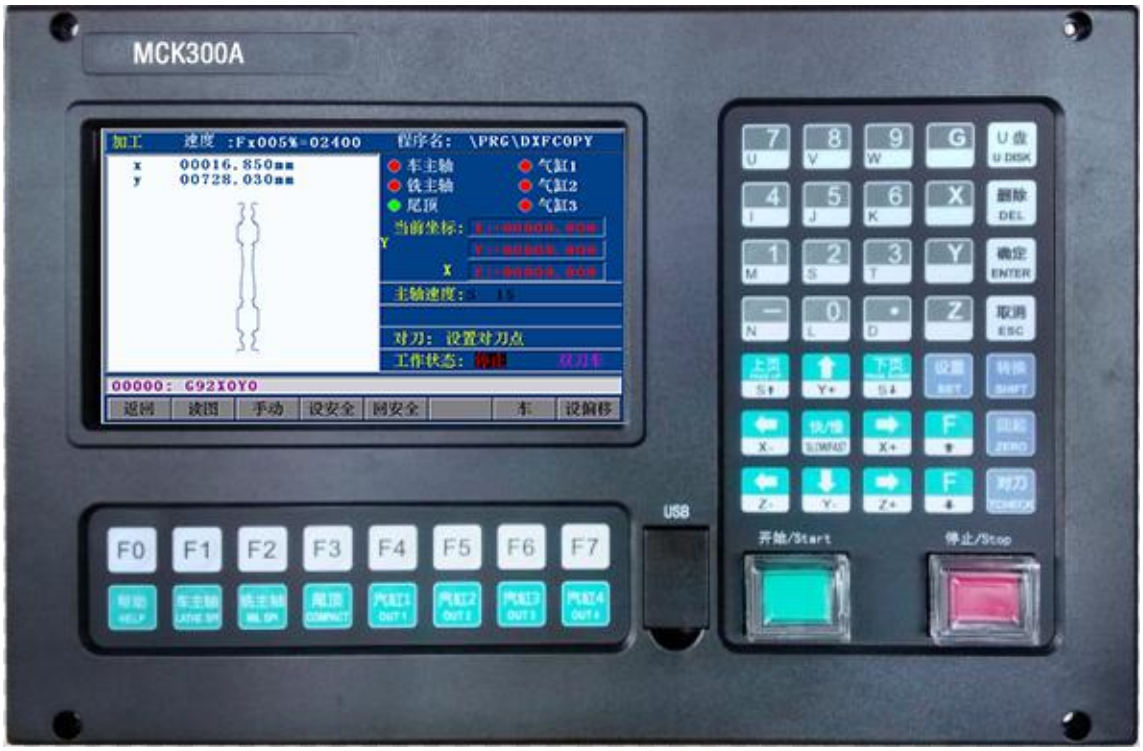
Carpenter Lathe/Miller Controller
MCK300A

Chapter I General Introduction

1.Product

Drawing



3. System Features

1) High-speed ARM9 microprocessor and hyper-scale FPGA; multi-task control and hardware interpolation technologies, ensuring stable motion and fast program running and interface refreshing.
2) 7”nature color display screen, user-friendly interface and easy operation.
3) Optimized product structure, together with opto-isolator control, provides high anti-interference capacity.
4) 256M memory satisfies requirements of large and complicated programs.
5) 3 stepping/servo motors controlled simultaneously, 3-axis G code supported。
6) System software includes the cutting programs for carpenter’s lathing and column twist-milling.
7) Workpiece can be shaped with single or dual tool on either one-time or multiple-times basis.
8) Compatible with machining files created thorugh scanning of laser workpiece’s profile.
9) Compatible with manual programming of G code instructions.
10）Break-point memory for the time of power failure

11) Tool offset and machine tool's backlash offset available.

12) Graphics and real-time machining track displayed.

13) System diagnosis helps the user to quickly eliminate failures.

14) Files that can be directly imported include: files with manually-programmed G instructions andUSB-flash-disk G codes, dxf files, files created by manual programming or laser scanning and Chinesized DBF files.

15) Supports Chinese file name and alteration of file name.

16) System can be upgraded through USB flash disk, and system files can be upgraded and controlled and by PC.

17) Backup and restoration of key files and system parameters.

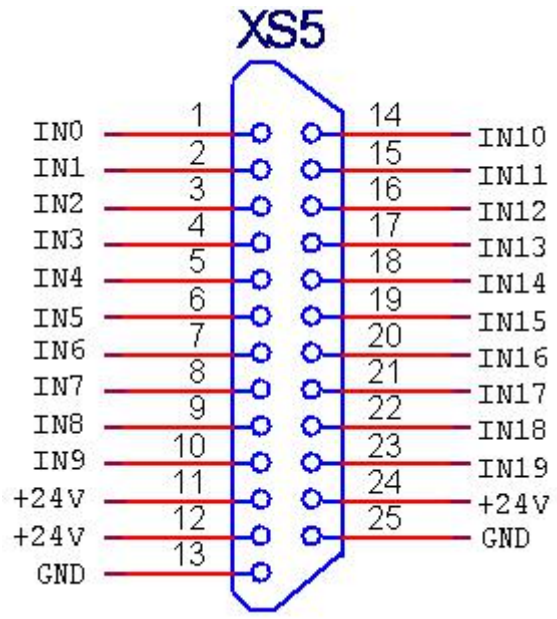
18) High/low level effective for input/output connection mode can be set as "high or low level effective" .

19) External wireless remote control.

4. Specifications

Operating Environment		
Temperature	Operating	-5℃～50℃
	Storage	-30℃～70℃
RH	Operating	20%～90%（℃）
	Storage	10%～95%（℃）
Barometric pressure		86Kpa～106Kpa
Power supply		DC 24V 50/60Hz
Power consumption		<10W

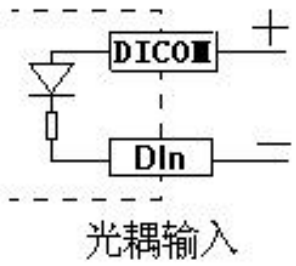
2.2.2 Input Control Interface—25-Pin Plug Seat (male)



Line No.	Definition	Details
1	IN0	Scan input
2	IN1	Axis A limit
3	IN2	Axis Z limit (as Z negative home position)
4	IN3	External pause
5	IN4	External manual X positive
6	IN5	External manual Y positive
7	IN6	Spindle ON
8	IN7	External manual Z positive
9	IN8	Axis X positive limit（>X+）
10	IN9	Axis X negative limit (X-<)（as X negative home position）
11	+24V	Controller inside 24V+ (please do not make external connection to any circuit)

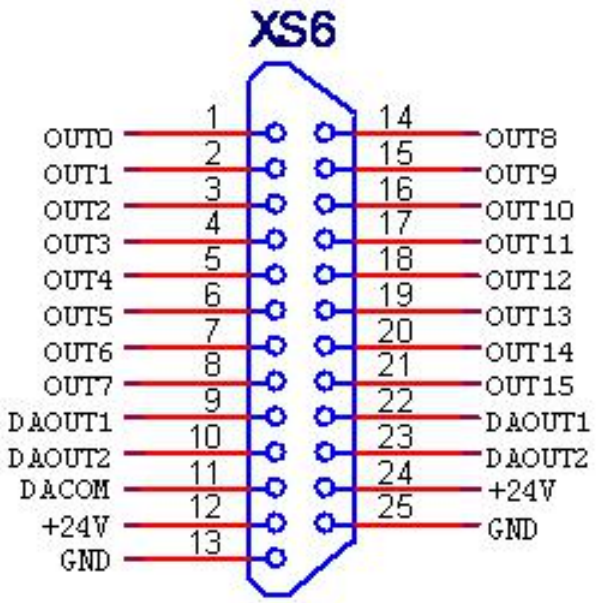
12	+24V	Controller inside 24V+ (please do not make external connection to any circuit)
13	GND	Controller inside 24V grounding
14	IN10	Axis Y positive limit（>Y+）
15	IN11	Axis Y negative limit Y (Y-<) (as Y negative home position)
16	IN12	Rear ejection
17	IN13	External start button (STA)
18	IN14	External remote control X- positive control (X-)
19	IN15	External remote control Y- positive control (Y-)
20	IN16	Load support
21	IN17	External manual Z negative
22	IN18	Manual fast/slow
23	IN19	
24	+24V	Controller inside 24V+ (please do not make external connection to any circuit)
25	GND	0V

Connection for input signals:



For DICOM terminals, the positive end has been connected, and the input signal should beconnected to the corresponding DIN terminal. In default settings, the system receives low level as effective.

2.2.3. Output control interface—25-pin plug seat (female)

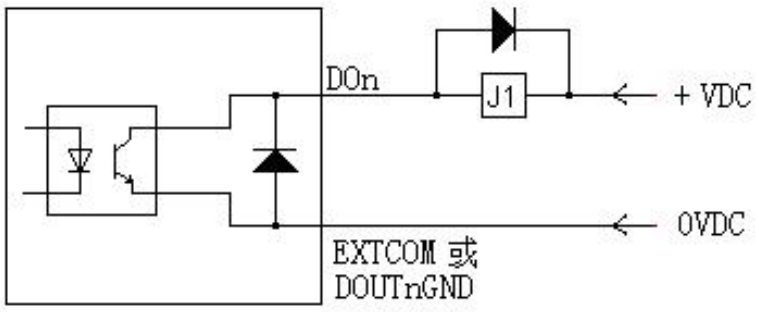


Line No.	Definition	Details
1	OUT0	Lathe spindle control (M10 on, M11 off)
2	OUT1	Miller spindle control(M12 on, M13 off)
3	OUT2	Rear ejection control (M12 on, M153 off)
4	OUT3	Output 1（M16 on， M17off）
5	OUT4	Output 2 （M18 on， M19 off）
6	OUT5	Output 3 (M20 on， M21 off）
7	OUT6	Spindle speed 1 （M22 on， M23 off）
8		
9	NC	Idle
10	NC	Idle
11	NC	Idle
12	+24V	Controller inside 24V+ (please do not make external connection to any circuit)
13	GND	Controller inside 24V grounding
14	OUT8	Spindle speed 2
15	OUT9	Spindle speed 3

16		
17	OUT11	Spindle speed 4
18		
19		
20		
21	OUT15	
22	NC	Idle
23	NC	Idle
24	+24V	Controller inside 24V+ (please do not make external connection to any circuit)
25	GND	0V

Connection for output signals:

As it is open-collector output, if the external load is of inductive, like the relay, the follow-current diode must be connected. See the figure below.



For EXTCOM terminals, the negative end has been connected, and the output signal should be connected to the corresponding DON terminal.

It is suggested the voltage be less than 24V, and even worst cases, it shall not be over 30V. Do not make the cathode and anode reversely connected. There should be no short circuit for the load. Otherwise, the module might be damaged.

2.2.7 Remote control

To facilitate the user's operation, MCK300A is configured with the remote control.

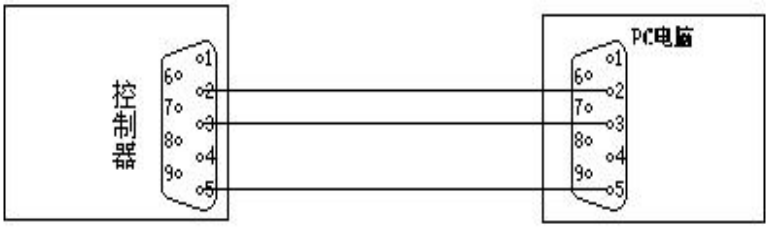
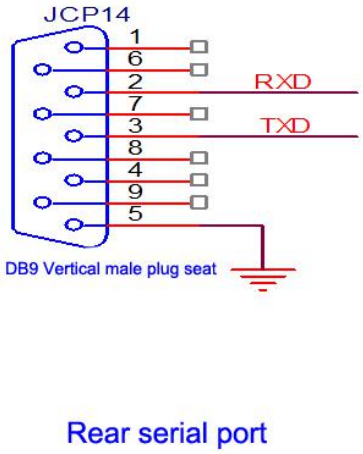


Fig. Remote-control Signal Receiver

Operating instructions:

1. Connect the remote-control signal receiver to XS9 port, and make sure the power indicator is illuminated.
2. Load four AA batteries into the remote control and make make sure the power indicator is illuminated.
3. Set the same frequency value on both the remote control and the signal receiver.

Definition of Pins



Rear serial port

Line No.	Description	Details
1		Empty pin
2	TXD	Serial port data receiving
3	RXD	Serial port data transmitting
4		Empty pin
5	GND	Signal grounding
6	Empty pin	
7	+5V	5V power output (for receiver)
8	+5V	5V power output (for receiver)
9		

4.3 Adjustment of Pulse Equivalent

Please refer to Chapter III—"Quick Start and Adjustment—Step 4".

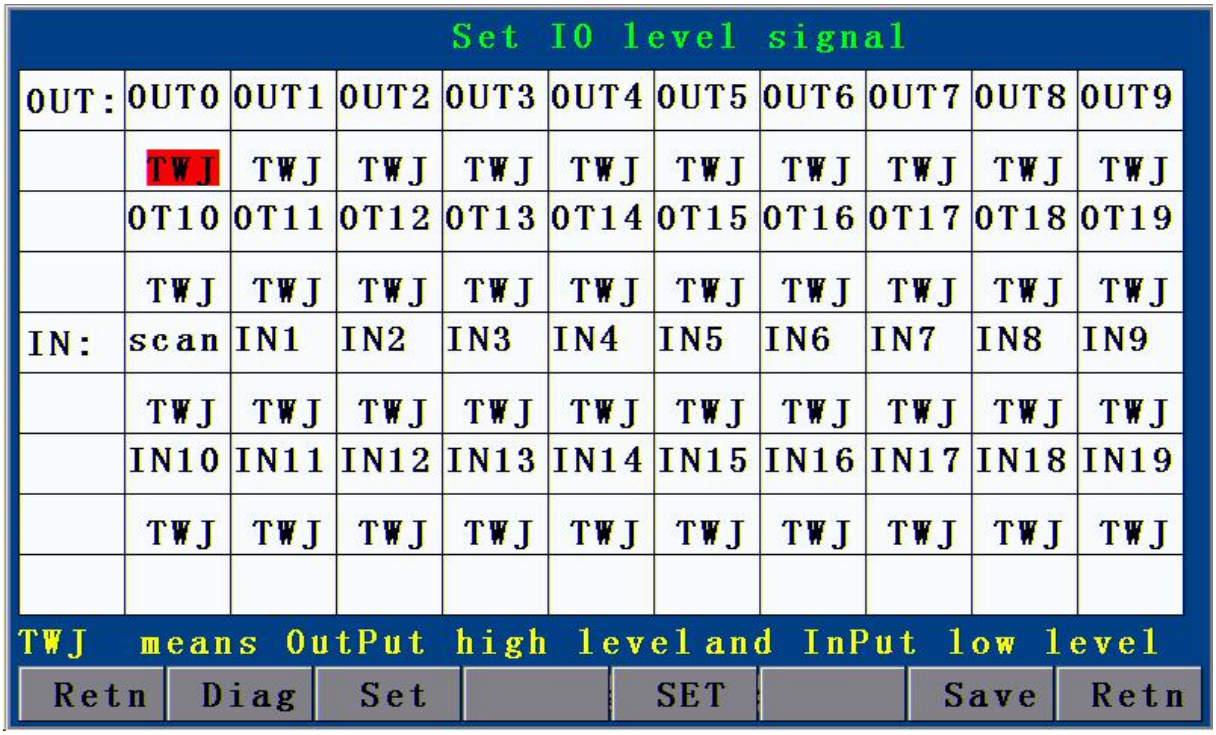
Chapter III Quick Start and Adjustment

Step 1: Access to Startup Interface

After the controller is properly connected with reference to the “Section 2—System Input/Output Interface” in “Chapter V—Hardware”, you can connect the system to the power source. When the system is powered on, the startup interface will appear on the screen. See the figure below.

