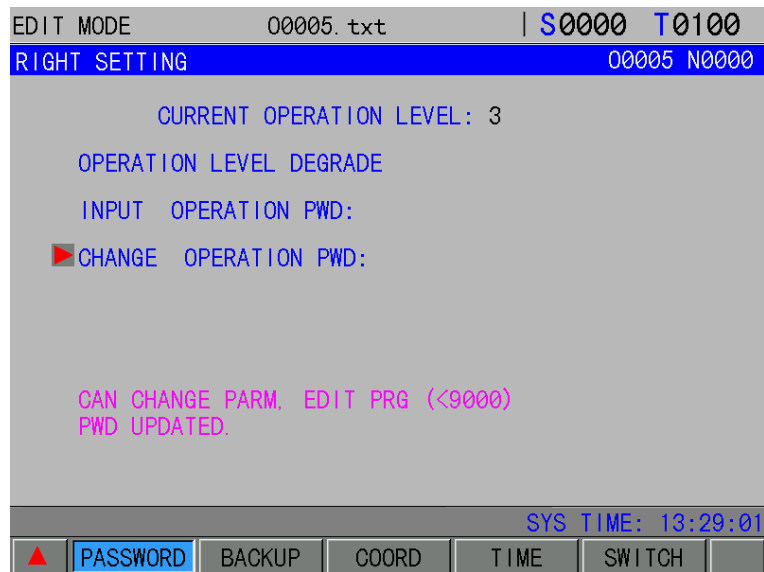
6. If the password entered twice is not the same, the CNC prompts "Change password verification does not match, please re-enter".

10.3.3 Operation Level Degradation

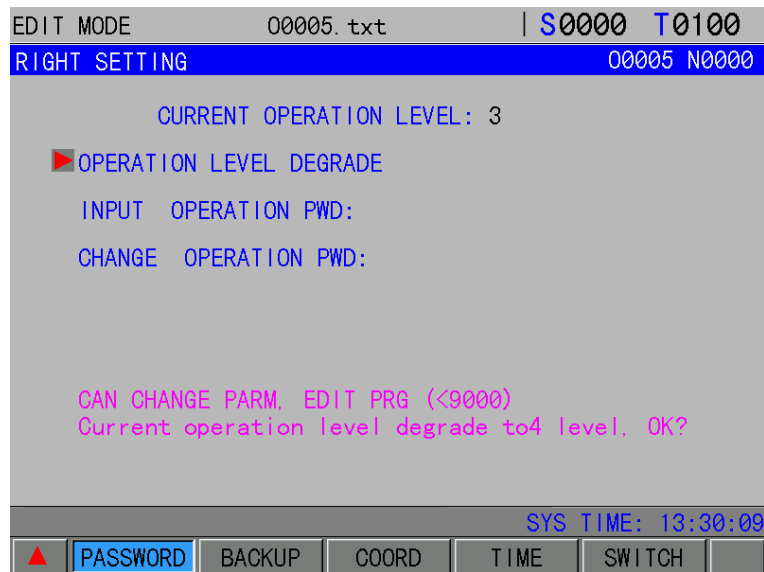
Operation level downgrade can facilitate the user to reduce from a higher level of operation level to a lower level of operation, the operation steps are as follows:

1. After entering the permission setting page, enter the password according to the method and steps described in Section 10.3.1;
2. Move the cursor to the "Operation Level Downgrade" row, if the current operation level

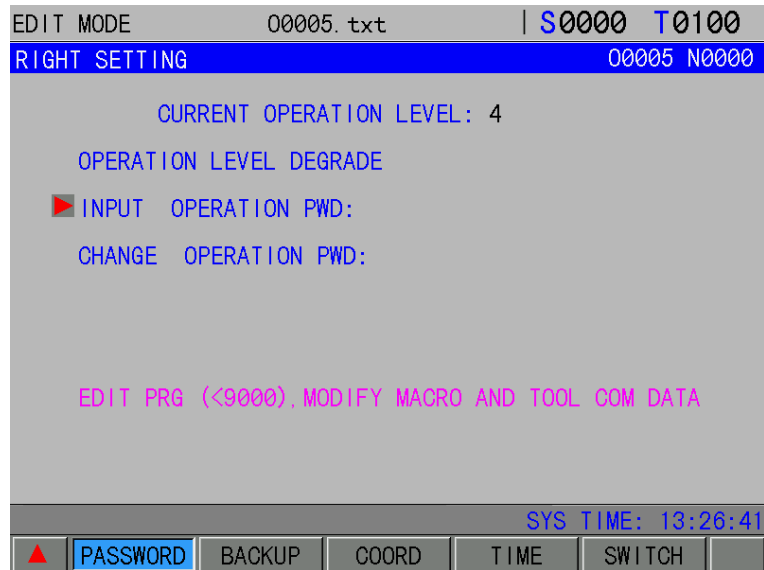
of the CNC is level 3, the display page is as follows:



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



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



Note: If the current operation permission has been level 5 , the operation level downgrade operation cannot be performed.

