Content

Chapter 1 Introduction

Welcome to use our computerized controller for engraving machine. Illustrated with a lot of examples and figures, this manual mainly introduces its features and operating procedures of each function in details. Before operation, please read this manual carefully to ensure correct use of the controller and no occurrence of accidents. And please keep it properly for the convenience of your reference at any time.

This control system is of professional three-axis motion controller based on the embedded platform, running independently without PC. The embedded operation system is adopted so that computer virus can be avoided. To achieve high machining efficiency and high quality machining surface, advanced motion control algorithm of self-adoptive look-ahead velocity and spline interpolation are applied in this system. It is very easy for users to operate, learn and understand. And the installation is easy with little space occupation. This system can be applied to all kinds of engraving machines, engraving and milling machines, and cutting machines.

1.1 Precautions

1.1.1 Installation Environment of Engraving Machine
◆ Solid ground;
◆ Avoid direct sunlight;
◆ Leave some space for maintenance;
◆ Space temperature: 5℃ to 40℃;
◆ Relative humidity: 30％ to 95％ RH;
◆ Install the devices in the horizontal position;
◆ Well-ventilated.

1.1.2 Usage Precautions of Engraving Machine
◆ Do not use this product in strong interference and magnetic field environment;
◆ Don't plug and pull cable in control box with power on;
◆ Pay attention to waterproof, dustproof and fire prevention;
◆ Keep conducting material like metal out of the inner case,
◆ Unauthorized dismantlement is not allowed as no inner parts are expected to be repaired by operators;
◆ Plug and pull U disk and other wirings with moderate strength;

◆ Cut the power off when the controller isn't in use for a long period of time and keep it properly;

◆ Don't touch the working engraving knives with your hands in case of injuries as they are very sharp. And don't contact them with handkerchief or silk scarves to avoid injuries or equipment damaging;

◆ Cut the power off when overhauling and adjusting the machine;

◆ Operators and servicemen shall be well-trained.

1.2 System Features

◆ Compatible with data format such as standard G code, PLT and Eng. Support mainstream CAD/CAM software, such as ArtCam, MasterCam, ProE, etc., and data generated by full series of ENG5.18 to ENG5.50;

◆ Max. amount of axes to control: triple; Support double/triple axes linear interpolation and double axes circular interpolation ;

◆ Adopt triple axes spline interpolation function, operating fitting-interpolation to small lines under condition of spline, improving surface quality;

◆ Operators can realize interaction of outside files with system through U disk without network completely;

◆ Pretreatment of multi-lines and advanced self-adoptive look-ahead velocity control over machining path to realize fast speed, high precision, and good continuity;

◆ Constant small lines machining at high speed and automatic selection of the most efficiency one among various kinds of small lines control algorithms;

◆ The standard 4G data storage space can be extended to 32G at most, supporting program file of large capacity;

◆ Manual, Auto processing function;

◆ Block-skipping executive function: machining according to specified machining lines;

◆ Possess the functions of breakpoint memorizing and power failure auto protection;

◆ Machinery fault diagnosis and system log;

◆ Auto tool setting function;

◆ Possess the functions of automatic returning to origin, tool setting

◆ Built-in program file editing manager: can check the file with standard G code

below 5M;

◆ The max pulse output frequency: 500KHZ;

◆ Accelerating and decelerating mode: supporting straight line, S curved line accelerating and decelerating.

Chapter 2 Function and Operation Method of Panel Keys

2.1 Panel Keys

The arrangement of panel keys is shown as the following figure 2-1:

Fig 2-1 S100 Panel Keys

2.2 Function of Press Keys

 ：XY axis coordinates back to 0; input number "1"; upward movement.

：Page up; input number "2".

：Y axis forward movement; input number "3".

：Z axis forward movement; input number "4".

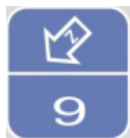：Z axis coordinate back to 0; input number "5"; downward movement.

：X axis backward movement; input number "6"; page down.

：High/Low Speed Switch; input number "7".

：X axis forward movement; input number "8".

：Z axis backward movement; input number "9".

：Y axis backward movement; input number "0".

：Enter Main Menu.

：Back to workpiece zero position; input symbol ".".

 ：Back to machinery zero position; input symbol "-".

 ：Main Shaft Start/ Stop; Delete function.

 ：SHIFT , used for combined keys.

 ：Select keys in manual mode, can select "Resume", "Stepping", "Step

Length".

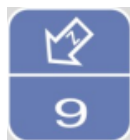 ：Can enter advanced starting interface directly.

 ：Start, Pause.

 ：OK.

 ：Cancel, Stop.

2.3 Combined Keys

 + ：Floating Tool Setting function.

 + ：Fixed Tool Setting function.

切换 SHIFT + **高级开始/断点继续 ADVANCED** ：Resume Breakpoint function.

切换 SHIFT + **手动模式 MODE** ：Feedrate Setting Function.

切换 SHIFT + **回机床零 HOME —** ：Switch to display workpiece coordinate and machinery

coordinate.

切换 SHIFT + **8** ：Save and load workpiece coordinate.

切换 SHIFT + **回工件零 ORIGIN ·** ：Open Minor Adjustment function during machining.

切换 SHIFT + **XY→0 1▲**, **2**, **3**, **4**, **Z→0 5▼**, **6** ：Switch

the coordinates among 6 workpieces from G54 to G59.

Chapter 3 How to Input Processing File

Input progressing file: import from U disk which need CAD/CAM software to assist in generating machining path and U disk to import into the system.

3.1 Import Processing File from U Disk

When begin to process a new file in the U disk, operators firstly have to import the new file into the system memory from the U disk to do engraving, instead of reading the file directly from the U disk.