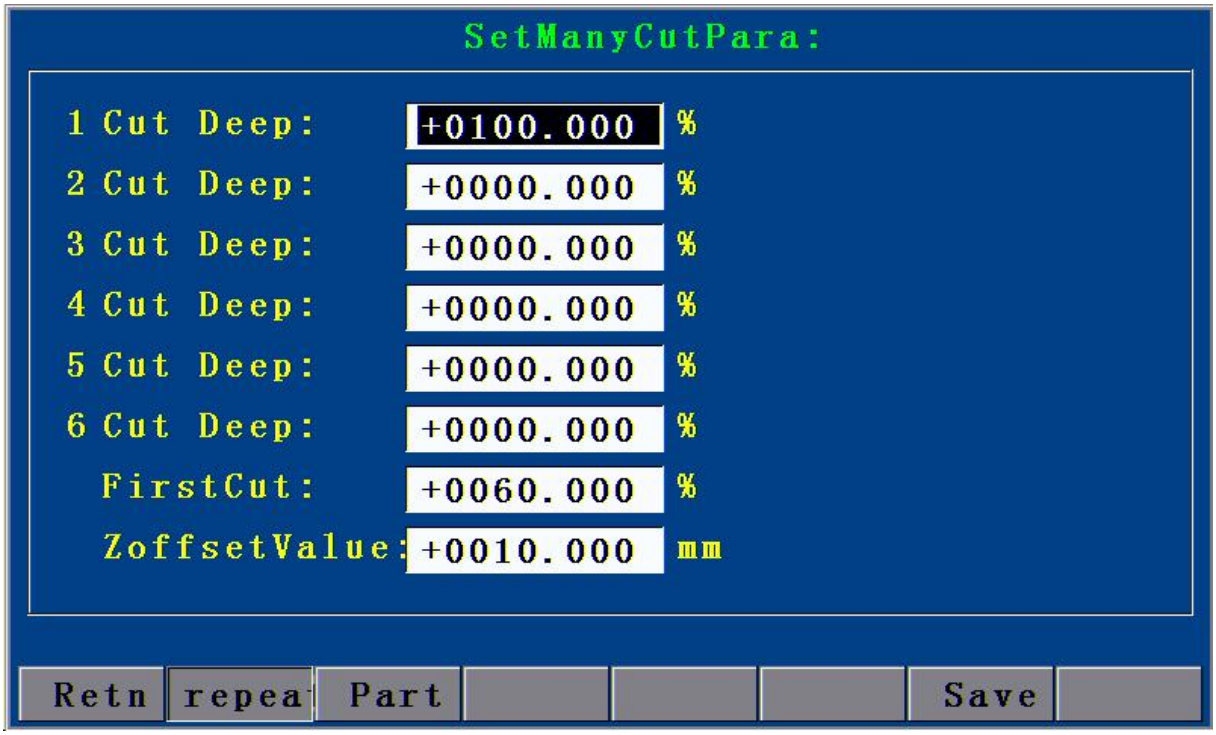
Step 2: Access to Diagnosis Interface

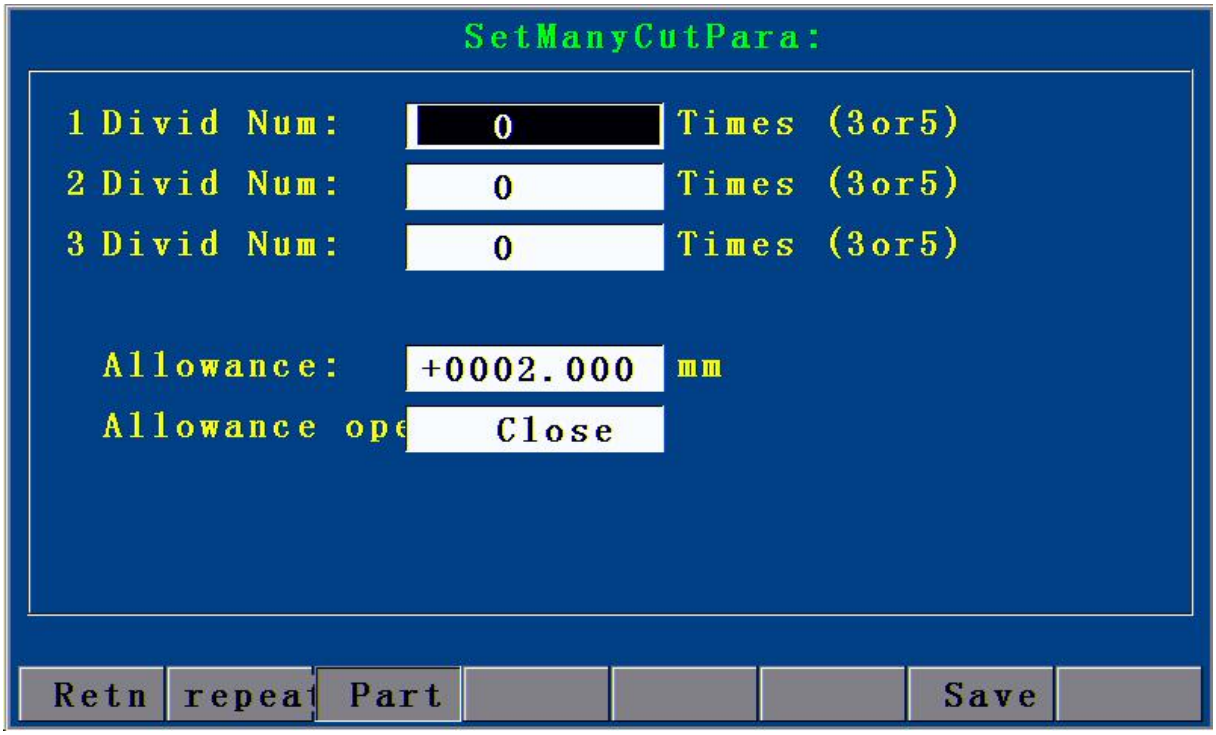
In the main interface, press 【F5】to make diagnosis options, and press 【F1】to access to the I/O (input/output) diagnosis interface, where you can check whether the wiring for I/O and motor is correct. If an input signal is available, the red circle underneath will turn green in the corresponding IN field. At the time, press “→←↑↓” to move the blue circle of OUT options to the designated place of M output. Then press the button Enter, and you can check whether the corresponding output is effective.



Step 3: Set the parameter of tool advance

After the system checks the I/O signals, you can press 【F0】to exit the main interface. Then press 【F4】to access to the interface for setting the parameters, and press 【F1】to set the parameters for tool advance of multiple times.





Cut depth 1-6: When there is only one time of cutting, you can set cut depth 1 as 100%. If multiple times of advance are needed, you should set the cut depth for each time according to the proportion between the absolute cutting depth and the final cutting depth.

Rough tool proportion: It refers to the proportion between the cutting amount of rough tool and the total cutting amount of the time when double tool is engaged.

Z postposition offset: It refers to the distance from axis Z tool to its cutting point when dual tool is engaged.

Step 4: Set the Precision

In the main interface, press 【F4】 to access to the parameter-setting page, then press 【F5】to set the precision. You can roughly set the pulse equivalent according to the segmentation, lead screw's length per round and reduction ratio. Normally, when the system is directly connected to the lead screw, the adjustment value of pulse equivalent is set as 0.001.

Calculation formula: Pulse equivalent=Lead screw's length per round/segmentation/reduction ratio

Precision adjustment: Setting pulse equivalent

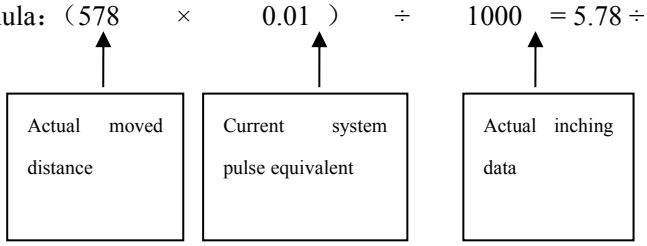
1) Press【F1】and then【F2】to access to the manual interface. After that, press【F1】, the worlds "inching increment: L=+00000.00"will appear on the screen. At the time, enter the length you desire.Here we

suppose the inching unit as 1000MM.

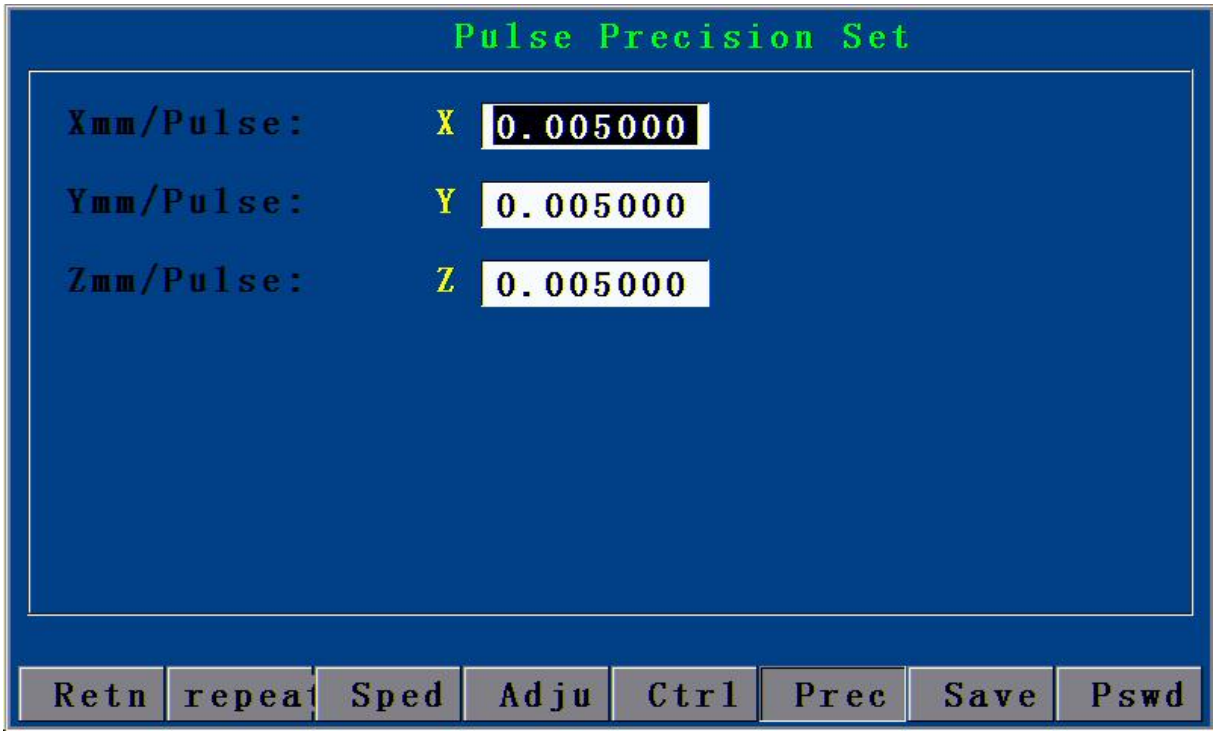
2) Press any of the following:【X+】,【X-】,【Z-】,【Y+】,【Y-】and【Z-】to start inching. Here we suppose actual moved distance is 578MM,

The calculation formula is: （578 × 0.01 ） ÷ 1000 = 5.78 ÷ 1000 = 0.005780。

The calculation formula：（578 × 0.01 ） ÷ 1000 = 5.78 ÷ 1000 = 0.005780。

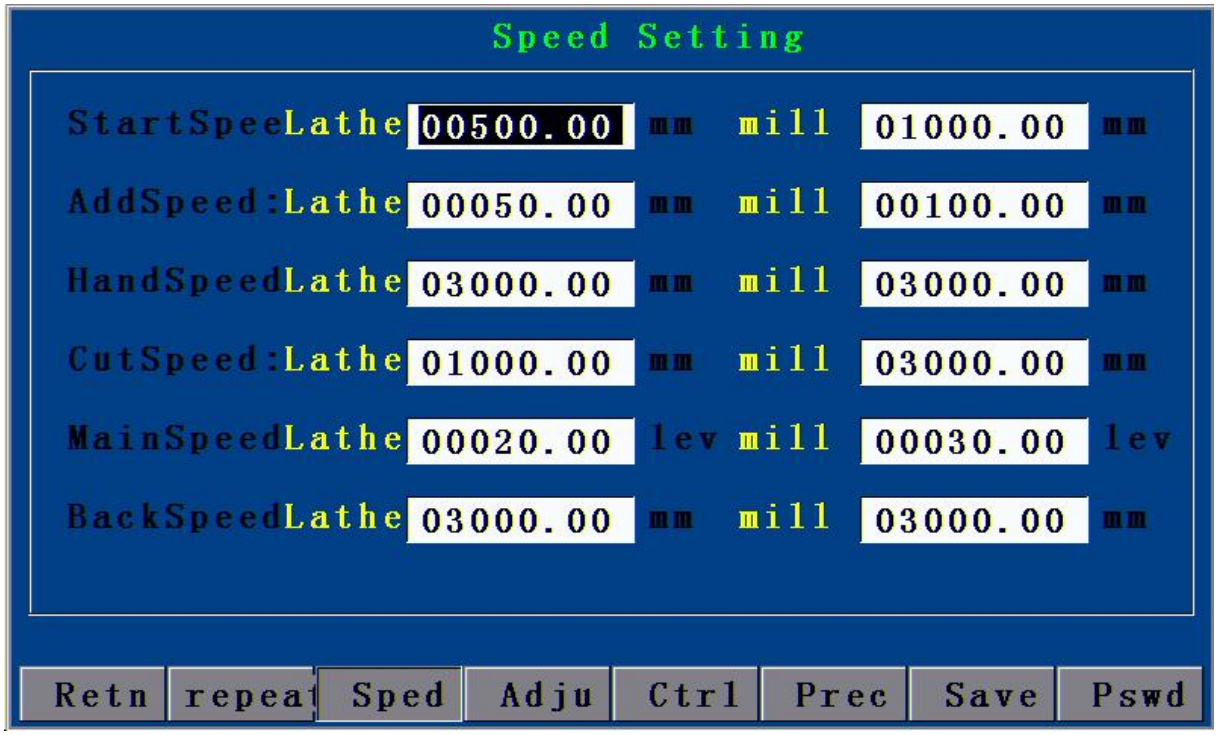


3) Return to the main interface. Press【F4】and then【F5】 to access to the interface for precision setting. Respectively enter the number “0.005780” as the pulse equivalent coefficient of axis X, Y and Z.



Step 5: Set the Speed

Press 【F4】in the main interface, then press 【F2】 to set the speed. When you set the speed, you can refer to the table below indicating the relations between the machining speed and startup speed, and the relations between the pulse equivalent and acceleration.