Chapter 11 USB flash drive operation function

11.1 File Directory Page

Press the  key to enter the program page in the non-editing state, press the  key to enter the [File Directory] interface, and insert the U disk as shown in figure:



The left side of the page displays the CNC disk directory information. The right side displays the USB disk directory information, if the U disk is not detected, the right display bar does not display the content.

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

Pressing the  cursor switches from the CNC to the USB drive, and pressing the  key or  key moves the cursor.

11.2 File Copying

Move the cursor over the file you want to copy and press the  key to copy it.

Chapter 12 Processing Examples

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

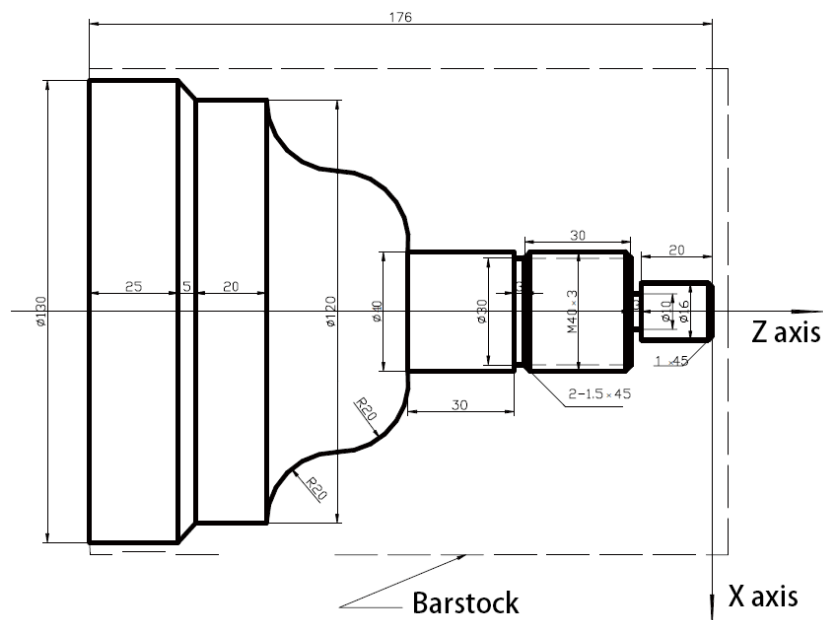


Figure 14-1

Machined 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 explanation of this manual, the coordinate system of the workpiece shown in Fig. 14-1 is established, and the editing procedure is as follows:

```

O 0 0 0 1 ;    //Part program name
N 0 0 0 0    G0 X150 Z50;    // Position the tool change to a safe position
N 0 0 0 5    M12;    //Clamp the 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;    // Close to the workpiece cut depth 1mm, retract 1mm
N 0 0 5 0    G71 U0.5 R0.5 F200;    // X axis reserves 0.5mm, Z axis 0.5mm headroom
N 0 0 5 5    G71 P0060 Q0150 U0.25 W0.5;    // Close to the end of the workpiece
N 0 0 6 0    G0 X16;    // Car Φ16 outer circle
N 0 0 7 0    G1 Z-23;    // Vehicle end faces
N 0 0 8 0    X39.98;    // Car Φ39.98 outer circle
N 0 0 9 0    W-33;    // Vehicle end faces
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;    // Taper
N 0 1 4 0    G1 X130 W-5;    // Car Φ130 outer circle