Press the button **"Menu"** on the main interface to enter main menu to select the **"Processing File Management** "on the main interface (Shown as Fig3-1):
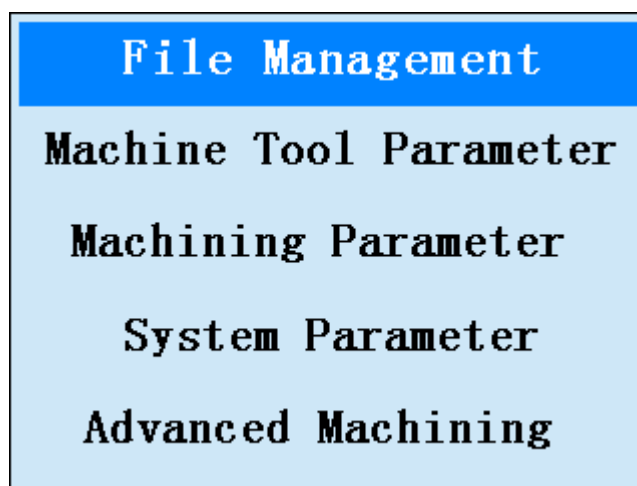
Fig 3-1 "Main Menu" Window

Entering next level menu, please select "Copy File" (Shown as 3-2):
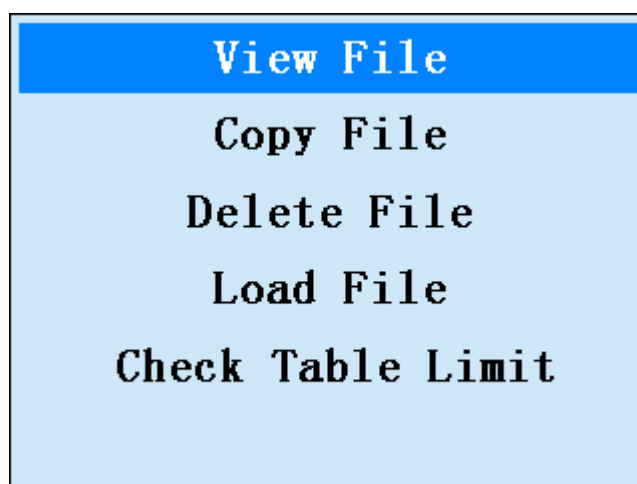
Fig 3-2 "File Operation" Window

Entering next level menu, please select "U Disk File List" (Shown as Fig3-3):
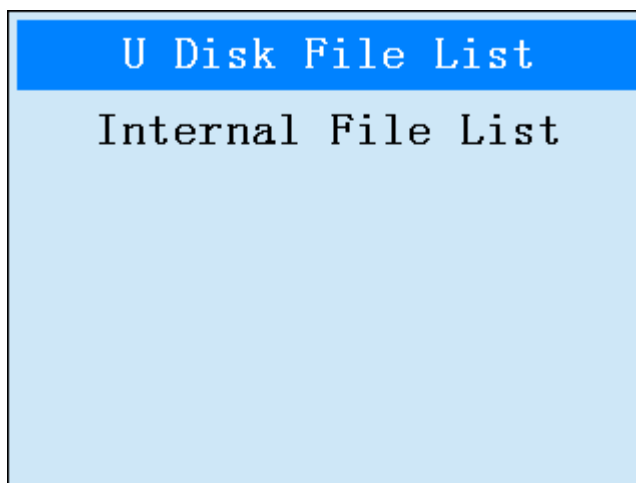
Fig 3-3 File Path Selection

After identifying U Disk, the system will display all supporting file names (Shown as Fig 3-4):

Fig 3-4 File List of U Disk

Select the file needed to process in the U Disk, and then click "OK" button. The system will import the file needed to process from the U Disk to the system memory. During the importing procedure, there is a progress bar shown on the window to prompt users the process of importing. After finishing the whole procedure, this process bar will disappear automatically.

If accessing U Disk failed or U Disk is not found, the notice window will pop up:
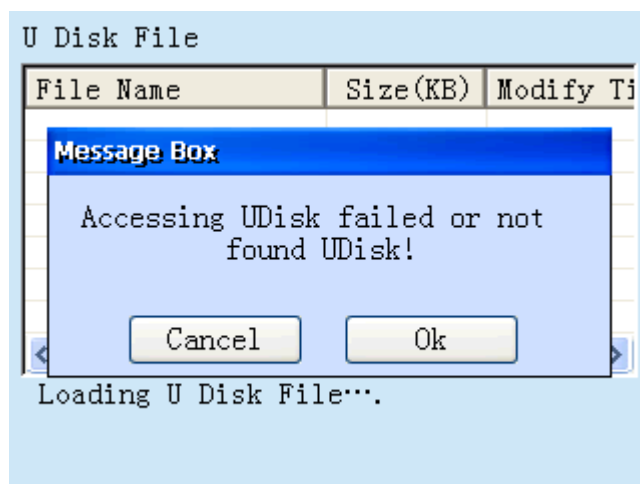
Fig 3-5 U Disk File Prompt Dialog Box

Chapter 4 How to Machine Manually

Manual Machining refers to machine tool carrying out machining by manual according to the parameters set by the users. There are three modes of machine tool manual operation, i.e., **constant micro mode**, **incremental mode** and **stepping mode**.

Operators can select the constant micro mode to process the program files. Click the **"Manual"** button at the window of the main interface to switch then you can implement the corresponding operation at this interface manually. There are six manual buttons at the window, corresponding to the positive and negative directions of X/Y/Z axes respectively. The manual window provides operators with an interactive operating environment to operate the machine tool manually.

Fig 4-1 Manual Key Area

4.1 Constant Micro Mode

If the main interface under the mode of idle state, users can switch to manual constant micro mode through the "Manual Mode" button to enter into constant micro mode. When press the corresponding direction button, the machine began to loosen the button, the machine stop working.

4.2 Incremental Stepping Mode

The same as the constant micro mode, the incremental stepping mode (use incremental mode for short) is another mode of manual machine tool operation. The difference is that the incremental stepping mode can precisely control the feeding distance of the motion axis of the machine tool.