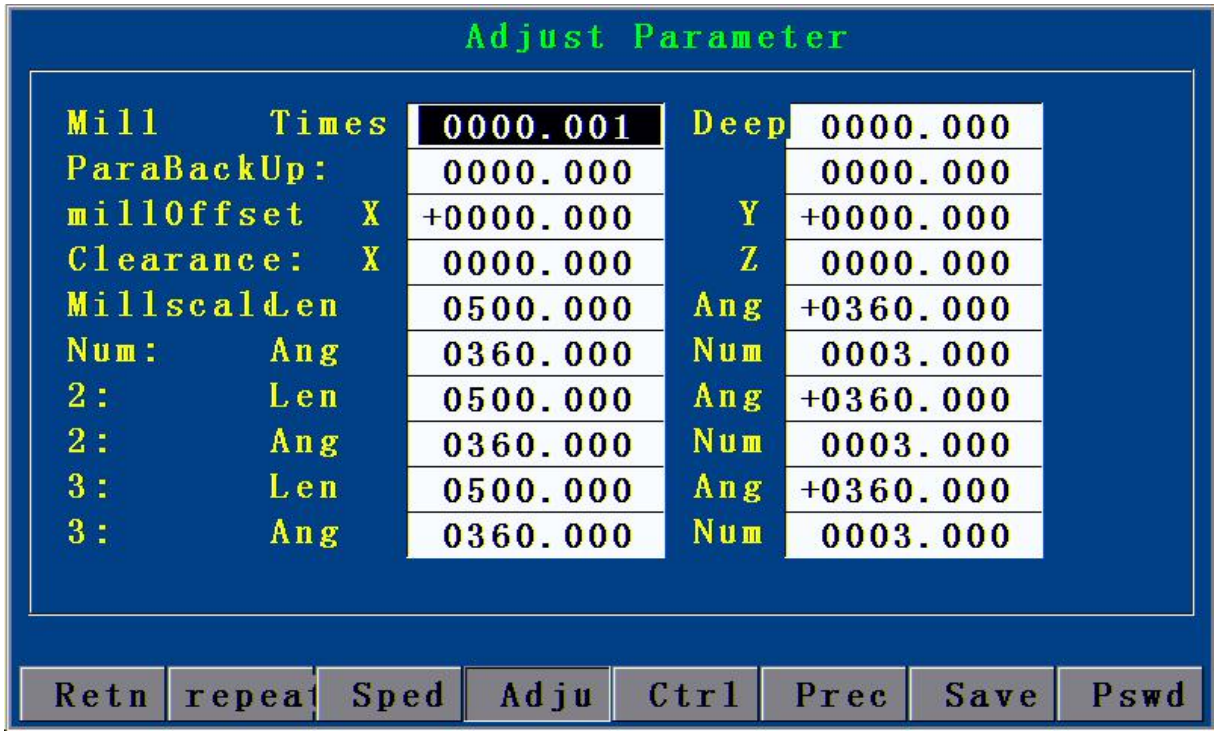


Machining Speed (mm/min)	Start Speed(mm/min)	Pulse Equivalent	Acceleration(mm/t)
100-200	100	0.001000-0.002000	10(In practice, adjustment is needed)
200-500	200	0.002000-0.003000	20
500-700	300	0.003000-0.004000	30
700-800	500	0.004000-0.005000	40
800-1000	600	0.005000-0.006000	50
1000-1300	700	0.006000-0.007000	60
1600-2000	1000	0.007000-0.008000	70
2000-3000	1500	0.008000-0.009000	80
3000-4000	2000	0.009000-0.010000	90
		0.010000-0.011000	100

		0.011000-0.012000	110
		0.012000-0.013000	120

Step 6: Set the Adjustment Parameters

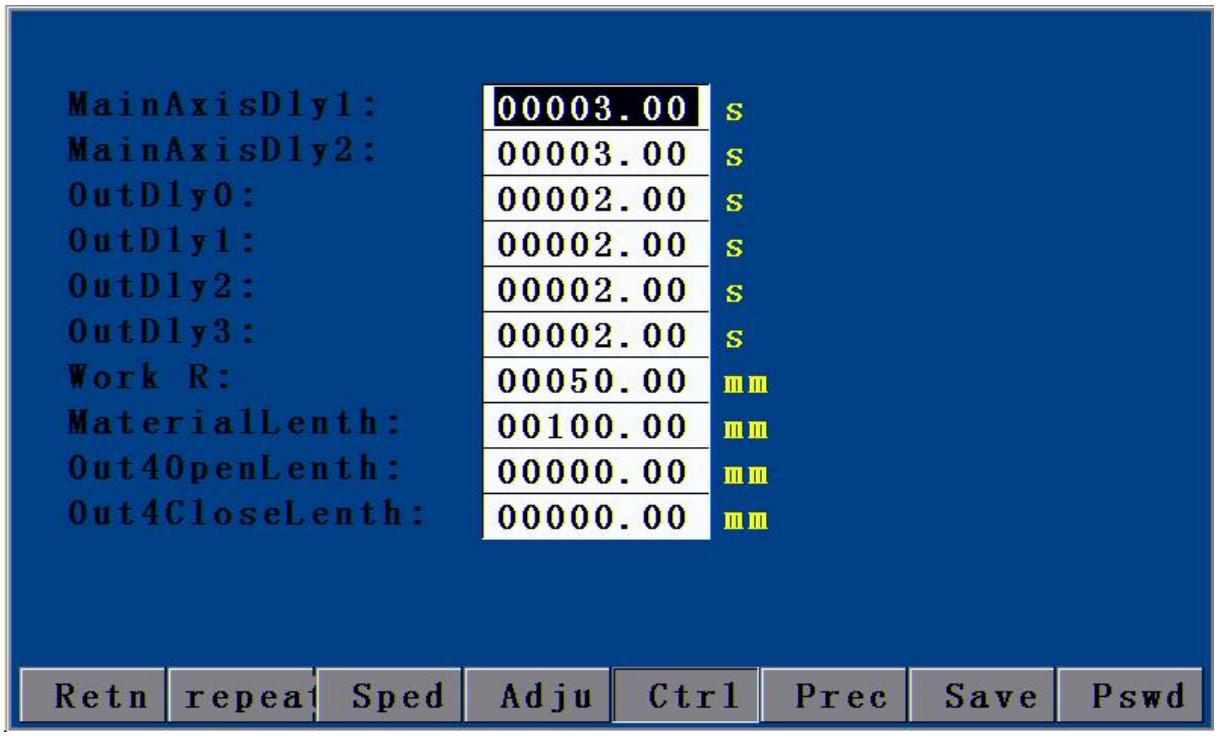
Press 【F4】to access to the interface for setting parameter, and then press 【F3】to set the adjustment parameters.



Parameter	Details	Unit
Soft limit	It is the software limit. When the current value is higher than the set value, the system will automatically stop.	mm
Milling cutter offset	The distance between lathe's cutter-aligning point and the miller's cutter-aligning point.	None
Backlash	The clearance offset for the time when axis X and Y change the direction reversely.	mm
Twist proportion	It refers to the relation between the total length run by axis Y and the revolution angle.	mm
Twist number	It refers to the total angles and numbers to be machined at the time of scale division.	

Step 7: Set the Control Parameters

In the main interface, press 【F4】to access to the interface for setting parameter, and then press 【F4】to set the control parameters.



Lathe spindle delay: Set the delay time between startof lathe spindle sand start of next program.

Miller spindle delay: Set the delay time between start of miller spindle sand start of next program.

Rear ejection delay: Set the delay time between start of rear ejection and start of next program.

Cylinder 1 delay: Set the delay time between start of cylinder 1 and start of next program.

Cylinder 2 delay: Set the delay time between start of cylinder 2 and start of next program.

Cylinder 3 delay: Set the delay time between start of cylinder 3 and start of next program.

Timber diameter: Set the interval between two tracks when dual tool is displayed.

Feed distance: Set the distance axis Y moves at the time of automatic feeding.

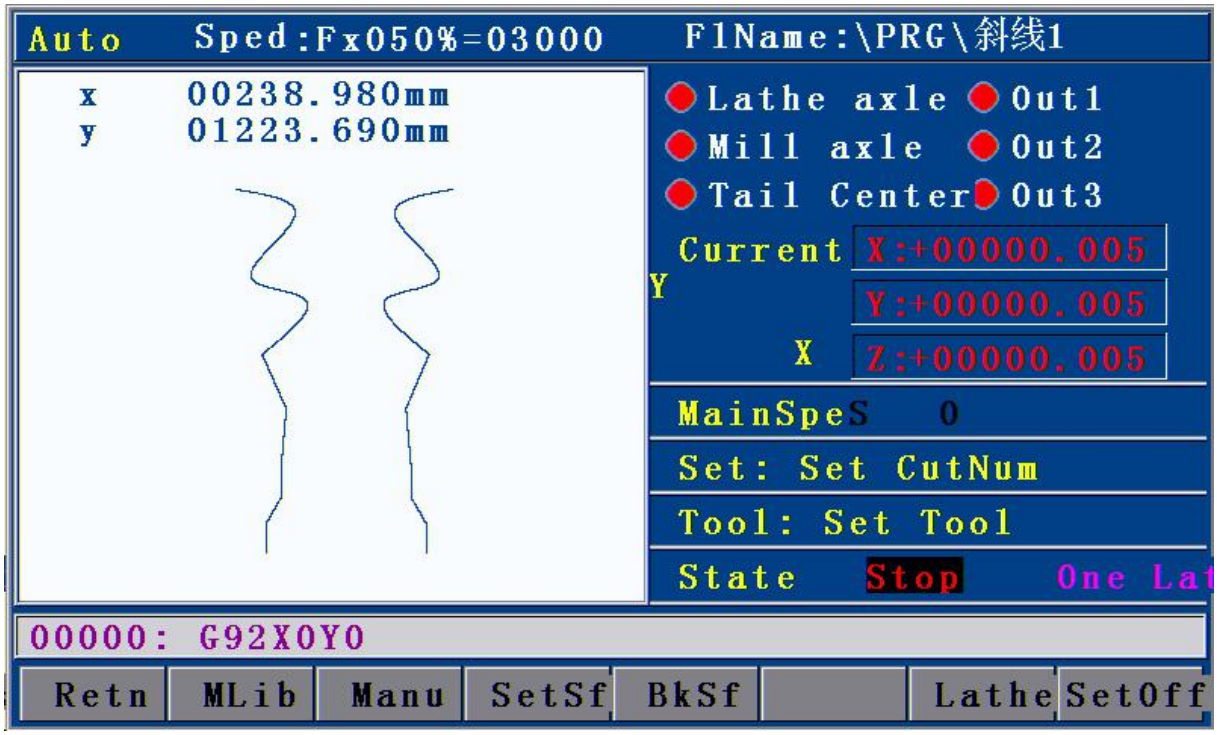
Back-tool-on distance: Set the distance from the starting point of the back tool in the drawing to the home position when a back tool is engaged.

Back-tool-off distance: Set the distance from the closing point of the back tool in the drawing to the home position when a back tool is engaged.

Step 8: Access to the Machining Interface

Press 【F1】in the main interface to access to the machining interface.

Press 【SHIFT】to switch over the system modes. Here three modes are available: single tool machining, dual tool machining and milling



Step 9: Read the Drawing

In the main interface, press 【F1】(machine) , and then press【F1】(draw reading) to access to the interface where you can edit or import data. See the figure below.

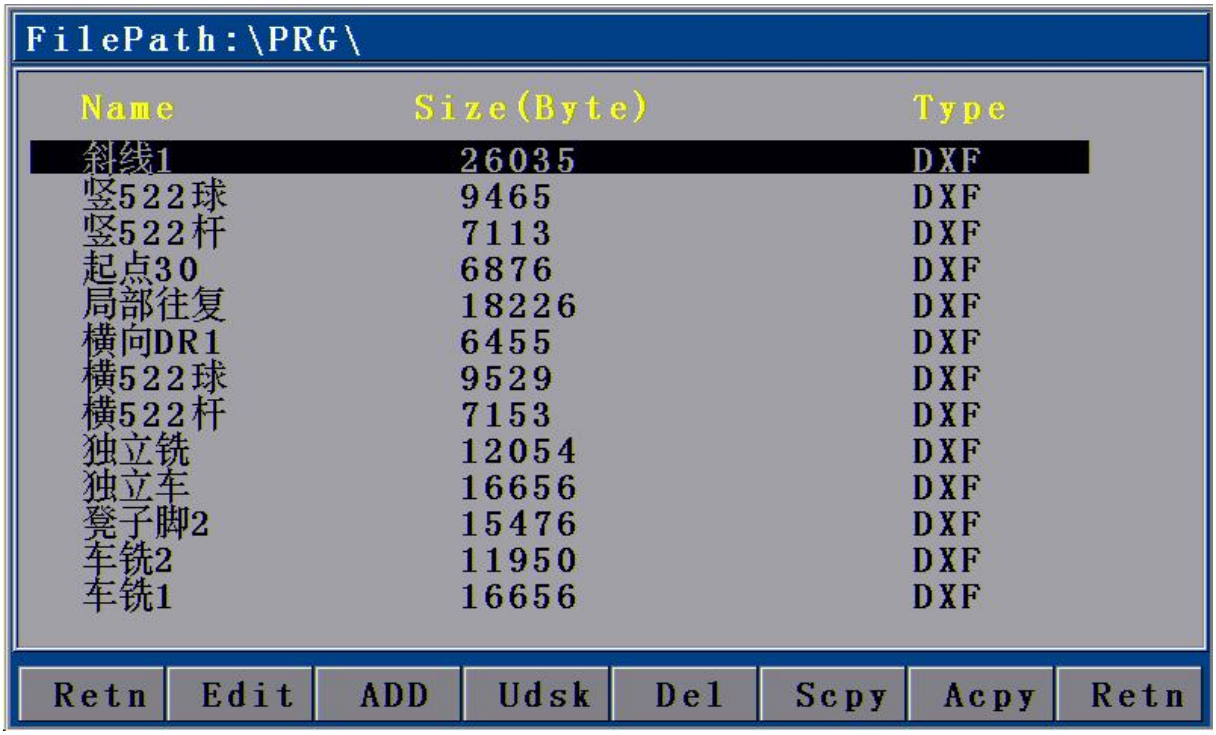


Fig2.2: File Operation Interface

1) Page scroll-up/down: Press 【Pagup / S↑】or【Pagdn / S↓】to scroll up or down the pages.
2) 【Y+】【Y-】: Move the cursor upward or downward.
3) Press 【F2】to import the machining files into the system, and the system will return the machining interface automatically.

In the machining interface, press 【F5】(reverse direction key) to change the order of program-based machining.

In the machining interface, press 【F7】(times key) to execute the multi-time machining parameters as set, and the drawing for multi-time machining will be re-created.

After the correct drawing is created, you can align the cutter and set the safety point.