```

```

N 0 1 5 0    G1 W-25;  // The rough car is finished changing the tool point
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 bias
N 0 1 8 0    G70 P0060 Q0150;  // Fine car cycle
N 0 1 9 0    G0 X150 Z185;  // The rough car is finished changing the tool point
N 0 2 0 0    T0303;  // Change the No. 3 knife and perform the No. 3 cutter deviation
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;  // Returns
N 0 2 5 0    G0 X42 W30;  // Chamfering
N 0 2 6 0    G1 X40;  // Give way to the groove knife width
N 0 2 6 2    G1 X37 W1.5;  // Chamfering
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;  // Chamfering
N 0 2 8 0    G0 X150 Z50;  // Returns to the tool change point
N 0 2 9 0    T0404 S100;  // Change the No. 4 knife and place the spindle at 100 speeds
N 0 3 0 0    G0 X42 Z-20;  // Close to the workpiece
N 0 3 1 0    G92 X39 W-34 F3;  // Cut thread cycle
N 0 3 2 0    X38;  // Feed 1mm cut second knife
N 0 3 2 0    X37;  // Feed 1mm and cut the third knife
N 0 3 3 0    X36.4;  // Feed 0.6mm cut fourth knife
N 0 3 3 2    X36;  // Feed 0.4mm cut fifth knife
N 0 3 4 0    G0 X150 Z50;  // Return the tool point
N 0 3 5 0    T0100 U0 W0;  // Switch back to knife No. 1 and execute the knife offset value
N 0 3 6 0    M5;  // Off spindle
N 0 3 7 0    M9;  // Turn off coolant
N 0 3 8 0    M13;  // Release the chuck
N 0 3 9 0    M30;  // End of program

```

12.2 Input to the Program

12.2.1 Viewing Existing Programs

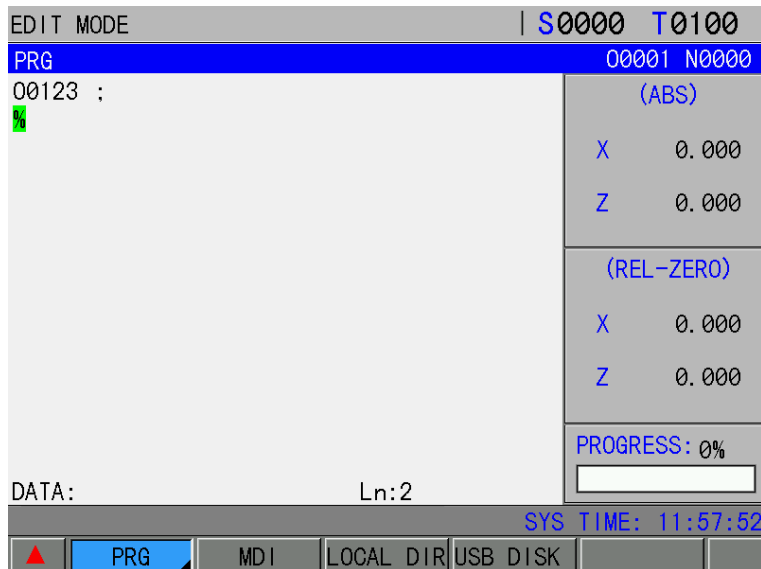
Under The non-editing operation mode, press the  key to enter the program interface, press the  or  key to select the program directory page, which is displayed as follows:



On the previous page, you can view the program names of the programs stored in the CNC to prepare for the confirmation of the new program names.

12.2.2 Establish new procedures

In the edit operation mode, press the  key to enter the program content page, which is displayed as follows:



Press the Address key , select a program name (such as 0001) that is not in the

program directory page, press the number key , , ,  press

 the key (or  key or  key) to create a new program, The page appears as

follows:



Follow the program written above to enter it character by character to complete the editing of the program.

12.3 Program Validation

12.3.1 Graphics Settings

1. Press the  key to enter the graphical interface, and the page is displayed as follows:



12.3.2 Validation of Programs

Press  or  key to enter the graphic display page, press the  key to enter the automatic operation mode, press ,  key to make the auxiliary function in the indicator area lock the lamp , the machine lock lamp  and the empty operation indicator light, enter the auxiliary function lock, machine lock and empty running state.

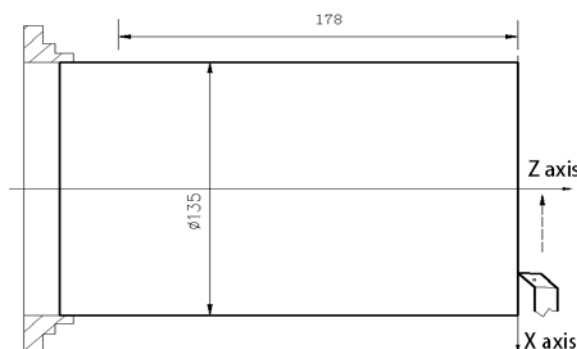
Press  start to make a drawing, press the  automatic operation program, you can display the trajectory of the tool movement, verify the correctness of the program, and the operation is completed.