If the main interface under the mode of idle state, users can switch to manual stepping

or incremental mode through the "Manual Mode" button. Under the stepping mode, press the corresponding direction button and then loose, the machine tool run constantly in the corresponding direction for 0.1mm step length. When switch to incremental mode, press the "OK" after setting the incremental distance. Press the corresponding direction key, and then loose, the machine tool run the corresponding distance of the indicated direction.

Note: Don't set the micro step of the Z axis too large so that the machine tool won't be damaged due to wrong operations.

Chapter 5 How to Setup Workpiece Origin

Before machining the files, operators can adjust the position of the tools and workpieces manually so that the machine can work at the preset position of the workpiece.

Workpiece origin of X/Y axes setting: run the X/Y axes to the preset position manually and press the button "xy->/1" to clear the coordinate value of X/Y axes at current position. The figure below is the button area of workpiece origin setting:

Figure 5-1 Workpiece Origin Setting

There are two ways to set up the workpiece origin of Z axis, i.e., **Manual Setting**, **Floating Tool Setting**.

1. **Manual Setting** is similar to the X/Y workpiece origin setting.

2. **Floating Tool Setting** makes it convenient for operators to locate the height of the workpiece surface and set the workpiece origin of Z axis. Specific operations are as follows: Put the feeler block on the surface of the workpiece and move the tool nose above the workpiece origin through manual operation. Then press the combined key "Shift" + "Z-", the system will pop up a dialogue box, inquiring if the position of the tool setting is right. Then click **"OK"**. When the machine tool is performing the tool setting operation, the tool nose will touch the feeler block and then it will raise 10 mm by itself. Together with the thickness of the feeler block, the coordinate of the Z axis will be identified.

Top

①

To the surface of
feeler block

Record workpiece of current point
in Z direction

②Raise 10mm

Surface

Feeler Block

Workpiece surface
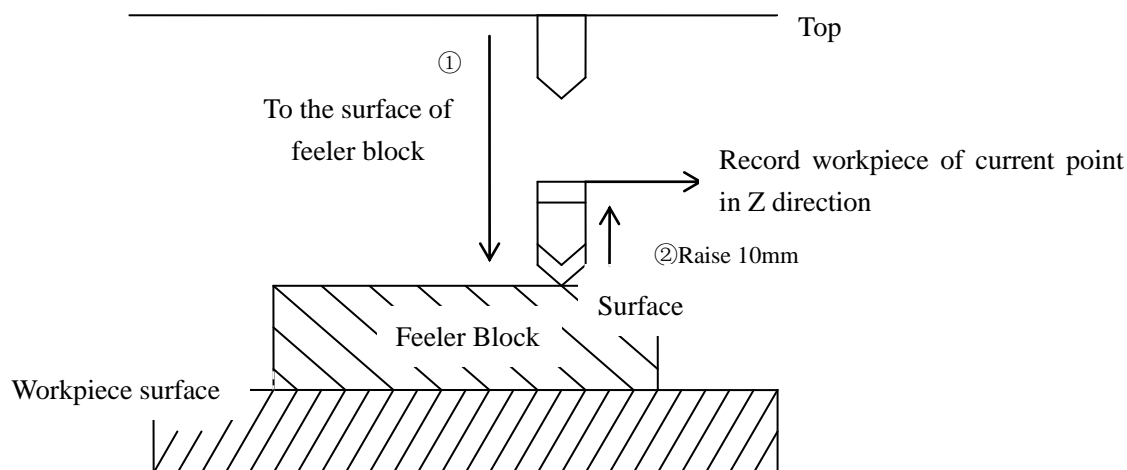
Fig 5-2 Diagram of Floating Tool Setting

Note:

(1) Before tool setting, operators have to make sure that the tool nose is above the feeler block, which means that the tool nose shall touch the feeler block when feeding the cutter, otherwise the cutter head and workpiece will be damaged as the machine tool keeps on feeding.

(2) The thickness of the feeler block can be set in **"Factory Parameter"**. The workpiece origin coordinate in Z axis will compensate the thickness after setting the tool.

(3) The speed of tool setting, ranging from 60 to 1000 mm/min, can be set in **"Parameter Setting"**. If the speed is over the maximum of the set parameter, then the cutter head or feeler block will be worn-out.

Chapter 6 How to Select File for Machining

6.1 File Loading and Auto Machining

Press "Start/Pause" button, if the system exists loaded file, the dialog box of current file processing information will prompt. Under this dialog box, you can switch to select Feedrate, Idling Speed and Main Shaft Parameter. After selecting, you can enter number key to modify these three parameters. Press the "OK" button to start machining current file. If you press the "Cancel" button, then you will enter processing file list selection through the Up/Down keys to select file. Click "OK" button to load. After finishing loading, the processing information will be displayed, press "OK" key to start machining. If the system does not exist processing file, you will enter file selection interface directly. You can select file through " ▲", "▼" button, and press "OK" button to load file. After loading, the machining file information will be displayed, press the "OK" button to start machining.
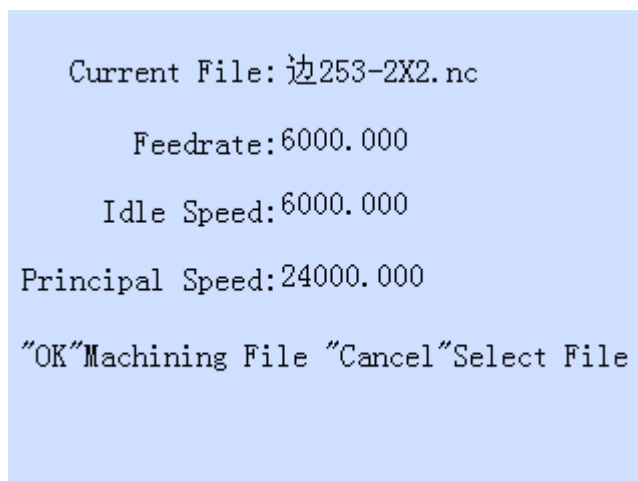
Fig 6-1 Machining File Information
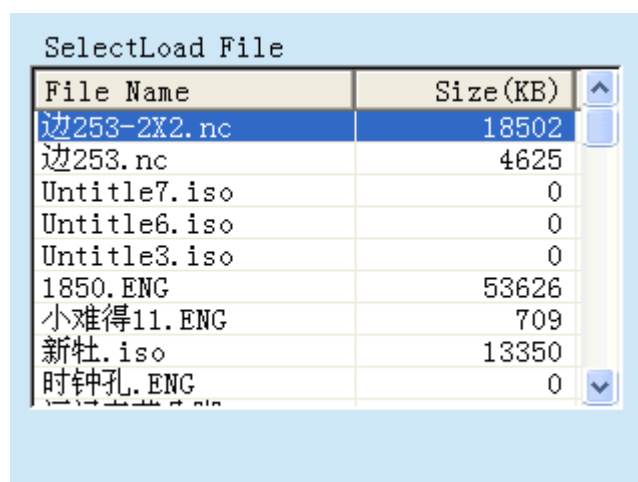
Fig 6-2 Machining File Selection