Step 10: Align the Cutter and Set the Safety Point

Rough cutter alignment: First fasten the standard workpiece to be machined, then successively press "→←↑↓" to align the cutter nose of X/Z to the external edge of the maximum outline of the workpiece and move axis Y to the machining start point. After that, press the cutter-alignment button, and the system will set the front edge position (i.e. the machining start position) as 0, and

return to the mechanical home position. This is the rough cutter alignment. (In aligning, you can press “G fast/slow” to change the manual operation speed)

Precision cutter alignment: Record the max. external diameter R for the start point on the workpiece with standard drawing machining, start spinning the spindle, move manually and slowly the cutter nose on advance axis Y for “advance y”, till a measurable outline forms by cutting. Then move manually and slowly the cutter nose on advance axis Z for “advance z”, till a measurable outline forms by cutting. At the time, manually move to axis X direction for a length x, and turn off the spindle. Then accurately measure the diameters(Ry and Rz) of the external circles cut by the two cutters, and measure the length from Y cutter outline to long axis X on the end face. Press the key 【Align】, and the system will pop up the prompt box for tool alignment. After the measured X length, diameters of X and Y and nominal maximum external diameter of uncut workpiece are entered, press 【Enter】, and the system will automatically align the cutter and return to the mechanical home position. After the tool alignment, you can press 【Start】to start machining.

Set the safety point: After the tool is aligned, you can set a safety-stop position for the completion of machining, so that the system can return to this position each time when the process is finished.

Miller cutter alignment: (When continuousturning and milling are turned on) move axis X and Y, align the center point of miller cutter revolution to the home position of lathe tool, and set the offset.

(When continuous turning and milling are turned off) move axis X and Y, align the center point of miller cutter revolution to the home position of lathe tool, press Align and confirm it.

Once the aforesaid settings are finished, you can press【Start】 to start machining.

Chapter IV Operating Instructions

With MCK300A carpenter-lathe/miller controller, you can process the timber by controlling the lathe and mill cutters on the machine tool. In the system, operations are prompted by menus in a step-by-step manner. Once you access to a menu interface, you can press from【F1】 to 【F6】 to allow the system to perform the corresponding function, and press from 【 F0】to【 F7 】to return to the previous menu.

After the system is energized, access to the main menu interface where the welcoming speech from the company is displayed, as seen in the figure below:

Fig.3: Main Menu Interface

Press from 【F1】to【F6】to access to different function interface.

Press 【F1】 to access to the main machining interface, where the follwing items are available: “draw reading”, “manual”, “reset”, “clear”, “reverse”, “restore”, and so on.

Press 【F2】to access to the interface of instruction programming.

Press 【F3】to access to the interface of laser scanning.

Press 【F4】to access to the interface of parameter setup and adjustment.

Press 【F5】to access to the interface for diagnosis, version inquiry and user-defined output

port.

Press 【F6】to access to the interface for parameter restoration and upgrading of apps.

Press 【F5】to access to the interface of parameter setup.

1. Machining

In the main interface, press 【F1】(machine) to access to the interface of auto machining, as seen in the figure below:

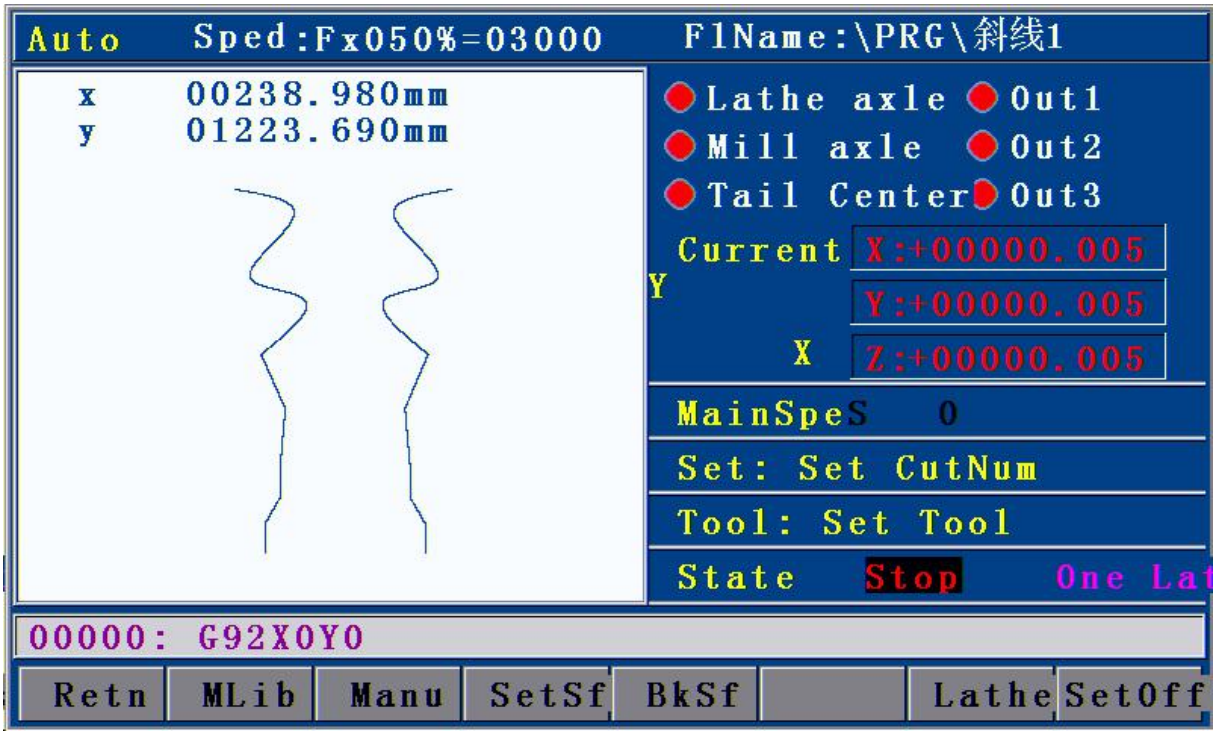


Fig. 2.1

Details of in the interface of Fig. 3.1①:

① The speed percentage can be adjusted through 【F↑】 and【F↓】;

② The current motion speed;

③ The machining file name;

④ The ON/OFF state of the solenoid valve;

⑤ The real-time coordinate value that axis X, Y and Z move;

⑥ States of some modes at the current time;

⑦ The machine model and the current operating state of the system;

⑧ Code segment of program running for the current machining;

⑨ Follow-up displaying of the current machining drawing.

Operating instructions:

Press 【F1】 (Draw Reading) to access to such items as “draw reading”, “manual”, “reset”, “clear”, “reverse”, “restore”, and so on.

Press 【F2】(Manual) to access to the manual-operation interface.

Press 【F4】(Home Position) to set the machine tool to the mechanical home position.

Press 【F5】(Reverse Direction) to switch over the left and right sequences for programmed machining.

Press 【F6】(lathing/milling) to exchange the tool point after the lathing and milling states are exchanged.

Press 【F7】(Offset) to set the current position as the offset between lathe tool and miller in continuous lathing/milling mode.

Press 【Set】on the panel, you can enter the reciprocal times and allowance for finish in single-tool mode, and enter the proportion of two tool advances and dual-tool offset in dual-tool mode. Press the numerical keys for entering the value and press 【Y+】 and【Y-】to switch over the items to be entered.

Press 【SHIFT】to switch over the system modes. Here the following three modes are available: Single tool maching, dual tool machining and milling.

Single tool machining: There is only advancing axis. If the timber to be machined is large in size, several times of reciprocal machining may be needed.

Dual tool machining: There are two cutters on the machine tool. They advance from different points and in a one-follow-the-other manner. Dual tool mode can achieve deep advance.

Milling: Axis Z drives step-control spinning axis to control the angled spinning of the timber. The other two axes interpolate on plane basis. This mode is applied to twist mill the column or other occasions.

In the machining interface shown in the above Fig. 2.1, press【Start】, and you can run the current program.

1.1 Draw Reading

In the main interface, press 【F1】(machine) , and then press【F1】(draw reading) to access to the interface where you can edit or import data. See the figure below.

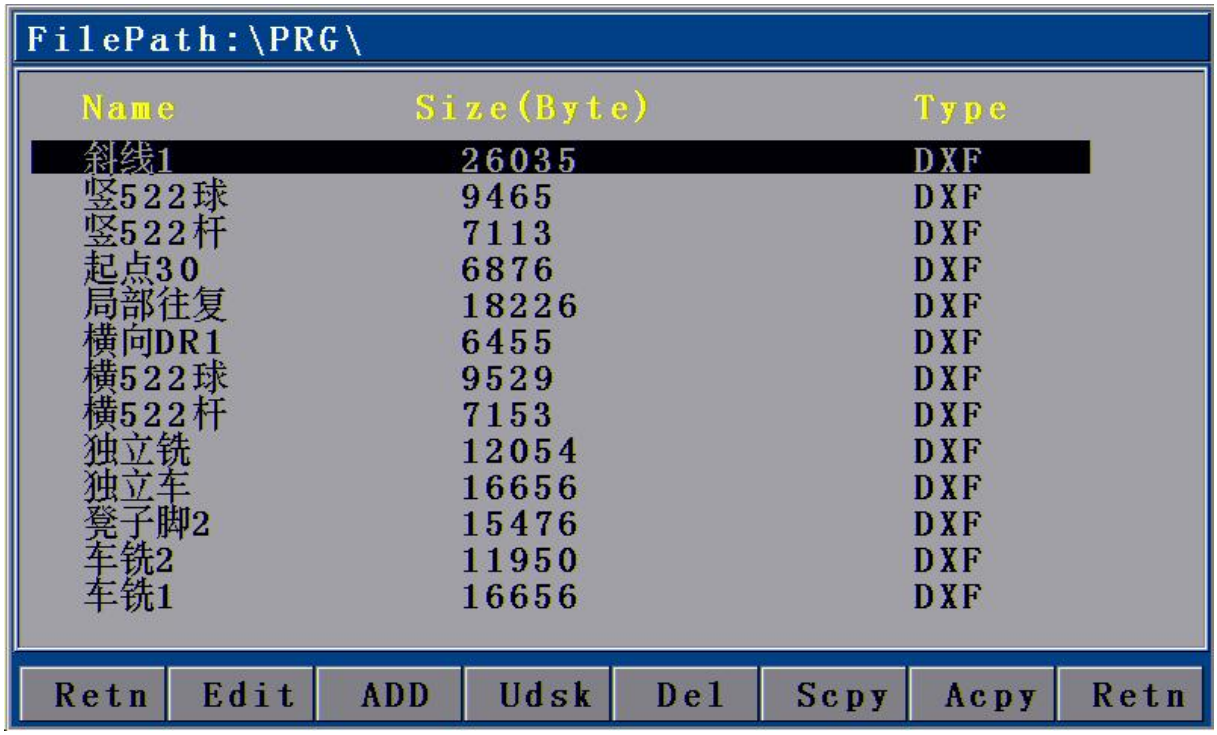


Fig2.2: File Operation Interface

1) Page scroll-up/down: Press 【Pagup / S↑】or【Pagdn / S↓】to scroll up or down the pages.

2) 【Y+】【Y-】: Move the cursor upward or downward.

1.2 Manual Mode

In the main interface, press 【F1】(machine) , and then press【F2】(manual) to access to the manual-operation interface.

1.2.1 Inching

Fig. 3.1.2.2.1 Inching Setup Interface

In the interface shown in Fig. 3.1.2.2, press 【F1】(Inch), and the above interface will appear. At the time, the field “inch” turns white, and the prompting words “enter L value” can be seen at the lower part of the screen. At the time, you can enter the length to be inched, with millimeter as the unit. After entering the data, press 【X+】, 【X-】, 【Y+】or【Y-】. Each time you press one of these keys, the corresponding motor will move the length you set. The same is true for plasma function (omitted).

When the field “inch” does not turn white, in default state, if you press 【X+】,【X-】,【Y+】, 【Y-】, 【Z+】 or【Z-】 and release the button, the chosen axis will enter the continuous motion mode. At the time, only when you press the key again or press 【Stop】, will the system decelerate and stop.

If you press the key 【G】, the system will switch over between manual and inching modes. In the inching mode, if you press 【X+】,【X-】,【Y+】or 【Y-】and hold down, the chosen axis will be in continuous motion. When you release the button, the system will decelerate and stop.

1.2.2 Reset

When the system drives the machine tool to the program home position, the coordinates will be

cleared and the machining state reset.

1.2.3 Break

Press Break, and the system will restore the machining state of the time when the power is interrupted. You must use this function immediately when the power is restored.

1.2.5 Clear

In the manual-operation mode, press 【F5】 to delete the parameters in the records. If you press 【X】or press 【Y】→【F5】(clear) →【Enter】, you can delete the X/Y coordinates of the whole line. If you only want to delete the last digit of the number, you just press 【DEL】on the panel.

1.2.6 Home position

In the manual –operation interface, press 【F6】, and the system will prompt to return to the mechanical home position. Press 【Y+】/【Y-】to select the mode for returning. Then press Enter, and the system will drive the axes to the mechanical home position according to the designated way. At the time, the coordinates will be cleared and machining state reset.

1.2.7 Coordinates setup for X, Y and Z

In manual-operation mode, press 【X】and【Y】on the panel, you can modify or delete the data of X and Y respectively.

After you press 【X】and【Y】, their current coordinate values will be seen at the lower left part of the interface. For example, X's value: X=+00123.456". At the time, you can press 【DEL】to delete the coordinates, and press【F5】(clear) to delete the data of the whole line. Press【Enter】after you set the data.

1.3 Reset

In machining interface (stop or pause state), press【F3】, and the system will first return the program home position.

Clear

In machining interface (stop or pause state), press 【F4】, and the system will automatically clear the

current coordinates to 0.

Reverse Direction

In machining interface, press 【F5】, and theleft and right machining directionswill be exchanged to each other.

Basic operation

1) I/O

Before the equipment starts machining, you can use manual knob to control the solenoid valve.

1) Press 【F↑】and【F↓】, and you can increase or decrease the speed rate. In standby mode, press this key and hold it down, the speed will continuously rise or fall. If it is in machining mode, the speed percentage will increase or decrease by 1 each time you press this key.
2) 【Pgup/S↑】and【Pgdn/S↓】are two compound keys. Press 【Pgup/S↑】 or 【Pgdn/S↓】, the cutting gun will move upward or downward, and stop when you release the key. In edition mode, press them to scroll the pages.
3) 3)Press 【F】to switch over the speed rate. If the rate ranges from 50% to 100% and you press 【F】, the speed rate will change to 5%. If 1% to 50%, to 80%.
4) Operation of graphic interface: Press the numeric keys to zoom in the graph. Press 0 to restore the original size of the graph. Press 1-9 to zoom in the graph. Here 1 means to zoom in the graph twice as much as before, 2 thrice as much as before, and so on. For instance, if you press 【2】, the graph will be zoomed in thrice as much as before, and press 【0】, restored to the original size. If you successively press 【2】 for two times, the graph will be zoomed in 5 times as much as before. And so on.

2. CAD Instruction

In the main interface, press【F2】, and the system will go to the instruction interface as seen in Fig. 2.1.