If the displayed program trajectory is wrong, analyze the error in the program and modify the part program, and check it again according to the above method until it is correct. In the

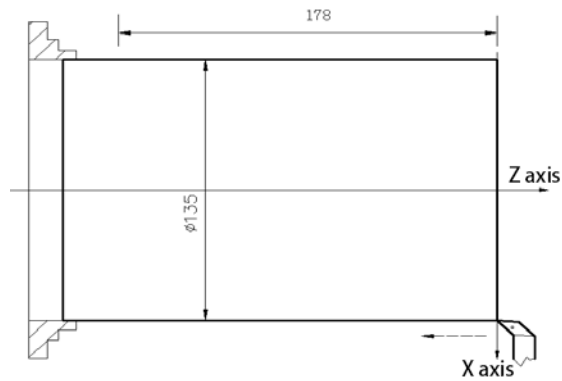
graphical display interface, press the  key in the panel to start drawing, press the  key to stop drawing, and press the  key to clear the graphic.

12.4 Pairing knives and running

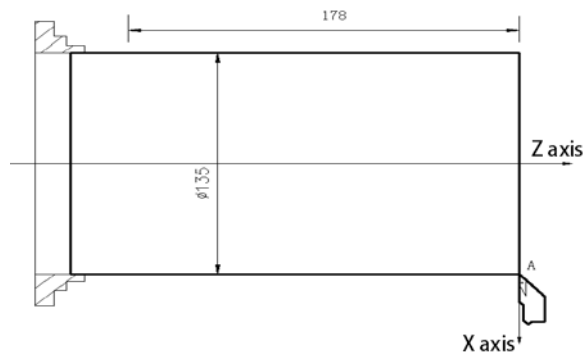
1. Move the tool to a safe position, perform T0100 U0 W0 on the input operation mode and program status page, and cancel the tool bias;
2. Move the tool and make the tool cut along the end face of the workpiece;



3. Release the tool along the X axis when the Z axis is not moving, and stop the rotation of the spindle, execute G50 Z0 on the input operation mode and program status page, and set the coordinates of the Z axis;
4. Switch to the tool bias page and enter Z0 in the 001 offset;
5. Move the tool and make the tool cut along the outer circle of the workpiece;

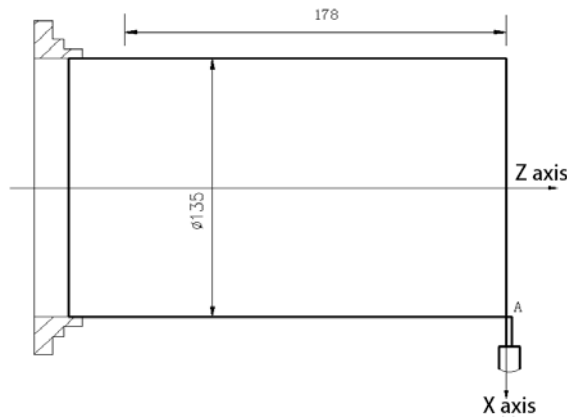


6. Release the tool along the Z axis without moving the X axis, and stop the spindle rotation to measure the outer circle size of the workpiece (such as the measured value is 135mm) ;
7. Execute G50 X135 on the input operation mode and program status page, and set the coordinates of the X axis;
8. Switch to the tool offset page and enter X135 in 001 bias;
9. Move the tool to a safe position, and press the tool change button in the manual operation mode to execute the second knife;
10. Start the spindle and move the tool to the knife point, as shown in figure A below;



11. Switch to the tool offset page, move the cursor to the 002 offset, and enter X135, Z0;
12. Move the tool to a safe position, and press the tool change button in the manual operation mode to execute the third knife;
13. Start the spindle and move the tool to the opposite point, as shown in point A of the

following figure;

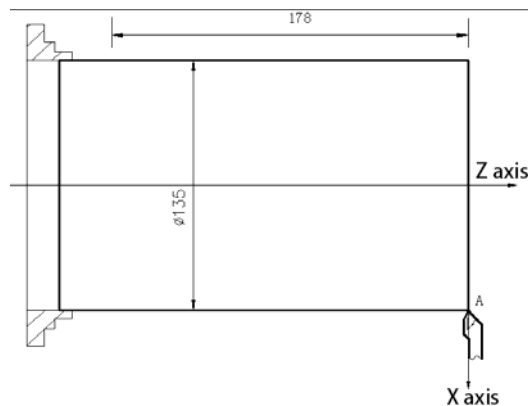


14. Switch to the tool offset page, move the cursor to the 003 offset, and enter X135 and Z0;

15. Move the tool to a safe position, and press the tool change button in the manual

operation mode to execute the fourth knife;

16. Move the tool to the opposite point, point A as shown below;



17. Switch to the tool bias page, move the cursor to the 004 offset, and enter X135, Z0;

18. After the knife is completed, remove the knife to a safe position;

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

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

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

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