6.2 Workpiece Origin Setting

See **Chapter Five** for details. And if the workpiece origin is already set, then there is no need to set again.

6.3 Relevant Operations for Auto Machining

Relevant operations for auto machining:

(1) Through "▲", "▼" to change feedrate.

(2) Through "∧", "∨" to change the federate of main shaft.

(3) Stop.

(4) Pause.

Feedrate Adjust   Spindle Feedrate Adjust
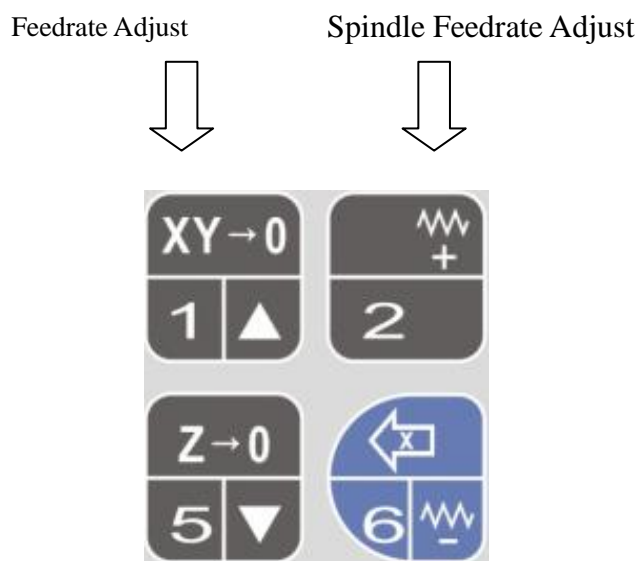
Fig 6-3 The Buttons of Feedrate Adjustment and Spindle Feedrate Adjustment on the Keyboard

```
1X      216.930    Auto
1Y       10.060    Start
1Z       10.000    Low
Consecutive          100%
                     100%
Main Spindle Being Started········
```

Fig 6-4 Interface Status When Auto Machining

Note:

The system will check the syntax of the automatic program file while machining and the syntax checking is more earlier than automatically machining (namely, syntax check has **"look-head"** function). If the syntax error is checked out in one line of the program in the program file, then the system will display the wrong sentence with the red highlight and alarm in the automatic machining window, and stop the machine tool automatically. Operators can output the file of U disk, and then check, modify and edit the grammar and syntax of the wrong sentence, in the computer, and then save it and input to the system again for loading. Using **"Area Machining"** in the advanced machining to specify starting from the modified line, the program will resume automatic machining from where it is modified.

During automatic machining, you cannot enter menu to operation it.

Both of the start and stop information of the automatic machining will be saved to the system log, which records important operations by operators and the events that have already happened. From the **"System Log Information"** window, operators can not only scan the log information produced since this start, but also review the records of the historical information. This function can help operators analyze and diagnose the system when it breaks down.

Fig 6-5 Function of System Log

The log information currently recorded by the system includes:

(1) Start and stop information of automatic machining;

(2) System alarm information;

(3) Finished information of the program file;

(4) Other information about the system.

Note:

Operators shall clear the system log regularly; otherwise the system will work slowly due to too many logs.

6.3.1 Pause

When there is a need to pause after starting automatic machining, operators can select the **"Start/Pause"** item. The machine tool will slow down from the current speed till it is zero, and it decide whether to lift tool or stop the main shaft according to the parameter setting.

6.3.2 Stop

When operators want to stop the program file after starting automatic machining, you can select the **"Stop"** item. The machine tool will slow down from the current speed till it is zero, then it will raise the cutter. And the system will save the breakpoint automatically after the stop.

6.3.3 Resume at Breakpoint

If operators want to resume machining from the last stop of the machine tool, you can

press combined key "Shift" + "Advanced Start" to enter **"Resume at Breakpoint Status"** item. And at this time, the window will pop up the information of processing file. Click "OK" and it will prompt whether to implement **"Resume at Breakpoint"** function, and operators can click the "OK" again to start machining from the stop point. If operators want to resume at breakpoint because of power failure during the machining, then operators shall make the machine tool go back to the mechanical origin before resuming. Shown as following picture:

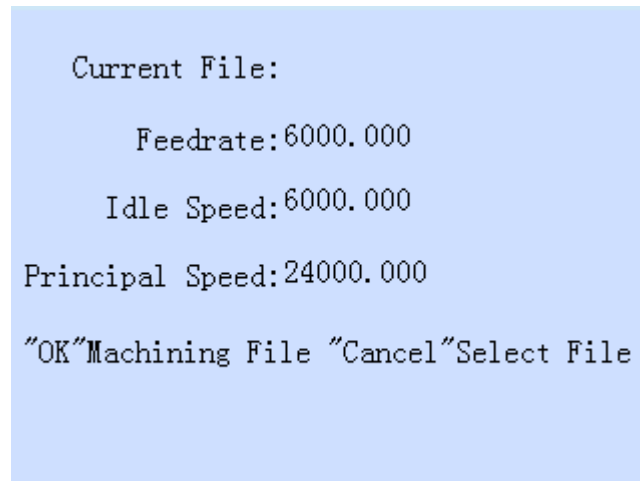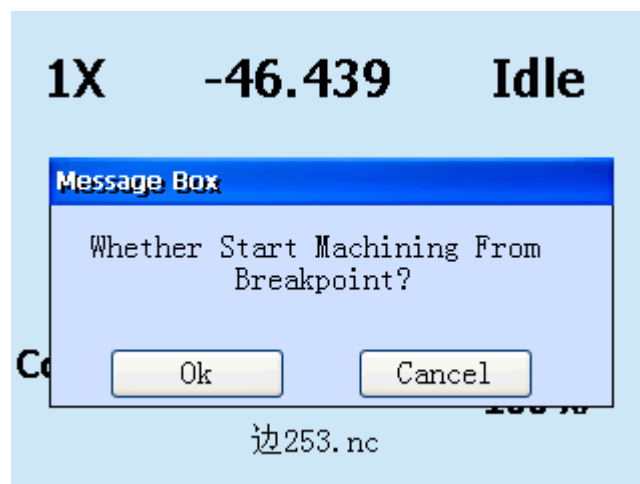Fig 6-6 Information of Current Processing File

Fig 6-7 Prompt Wheter to Start Machining from Breakpoint

6.3.4 Advanced Start

Sometimes there is no need for operators to perform the whole file machining, but starting from and finish at the specified line of the file, which is defined as **"Area Machining"** in auto machining.

Press **"Advanced Start"** and select the **"Area Machining"** item through up/down

button on the menu. This function can realize program skipping execution. Select this function and press "OK". The system will pop up the dialogue box of **"Execute"** (**"Advanced"** item) after selecting this function. Input corresponding start/end line number, and then press "OK" to start machining. See the figure 6-8.
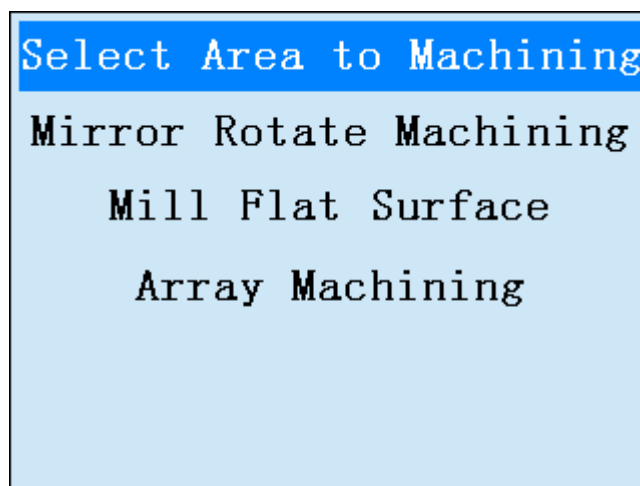
Fig 6-8 Advanced Machining Menu
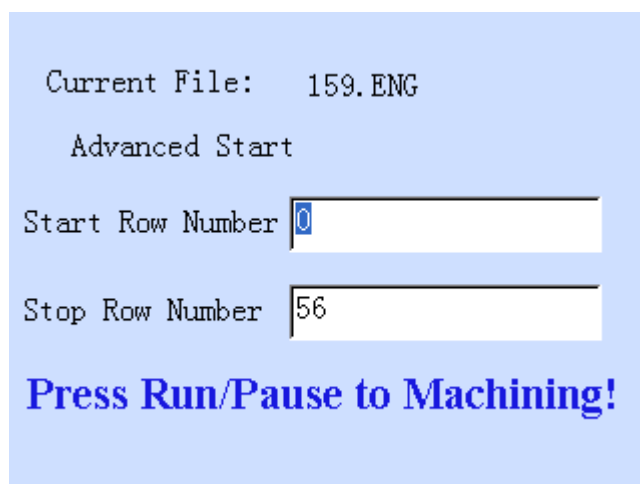
Fig 6-9 Area Machining

Users can set the line number of start and finishing position, and click the **"OK"** button, then the machine tool will only executive the specified program segment of the whole machining program according to your setting.

Note:

If operators select from the beginning and ending of the file, then the machine tool will machining the whole program file, during which we consider it the max range of the skipping execution.

The **"Advanced Start"** function will make it convenient for operators to perform the

machining of the program segment which they are interested, and it also can be used to check whether the certain program segment of the program file is right or not.

6.3.5 Array Machining

Press "Advanced Start" in the main interface or click "Advanced Machining Configuration" menu in the main menu to enter advanced machining menu. Select "Array Machining" to enter the source list of array files (as shown in figure 6-10)

Fig 6-10

Selected a file, and then click "OK" to enter the input interface of array machining information (as shown in figure 6-11).