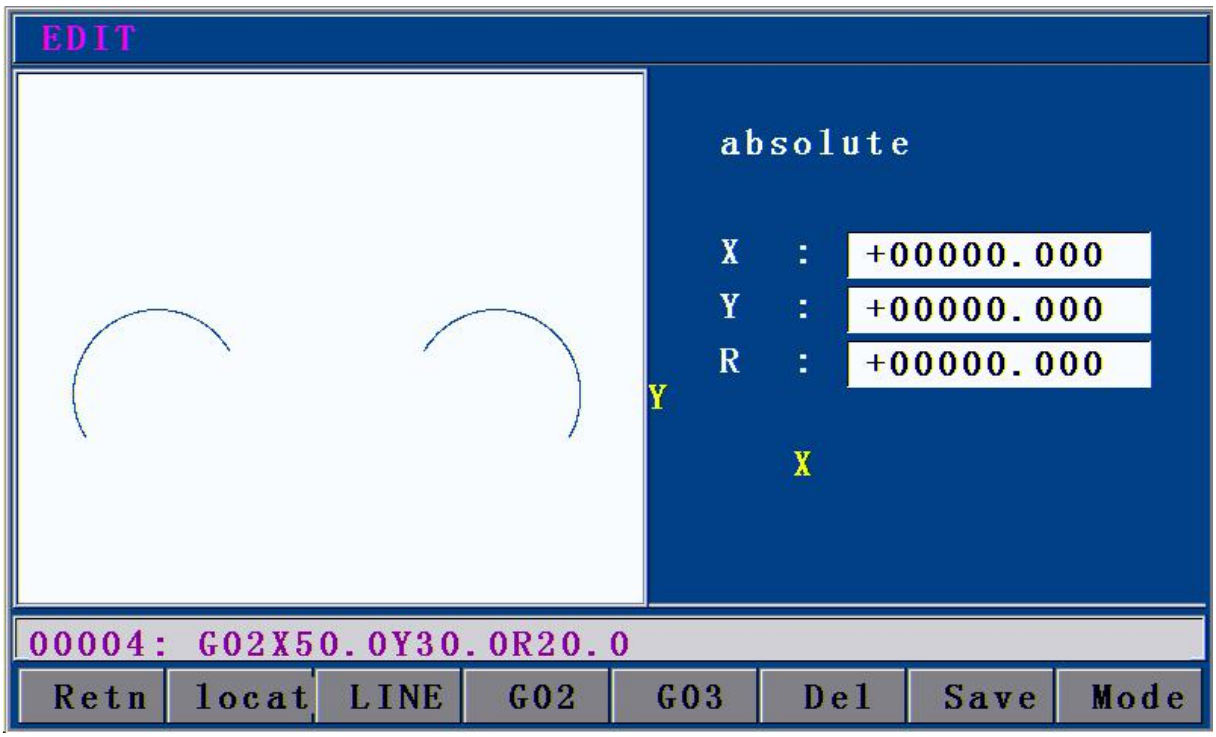


Fig. 2.1

Definitions of F0—F7 are as follows:

【F0】-Return: Return to the main interface.

【F1】-Non-load move: Press 【F1】, and the system will ask whether it is assured to edit the non-load move line. If you press 【Enter】, you should enter the coordinate of axis X, and then press 【Y+】/【Y-】to switch over to enter the coordinate of axis Y. After entering the data, press 【Enter】, and the instruction of non-load move will be created.

【F2】-Straight line: press 【F2】,and the system will ask whether it is assured to edit the machining straight line. If you press 【Enter】, you should enter the coordinate of axis X, and then press 【Y+】/【Y-】to switch over to enter the coordinate of axis Y. After entering the data, press 【Enter】, and the instruction of straight-line machining will be created.

【F3】-Clockwise arc: press【F3】,and the system will ask whether it is assured to edit the clockwise arc. If you press 【Enter】, you should enter the coordinate of axis X, and then press 【Y+】/【Y-】to switch over to enter the coordinate of axis Y and radius R. After entering the data, press 【Enter】, and the instruction of clockwise arc will be created.

【F4】-Counterclockwise arc: press 【F3】, and the system will ask whether it is assured to edit the counterclockwise arc. If you press 【Enter】, you should enter the coordinate of axis X, and then press 【Y+】/【Y-】to switch over to enter the coordinate of axis Y and radius R. After entering the

data, press 【Enter】, and the instruction of counterclockwise arc will be created.

【F5】-Delete: Press 【F5】, and the system will delete the current bar code. Press it again to delete the previous instruction.

【F6】-Save: Press it to save the edited machining code to the file.

【F7】-Method: Press it to switch over the absolute or relative programming method. The default value is the latter.

3. Scanning

In the main interface, press 【F3】, and the system will access to the scanning interface, as seen in Fig.3.1. What appears in the box is the code of the current machining file, which you can only view, but not edit. If the size of the code is more than one page, you can press 【pgup↑】/【pgdn↓】to scroll up or down the pages.

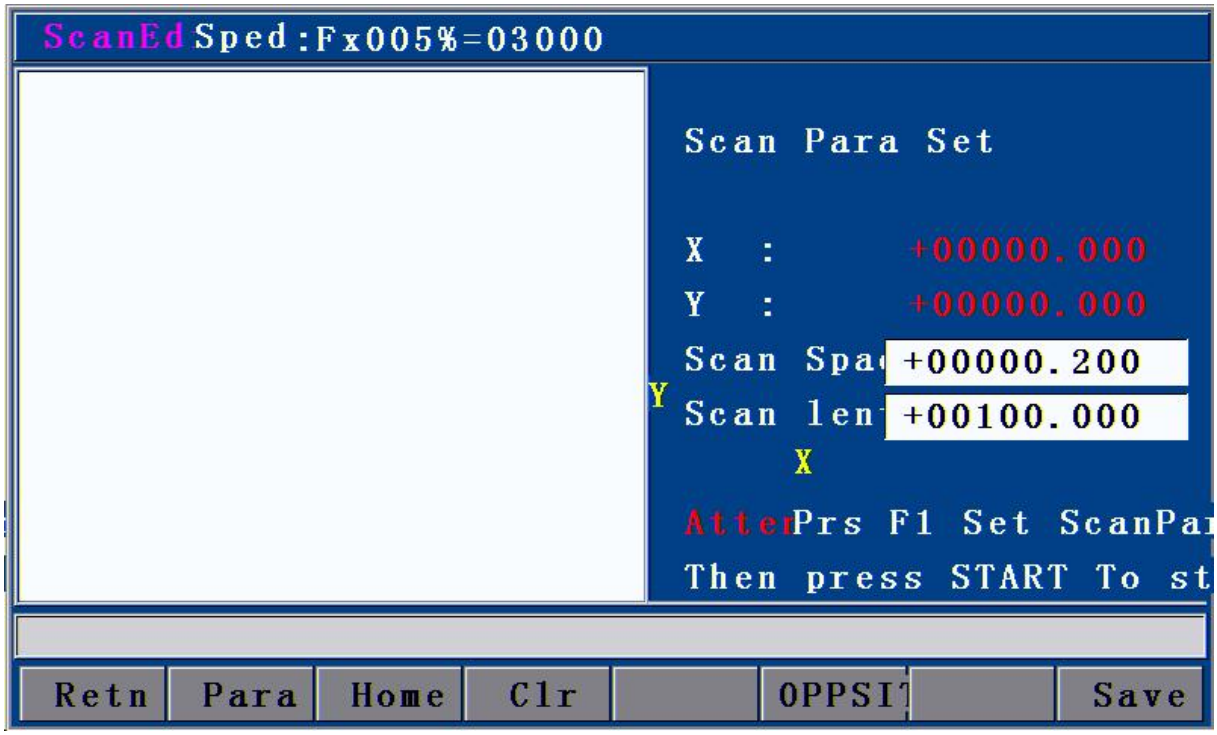


Fig. 3.1 Edit Interface

Definitions of F0—F7 are as follows:

【F0】-Return: Return to the main interface.

【F1】-Parameter: Press【F1】to set the scanning interval and total scanning length. Press 【Y+】/【Y-】to switch over the options of entering.

【F2】-Reset: Return to the program home position as set.

【F3】-Clear: Clear the current parameters and status to zero.

【F5】Reverse direction: Change the direction of the scanning axis X.

【F7】-Save: Save the current scanned code to the file and return.

4. Parameters

In the main interface, press 【F4】, and the system will access to the parameter setup interface, as seen in the figure below:

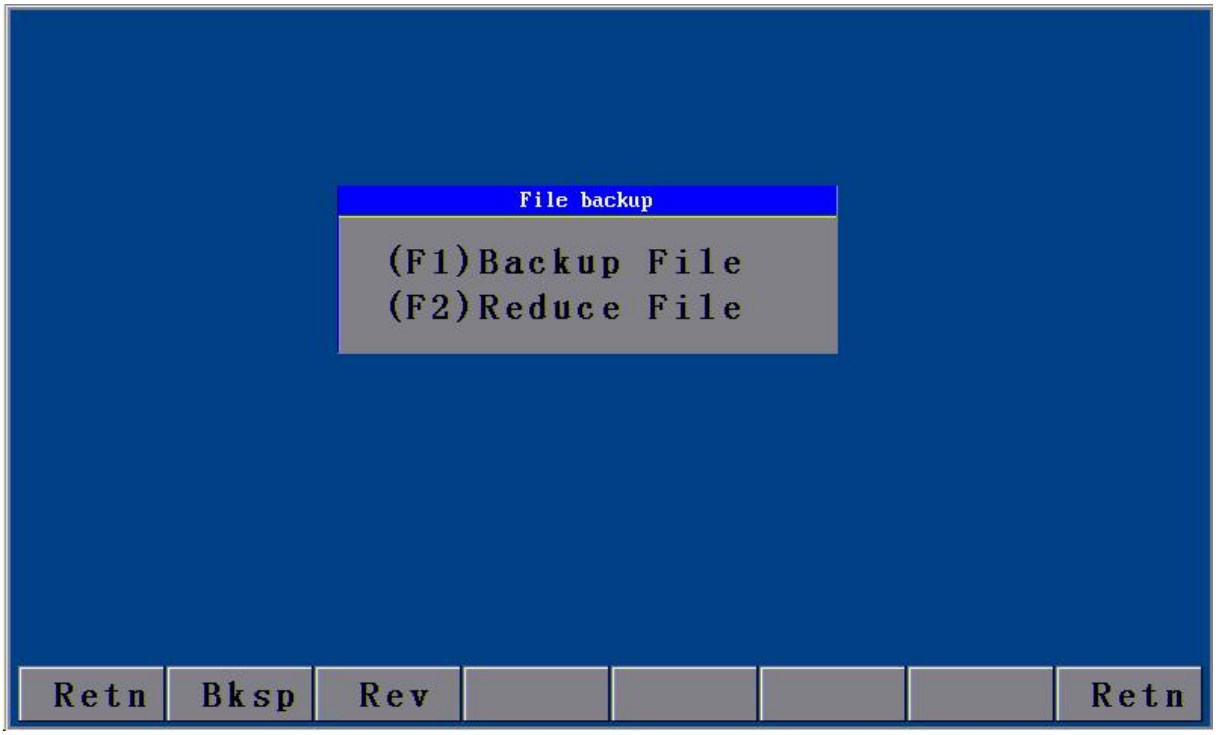
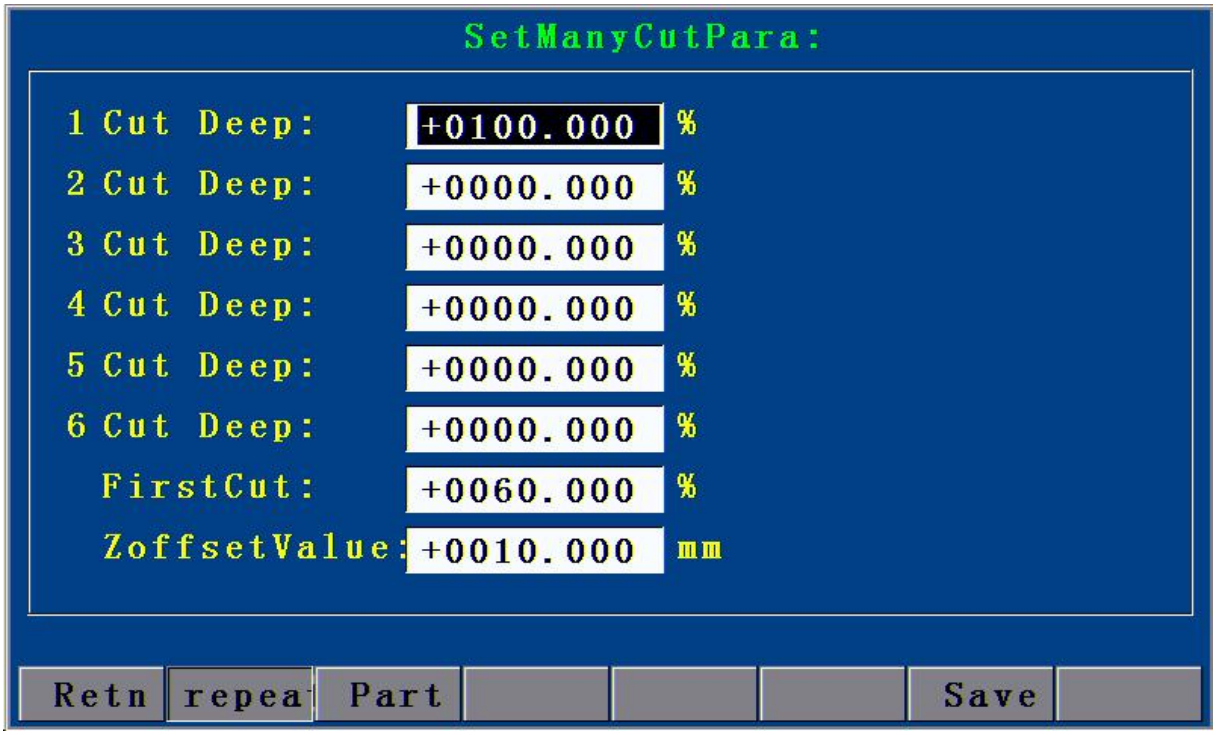


Fig.4 System Parameters Setup Interface

4.1 System Circulation



Cut depth 1-6: When there is only one time of cutting, you can set cut depth 1 as 100%. If multiple times of advance are needed, you should set the cut depth for each time according to the proportion between the absolute cutting depth and the final cutting depth.

Rough tool proportion: It refers to the proportion between the cutting amount of rough tool and the total cutting amount of the time when double tool is engaged.

Z postposition offset: It refers to the distance from axis Z tool to its cutting point when dual tool is engaged.

4.2 Speed

In the main interface, press 【F2】, and the interface below will appear, where you can set the axis speed, manual-operation speed and deceleration.