Fig 6-11

Input the corresponding number of rows, columns, column spacing, row spacing and then press the "Start/Pause" key to start generating array files. The prompt dialog box will pop up after array files are generated (as shown in figure 6-12).
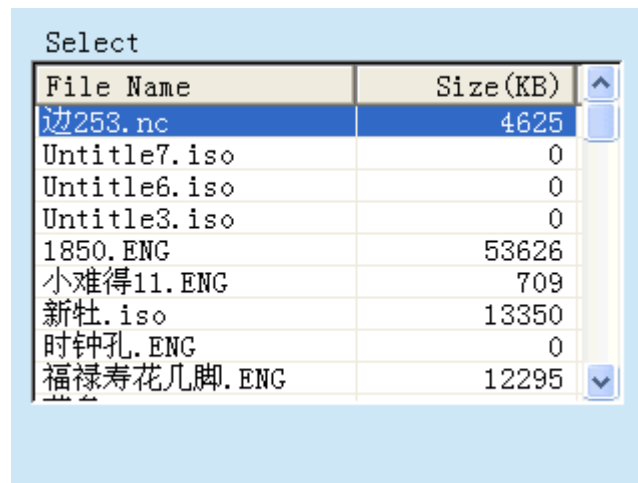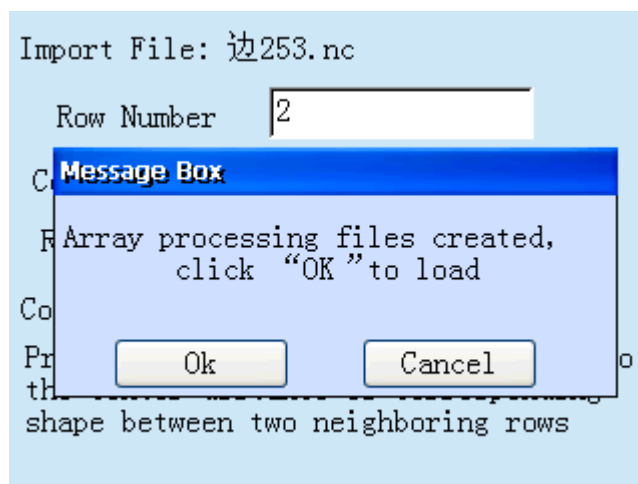
Fig 6-12

Press "OK" to enter internal document list, and then select the generated array to process.

6.4 Speed Rate Setting

Under the idle state, press the combined key "Shift + Mode" in the main interface to enter the Speed Rate Setting interface (as shown in figure 6-10).
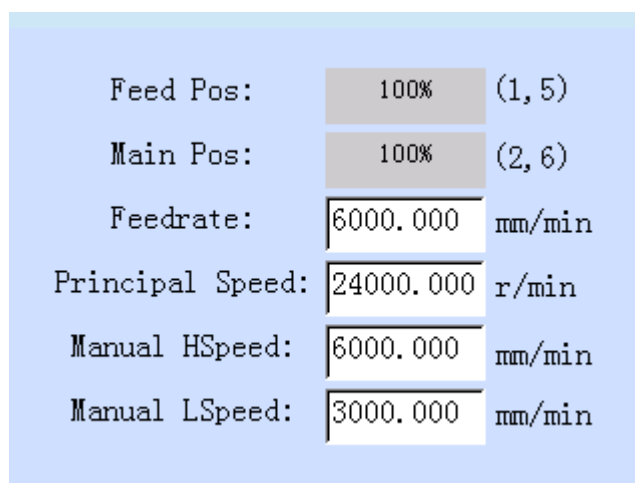
Under the interface, when selecting the rate (the background of rate value viewed in gray) users can press "1", "5" to adjust federate; press "2", "6" to adjust the federate of main shaft. Switching to the selected item to set speed, users can press the number keys to change the value of the speed, and then press "OK" to save and switch to select another item.

Under the machining state, in the main interface, pressing "1", "5" can adjust federate; pressing "2", "6" can adjust the spindle rate.

Chapter 7 How to Operate Milling Bottom

When performing easy milling bottom, operators don't have to compile the G code manually or generate the program file by CAM software. By performing the function of **"Execute Machining Command"** provided by the system, only a few parameters have to be inputted to finish the operation.

```
Rectange Mill
Initial Coordinate: X [0]     Y [0]
            Depth [0]    Layer Depth [0]
            Width [1]         height [1]
Tool Space [1]   Tool Diameter [1]
The Path [0]      0:Horizontal 1:Vertical
Press "OK" to Switch, Press "RUN" to
Machining
```

Fig 7-1 Parameter Window of Round Milling

Milling bottom has the below parameters:

（1）Start Coordinate: X, Y value of specified milling bottom, relative to current workpiece coordinate.

（2）Engraving Depth.

（3）Depth of Each Layer.

（4）Width, means the horizontal distance of workpiece.

（5）Height, means the vertical distance of workpiece.

（6）Distance of Tool Point.

（7）Tool Diameter.

（8）Machining Path, includes horizontal and vertical, using 1 and 2 for representing respectively.

Input the above parameters and then press "Start/Pause" button to start milling bottom machining.

Chapter 8 How to Implement Mirror Rotation Process

Mirror Rotation Process can be selected in advanced processing menu. Users can enter this function through main menu or by clicking "Advanced Start" button to enter advanced processing menu to select Mirror Rotation Process. Click "OK" to enter the selecting window of Mirror Rotation Process, which users can select through "▲", "▼". But this selection is effective currently. Next time when you want to process, you have to select again and click "OK" to auto machining. See as following:

```
◉ No Rotate No Mirror
○ X-axis Mirror Machining
○ Y-axis Mirror Machining
○ Clockwise Rotate 90°
○ Clockwise Rotate 180
○ Anti-clockwise Rotate 90°

Select Type by Up and Down
Keys, Press "OK" to Implement
Mirror Rotate Type Machining
```
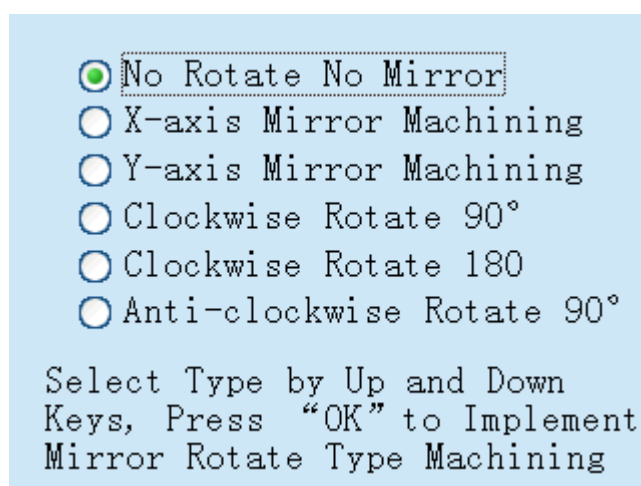
Fig 8-1 Selecting Window of Mirror Rotation Process

Chapter 9 Return to Mechanical Origin

"Mechanical Origin" is a fixed position in the machine tool. It is the zero position of the mechanical coordinate system and determined by the mechanical switch and electrical system together. To execute the **"Return to Mechanical Origin"** function, the machine tool itself needs to be installed the origin switch. If it doesn't have the related hardware support, this function shall be prohibited. See the origin parameter setting in **Chapter 11 Parameter Management**. As mechanical origin is the basic standard of the whole machine tool, thus its main function is to revise the present coordinate. Please execute Return to Mechanical Origin when system starts or stops urgently in order to prevent from power off or wrong current position.

When the system starts, the dialogue box **"Return to Mechanical Origin"** will automatically pop up. Click the button, the corresponding axes will go back to the mechanical origin automatically and revise the coordinate of the system. Before the X/Y axes go back to the origin, please make sure that the Z axis will go back to the mechanical origin first.

Select the item **"Return to Mechanical Origin"**; the system will pop up the dialogue box **"Return to Mechanical Origin"**. See the figure below:
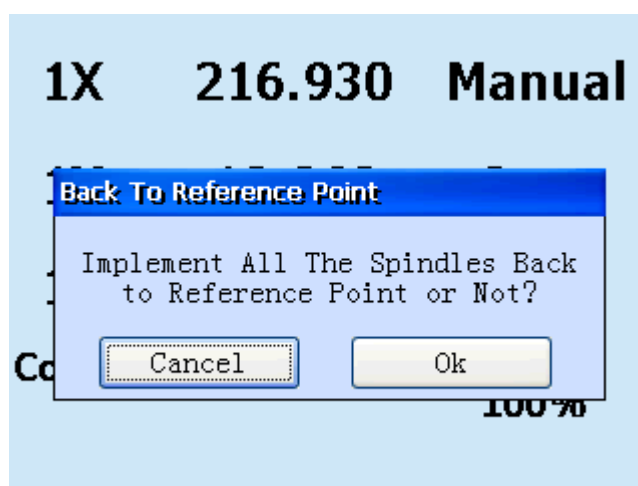

Fig 9-1 Prompt Information of Backing to Reference Point

Sequence of all axes going back to mechanical origin:

Firstly, Z axis return to mechanical origin, and it should stop right now when meeting the hardware limit during the process of returning to origin. After Z axis finding origin, please clear Z axis workpiece coordinate. In the following, X, Y axes will

execute the order of finding origin simultaneously. After X, Y finding the origin, please clear the workpiece coordinate of X, Y axes to finish the process of finding origin.

Note:

If the Explanation **"Return to Mechanical Origin"** is not executed, please try to raise the Z axis when operation manually to make sure that the cutter head won't run into the machining workpiece.

The current coordinate information will be saved automatically when exiting the system. When sudden power failure occurs during the auto machining, the system will save the related information before power failure to the breakpoint protection file (which refers to save the breakpoint information and file name into the system memory when power is failure and each program file is only in correspondence with one breakpoint protection file). When the power is restored, a notification box will pop up, prompting the operators which file was machining when the power was failure. Operators have to perform the **"Return to Mechanical Origin"** operation manually and then resume machining the last file when the power was failure, or select a new program file:

1. If operators want to resume machining the last file when the power was failure, then you can click the button **"Return to Mechanical Origin"**. After returning to mechanical origin, use combined keys "Shift" + "Advanced Start" to enter Resume Breakpoint. After clicking "Start", the machine tool will resume machining the unfinished file seamlessly from the breakpoint.

Chapter 10 Program Management

Press "Menu" button in the main interface to enter the window of main menu. Select the item **"Machining File Management"** to enter "File Operation Window". Use this "File Operation Window" function, you can view, copy and delete file. See the figure below:
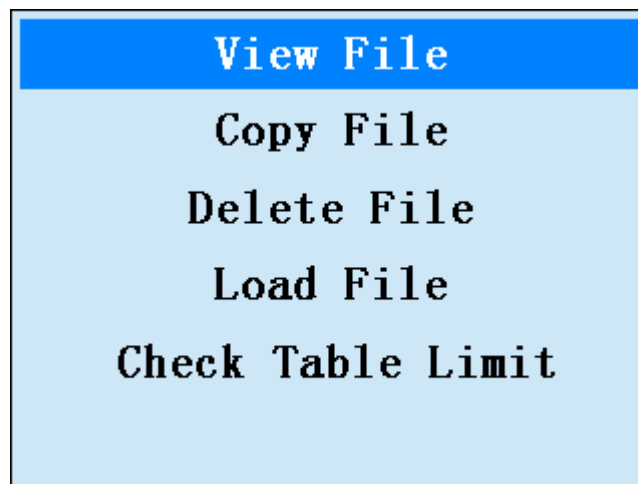

Fig 10-1 File Operation Window

10.1 View File

In the File Operation Window, select **"View File"** to enter the selection interface of file path. You can select U disk or internal file to enter the corresponding memory to select the file you want to view. Select the existing file on the list of U disk or internal, and then click "OK" button to view the file. After viewing the file, users can press "OK" or "Cancel" to exit. See as the following figure:
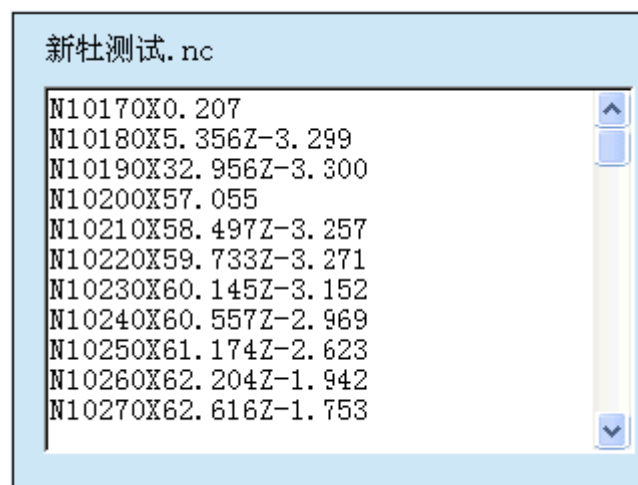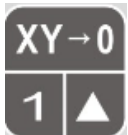

Fig 10-2 View File

While viewing the file, you can press buttons and to view by turning over lines, and press buttons and to view by turning pages.