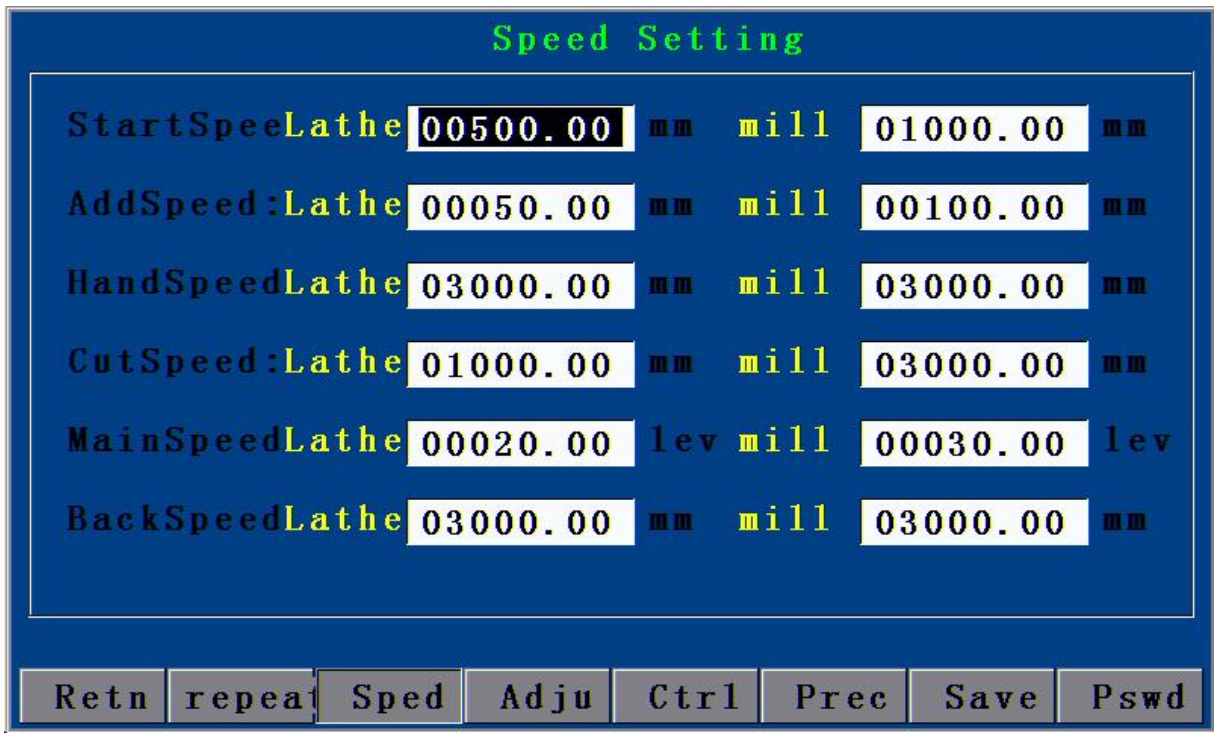


Fig. 4.2 Setting the Speed Parameters

4.2.1Setting the speed

Parameter	Details	Unit
Start speed	Initial speed of axis X/Y	mm/min
Acceleration	Each time of acceleration when X/Y raises its speed from initial value to the max. value	mm/min
Manual speed	The speed of X/Y, G00 and resetting in the manual mode	mm/min
Machining speed	The speed of X/Y and G01,G02 and G03 when the program is running	mm/min

Fig.4.2.1

Note:

In practice, the system must undergo a periodof both acceleration and deceleration during its running. Therefore, you should optimize the value of "start speed" and "acceleration" according to the pulse equivalent at the time.

.

Reference values:

Machining Speed (mm/min)	**Start Speed (mm/min)**	**Pulse Equivalent**	**Acceleration (mm/min)**
100-200	100	0.001000-0.002000	10(In practice, adjustment is needed)
200-500	200	0.002000-0.003000	20
500-700	300	0.003000-0.004000	30
700-800	500	0.004000-0.005000	40
800-1000	600	0.005000-0.006000	50
1000-1300	700	0.006000-0.007000	60
1600-2000	1000	0.007000-0.008000	70
2000-3000	1500	0.008000-0.009000	80
3000-4000	2000	0.009000-0.010000	90
		0.010000-0.011000	100
		0.011000-0.012000	110
		0.012000-0.013000	120

Fig.4.2.2

4.3 Adjustment

In the parameters setup interface as shown in Fig. 4, press 【F3】, and the interface below will appear, where you can set the following parameters: coordinates for limitation, dual-tool parameters, backlash, advance times, parameters indicating the relations of spinning axis at the time of milling, etc.

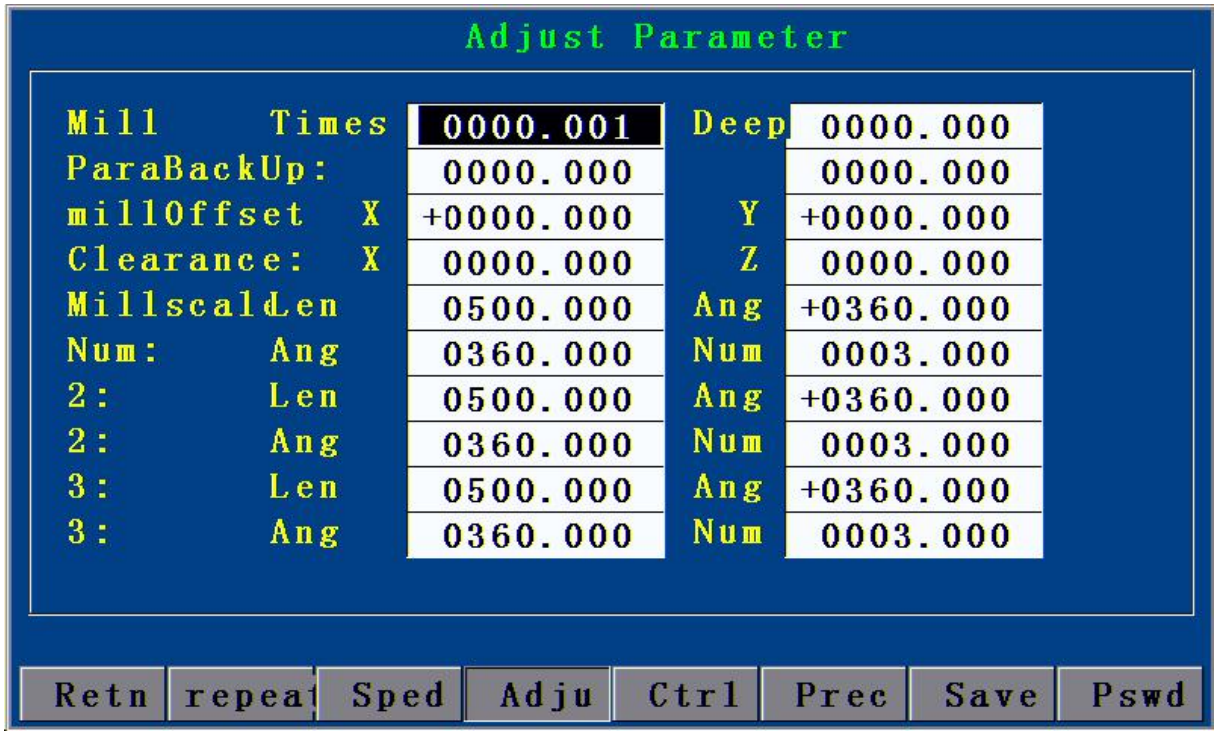


Fig.4.3 Adjusting the Parameters

4.3.1Adjust the setups

Parameters	Details	Unit
Soft positive limit	It refers to the system software limit. When the current value is higher than this value, the system will automatically stop.	mm
Dual-tool parameter	Set the proportion of two feed cutters of YZ and offset of dual tool	mm
Advance times	Times of reciprocal machining at the time of single tool	mm
Backlash	The clearance offset for the time when axis X and Y change the direction reversely.	mm
Twist proportion	It refers to the relation between the total length run by axis Y and the revolution angle.	mm
Twist number	It refers to the total angles and numbers to be machined at the time of scale division.	Angle

Fig.4.3.1

4.4 Control

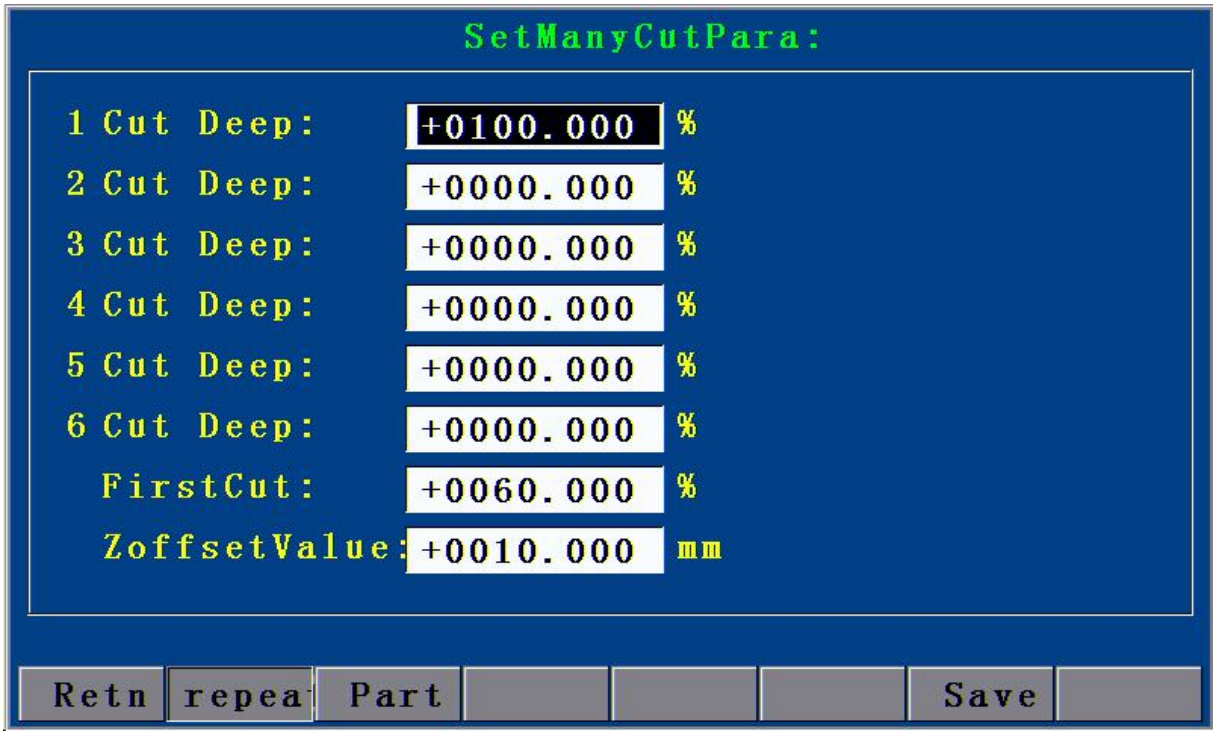


Fig.4.4.

Parameters	Details	Unit
Lathe spindle delay	The time that delays till stable running each time when the lathe spindle converter is started.	Second
Miller spindle delay	The time that delays till stable running each time when the miller spindle converter is started.	Second
Tool offset	Set the tool offset here when the system need it.	mm
Timber diameter	It refers to the diameter parameter, used as the partition of dual tool graph at the time of dual tool displaying.	mm

Fig.4.4

4.5 Precision

In the parameters setup interface as shown in Fig. 4 in this section, press【F5】, and the following interface will appear, where you can set how many millimeters axis X, Y and moves for each pulse equivalent. This value affects the precision of the track.

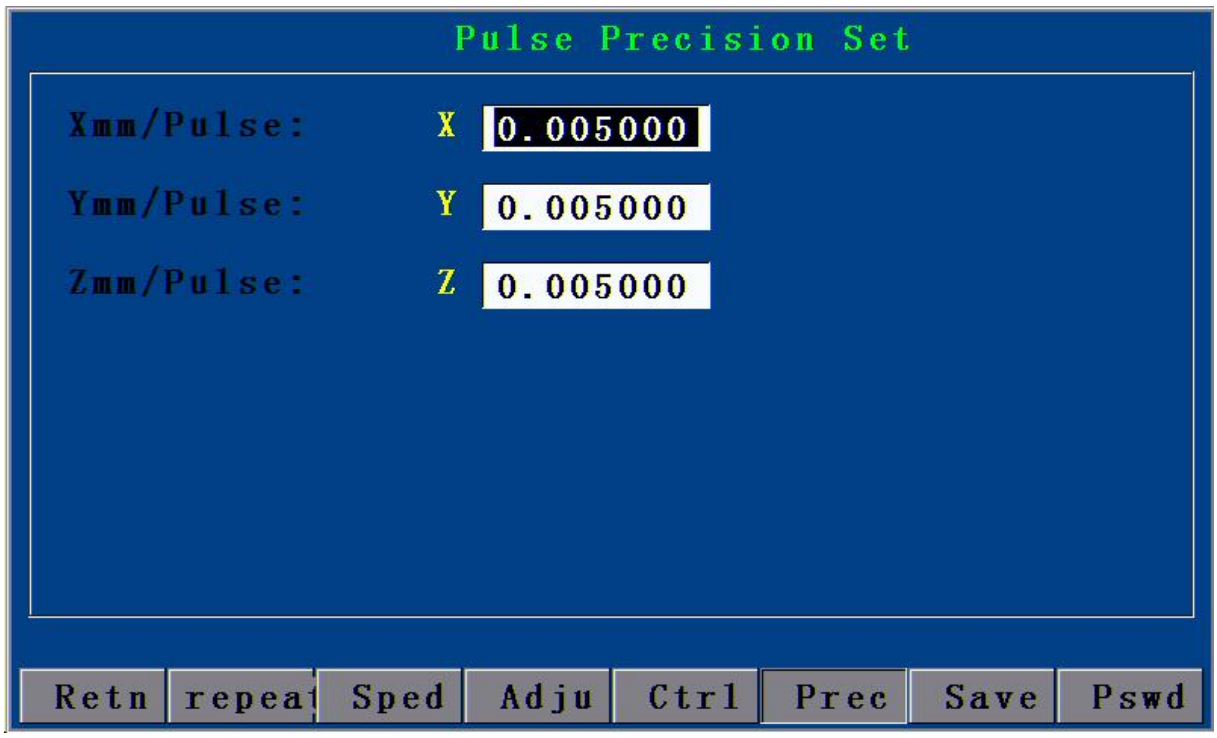


Fig. 4.5 Precision Parameters

4.6 Save Parameters to File

In the parameters setup interface as shown in Fig. 4 in this section, press【F6】, and the following interface will appear. Please refer to the prompted information and press ↑↓ to make the options. Then press Enter to allow the system to perform the function.

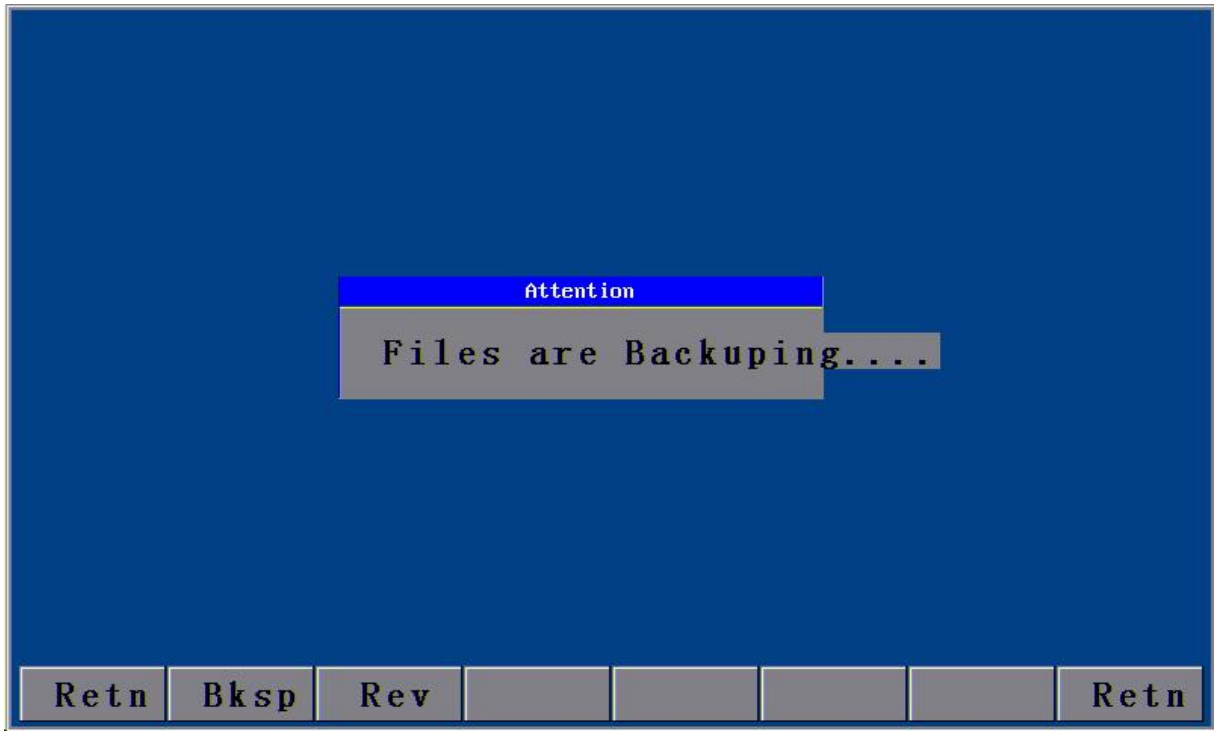


Fig. 4.6.1 Save Parameters to File

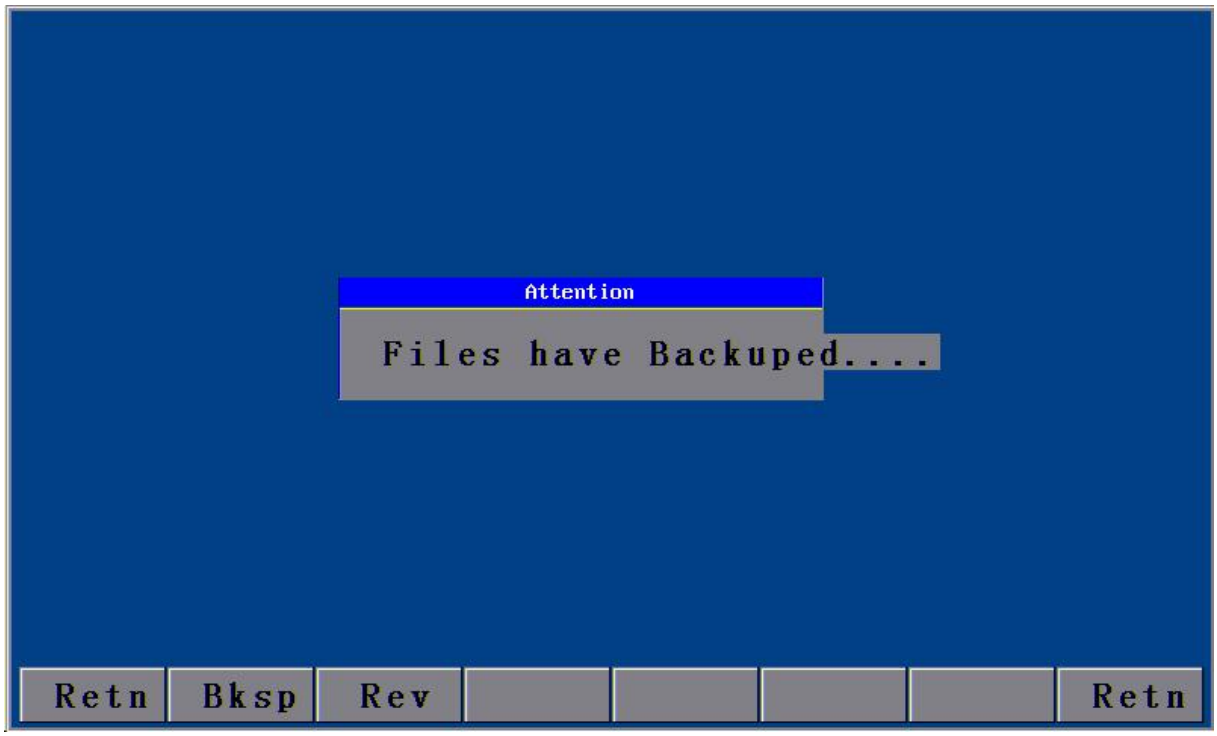


Fig. 4.6.1 Save Parameters to File

5. Diagnosis

In the main interface, press 【F5】, and the diagnosis interface will appear, as shown in Fig. 2.6 below:

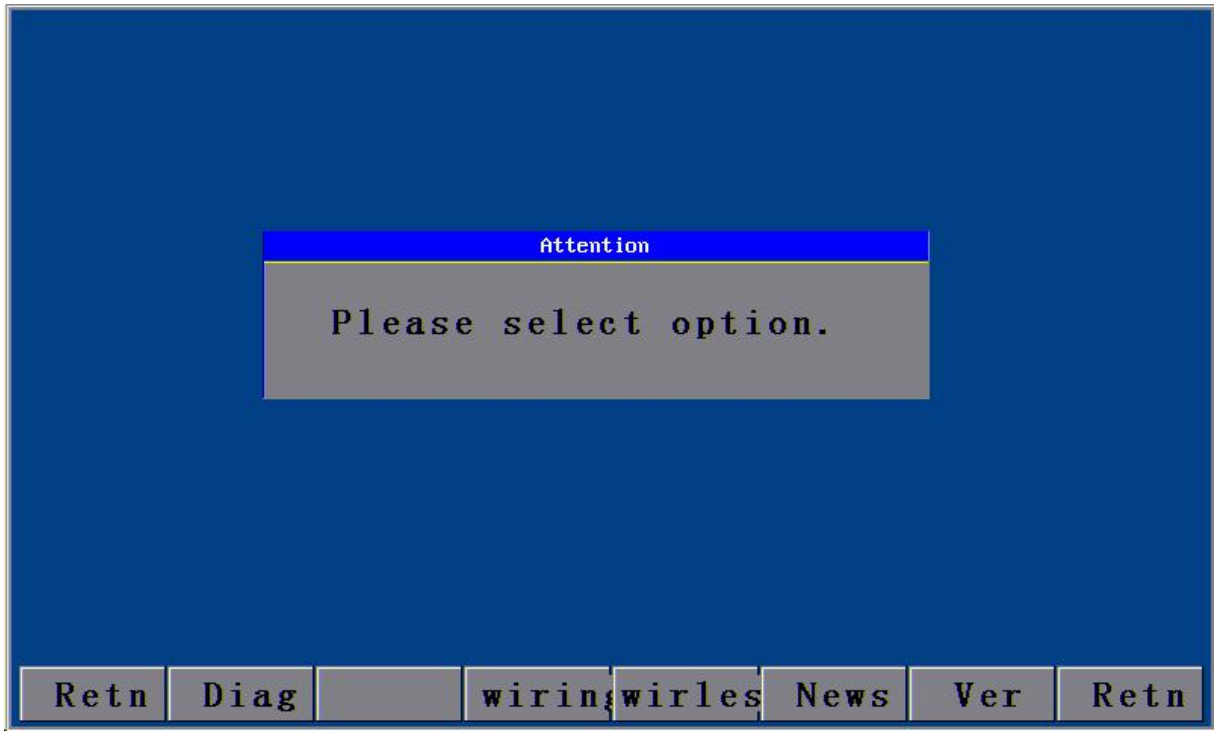


Fig. 5 Main Diagnosis Interface

5.1 Submenu

1) Diagnosis interface

In the diagnosis interface as shown in Fig. 5, press【F1】to access to the diagnosis test interface, as shown in Fig.5.1.1 below.

Press 【F1】to test the positive rotation of axis X.

Press 【F2】to test the reverse rotation of axis X.

Press 【F3】to test the positive rotation of axis Y.

Press 【F4】to test the reverse rotation of axis Y.

Press 【F5】to stop the rotation of axis X/Y.

Press 【X+】【X-】【Y+】【Y-】to move out the cursor, and press Enter to output the related switching signal.

If the switching signal is available, the red circle under the pin mark corresponding to the input diagnosis will turn yellow.

This function can be used to judge whether the input/output port is correct, damaged or normal.

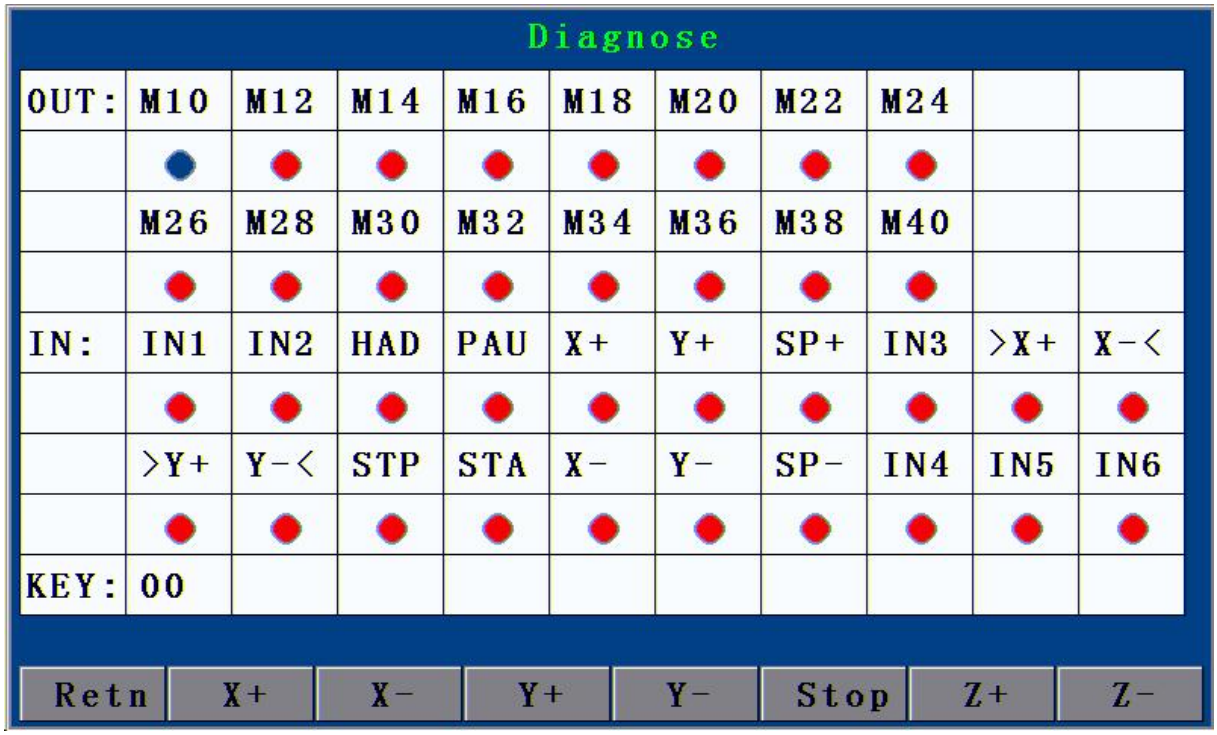


Fig.5.1.1 Diagnosis Test Interface

1) In the main system diagnosis interface as shown in Fig. 5, press 【F5】to display the latest updates of the system.

2) In the main system diagnosis interface as shown in Fig. 5, press 【F3】to access to the interface where high/low level for input/output can be changed over. The user can select the external connection as NO or NC.

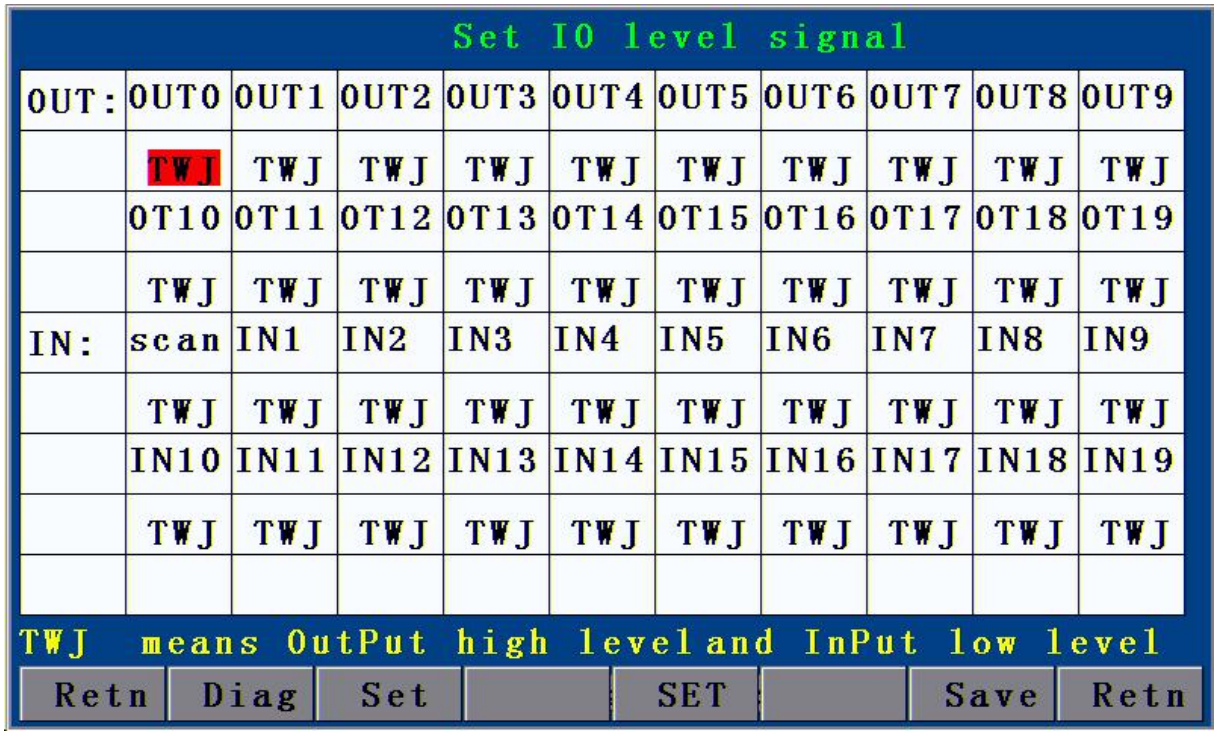


Fig. 5.1.3 Selection of Connection Mode

3) In the main system diagnosis interface as shown in Fig. 5, press 【F6】to access to the current-version interface where the current version number and other information are displayed. Like in Fig.5.1.4, the version information of MCK300A is shown.

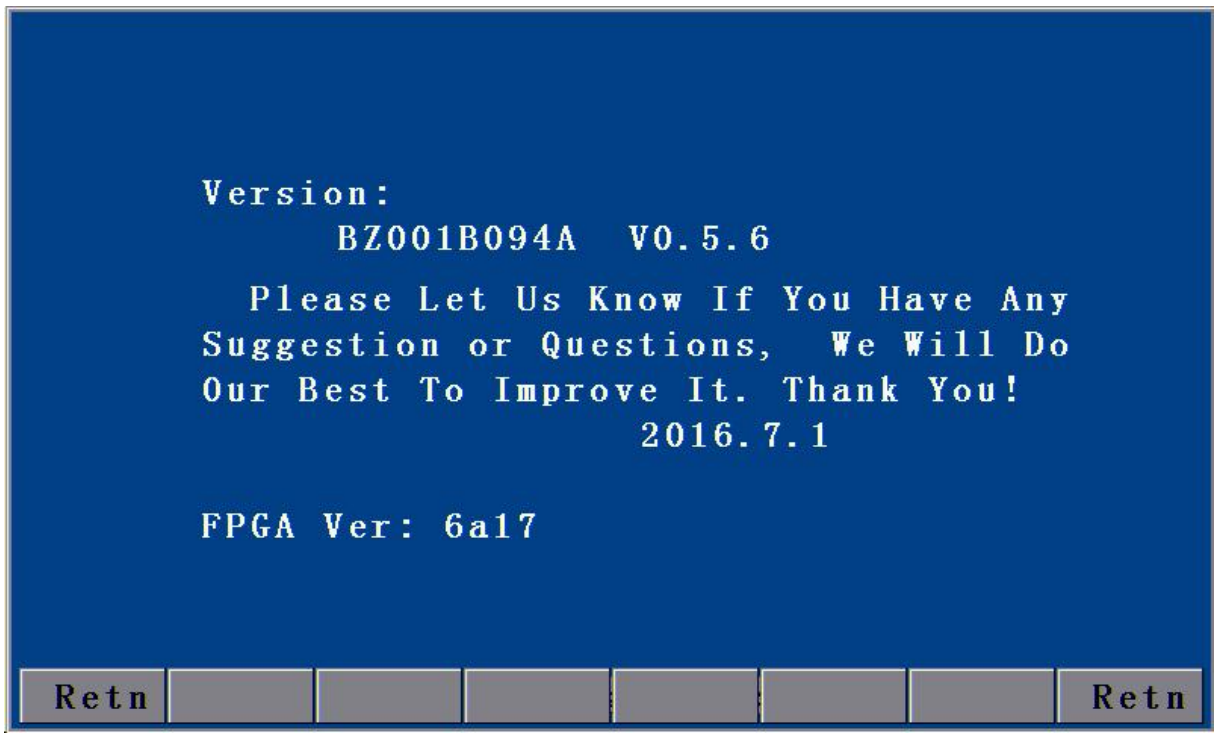


Fig. 5.1.4 ① Version Information of MCK300A

7. System Setup

System setup interface is used to set OSD language, default process or user-defined process. In the main interface, you can press【F7】to access to the system setup interface, where you can press【Y+】/【Y-】to move the cursor to the position where the item needs to be set, and press【X+】/【X-】to complete

the setting.

M07: M07 is the indicative instruction for the start of system's cutting process, whose default value includes the spindle-starting command. If it is set as user-defined, when it executes M07, the system will execute the modified instruction set of M07 as set in the edit interface.

M08: M08 includes the instruction to turn off the spindle. If it is set as user-defined, the system will also execute the instruction set of M08.

OSD language: The system provides two OSD languages—Chinese and English. You can switch it over here.

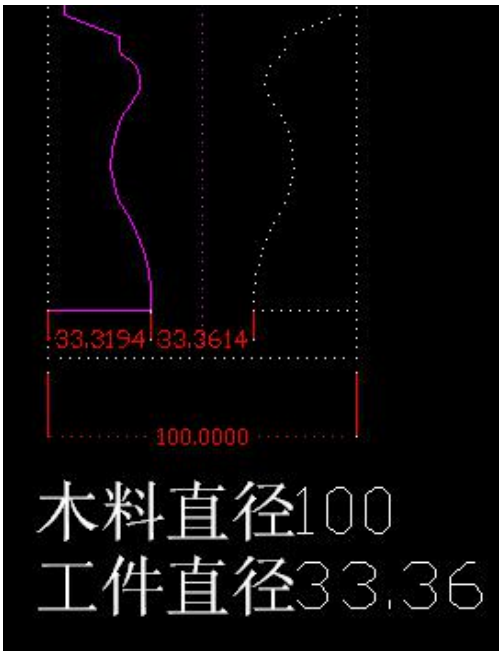
Long axis direction: It is normally set as vertical, unless otherwise specified.

Continuous twist: If the function of continuous twist is enabled, the system will start twist milling on the basis of file name order after the lathing of dual tool is completed.

8. Direct Imports of CAD Files

1. Graphics

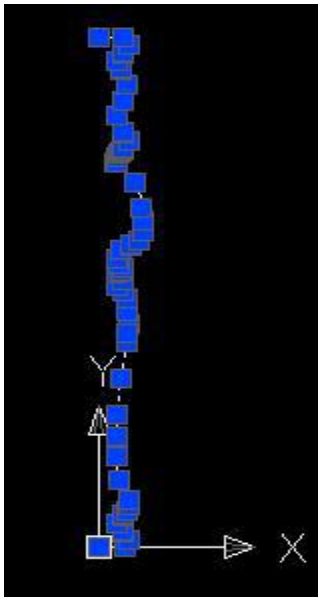
Draw the outline graphin accordance with the actual size, and use the instruction "translation"()to move the start point of the graph to the home position of CAD, as shown in the figure.



2. Save as CAD R12/LT2 DXF

If the graph contains splines, you must first save this file as CAD R12/LT2 DXF.

3. Re-open the saved CAD R12/LT2 DXF file, and select the item that allows you to see the change of the spline.



4. Use decomposition instruction to smash the graph.

5. Change the graph into multiple line segments. See details as follows.

```
Command: Designate the diagonal point
Command: *Cancel*

Command: pe
```

Input “PE” and press Enter.

```
Command: *Cancel*
Command: pe

PEDIT selects multi-segment line or [multiple pieces(M)]
```

Enter multiple M codes and press Enter, and then select the graph.

```
Selection: designate diagonal point: find 784
Selection:

Want to change the straight line and arc to multi-segment line? [Yes (Y)/No (N)] ? <Y>
```

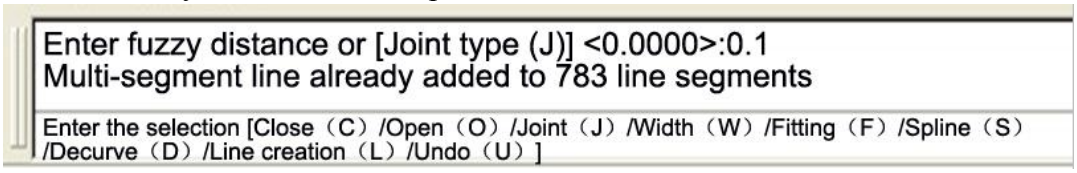
Press Enter.

```
Selection:
Want to change the straight line and arc to multi-segment line?

Enter the selection [Close（C）/Open（O）/Joint（J）/Width（W）/Fitting（F）/Spline（S）
/Decurve（D）/Line creation（L）/Undo（U）]
```

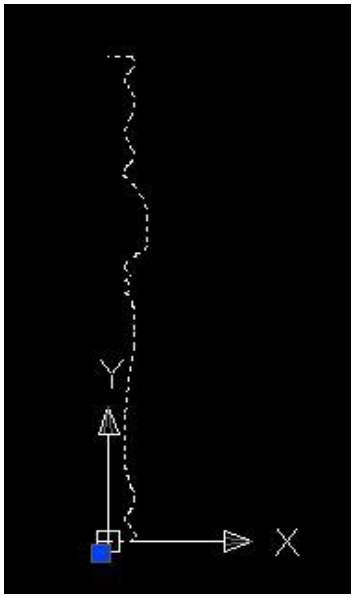
Enter “J” (joint) and press Enter.

```
Undo（U）]: j
Joint type=extend

Enter fuzzy distance or [Joint type (J)] <0.0000>:0.1
```

Enter the fuzzy distance 0.1, and press Enter.



At the time, multi-segment line has already been formed.



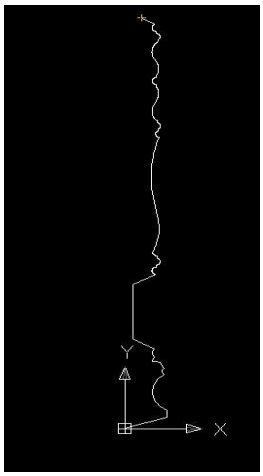
6. Save the transformed multi-segment line as CAD R12/LT2 DXF file and use an USB flash disk to copy it to the controller of MCK300A, where it will be used for machining.

9. Chinesized Software

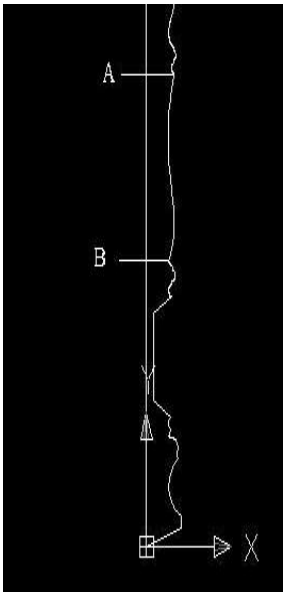
This system supports DBF type files created by Chinesized software and can directly import them for machining.

10. Twist Milling

This system supports simple twist milling.



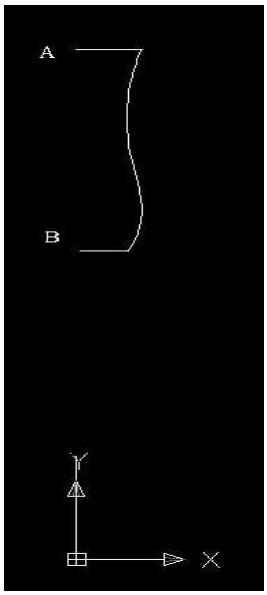
As shown in the figure, it is a DXF lathing drawing. If we want to machine a twist pattern on the seciton between point A and B in the drawing below, we can **use the decomposition instruction**  **to smash the graph afer we open the file.**



。

Draw a vertical line from the home position to the end position as the auxiliary line, and make a vertical line for the auxiliary line by passing point A and B. Then trim it.

The leftover lines from the start point to end point in machinng include: start-point advance line, machining graph for the concerned segment and retract line. Other graphs are all deleted. The leftover graph serves as the one for section A-B twist milling.



Referring to the joint method as described in the section above, enter PE instruction, and combine the leftover lines into a multi-segment line and save it in R12 format. Then exit.

Access to the parameters interface and make adjustment. Modify the twist proportion and twsit number.

Twist proportion:

It refers to the relation between the motion length of the long axis and the rotation angle.

Twist number:

It refers to the relation between the angle of the twist pattern to be machined on the column and the number to be machined.

After the parameters are set, access to the machining interface. Then swtich to the carpenter milling mode, and read the graph, align the toll and start machining.

Chapter VI Troubleshooting

1． Incorrect Drawing Size

1. Check whether the pulse equivalent is set correctly. To do that, manually press the inching button and move the axis to the corresponding length to measure it.
2. If the pulse equivalent is correct, compare the read size of CAD graph and the CAD graph itself to see if there is any deviation.
3. Check whether the tool-alignment point is correctly set.

4. Check whether there any unexpected thing happens during machining, like limit, tool collision, out-of-step, etc.

5. In the case of dual tool, check whether the offset distance between the front and back of the dual tool is correctly set.

2．Dead System

1. If the system dies after the files are imported, you can re-start the system and press 【USB FD】to access to the file edit interface where you can import the correct files. At the same time, you can check the suffix, file size or the drawn CAD graph to see ifthe imported files are correct.
2. If it is not because of the imported files that cause the death, you can check whether the pulse equivalent and acceleration are correctly set. Besides, you can restart the system and access to the diagnosis interface to see if there is any abnormal signal or the key is in effective state all the time.
3. Check “Parameters/Control” to see if the delayed time is set too high.

3．Some Solenoid Valve Doesn’t Work

If the solenoid valve doesn’t work, you can follows the rules below and judge whether it is because of the damage of the system, the relay or the solenoid itself.

1) Replace it with the solenoid valve of the same model on the equipment to see if it is damaged. Or give a manual output signal in the auto machining interface and use a multimeter to check whether there is any voltage signal at the input end of the solenoid valve.
2) If there is no voltage signal from the input end of the solenoid valve, you can open the housing, and use the same method as above to see if there is any voltage signal on the input end of the relay so as to judge whether it is damaged.
3) If there is no input voltage signal, the inside of the output end of the CNC system might have been damaged. In this case, you can replace the output end by following the steps below.

4．Inconsistence between Displayed Graph and CAD Drawing

1. When making a CAD drawing, please set the start point to the home position (0,0) of the coordinate system.
2. Make sure the first start point of the drawing is set on the outmost edge of the graph for machining. If the drawing is made vertically, the minimum value of X direction should be the start position.
3. Make sure the graph is in the first quadrant of the whole coordinate system.
4. Make sure the drawing has been combined into a multi-segment line.
5. Make sure there is no intersection or enclosure in the drawing.
6. After combination, check whether there is any needless dot or line segment aournd the drawing.
7. Save the drawing in R12 format.

5．Inconsistence between Dual-Tool Machining and Set Proportion

1. Make sure the dual-tool circulation parameter is set correctly.
2. Check whether there is any treatment that the machining times are created by selecting the gradation function after the graph is imported.
3. Check to see if the nose of the two cutters point to the central axis point.

6．Ineffective Wireless Remote Control

1. Keep the remote control and MCK300A in power-on state.
2. Check whether transmission and reception are in the same channel.
3. Check whether the matching is successful—lamp doesn't blink any more.
4. Switch to Diagnosis—Remote Control, and check whether the related analog displaying is effective after you press the key.

7．Low Speed for Scanned Drawing

1. Adjust the machining speed in the parameters.
2. Set a higher value for the scanning interval.

8．USB Flash Disk Can't Be Identified.

1. Please try to use a USB flash disk whose capacity is less than 2G.
2. Please try to format the USB flash disk to eFAT format.
3. The file system of some USB flash disks have poor compatibility. In this case, check whether the disk is damaged. If so, change to another one.

9．How to Treat the LOGO Graph

1. Open the drawing board in the accessories of the windows system, and set a picture with pixel as 800*480. Then save it as LOGOC.BMP.

2. Copy the file to USB flash disk, insert the disk to the USB port on the system panel. Then press F6 upgrading—USB flashi disk. Copy the file to the system after selection.

10．Reverse Operation Ineffective after Gradation

1. According to the system nature, reverse operation can't be performed after gradation. Please carry out the gradation operation after you confirm the machining direction and set the parameters.

Appendix 3　M Code Zoom Table

Item No.	Name	Details
1	M02	Program end
2	M07	Start cutting and turn on the spindle
3	M08	Stop cutting and turn on the spindle
4	M10/M11	Lathe spindle switch: M10（ON），M11（OFF）
5	M12/M13	Miller spindle switch: M12（ON），M13（OFF）
6	M14/M15	Rear ejection switch: M14（ON），M15（OFF）
7	M16/M17	Output 1 switch: M16（ON），M17（OFF）

8	M18/M19	Output 2 switch: M18（ON），M19（OFF）
9	M20/M21	Output 3 switch: M20（ON），M21（OFF）
10	M22/M23	Output 4 switch:M22（ON），M23（OFF）

Modification Records (I)

Feedback by		Date		Current Version/Total Pages	
Problem Description					

Confirmed by Engineer					
Version (after modified)		Total Pages (after modified)		Modified by	