10.2 Copy File

In the File Operation Window, select **"Copy File"** to enter the selection interface of file path. You can select U disk or internal file to enter the corresponding memory to select the file you want to copy. When select the file list in U disk, if the U disk exist this file, select one file to press "OK" after entering file list in U disk; if the system exist the same file, it will pop up the prompt window (see Fig 10-3).
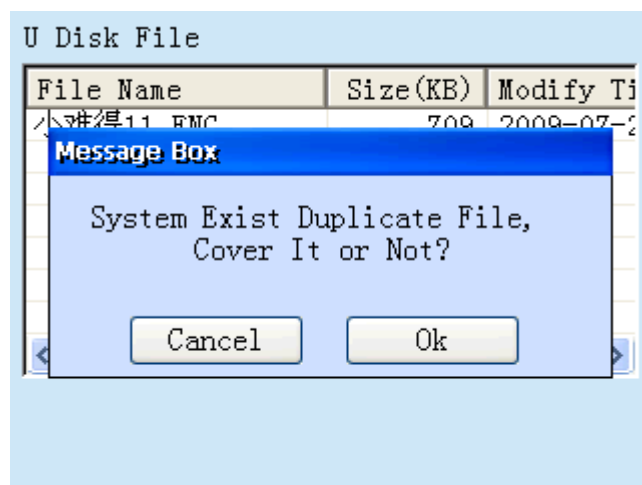

Fig 10-3

Press button "Cancel" to return to list window of U disk. Press "OK" to cover the files with the same names in system and start to copy the selected files to the system (see Fig 10-4). You can press button "Cancel" to stop copy during the process of copy.
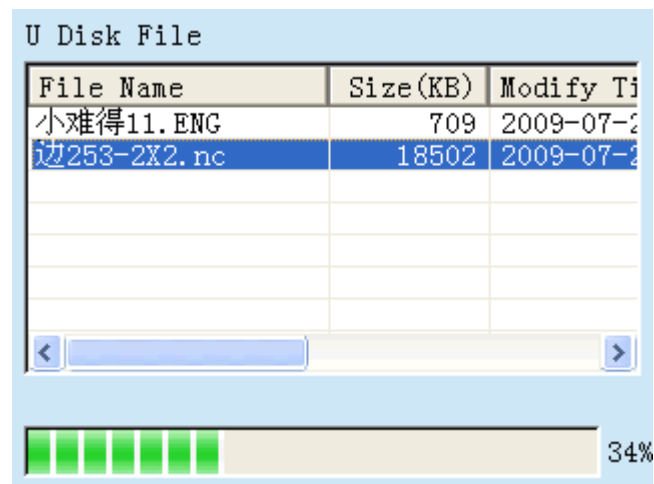
Fig 10-4 Copy File

You can copy the system file to U disk after entering the file list of the system. The detailed operation procedure is the same as copying U disk file to the system.

10.3 Delete

In the File Operation Window, select **"Delete File"** to enter the selection interface of file path. You can select U disk or internal file to enter the corresponding memory to select the file you want to delete. Select the existing file on the list of U disk or internal, and then click "OK" button to delete the file. After deleting the file, users can press button "Cancel" to exit this interface.
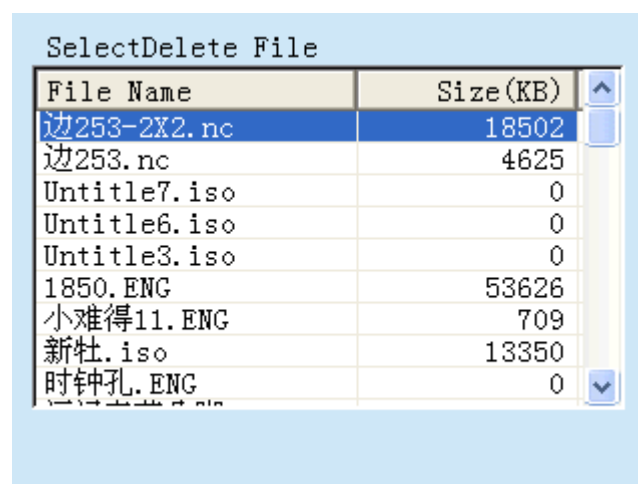

Fig 10-5 Delete File

10.4 Load File

Under file operation interface, select "Load File" to enter the internal document list directly (as shown in figure 10-6). Select the file you want to load, and then press

"OK" to load the file. After loading the file, users can enter information processing interface, press "OK" in the interface to process loaded file. Press button "Cancel" to return to Program Management interface.
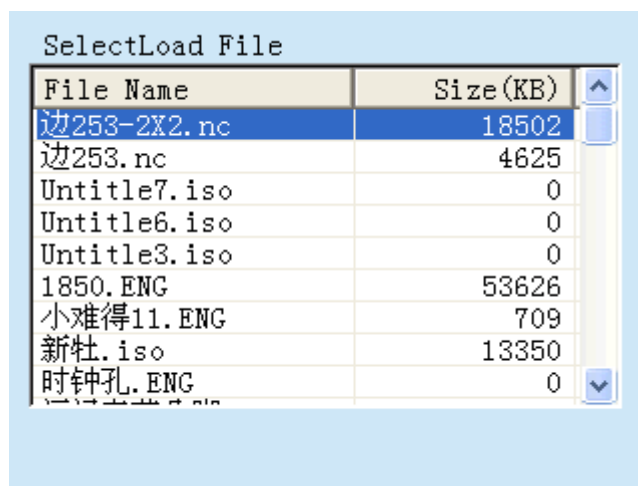

Fig 10-6 Load File

10.5 Check Travel Range

Under the operation interface, select "Check Travel Range" to enter the interface of travel range checking (as shown in figure 10-7).
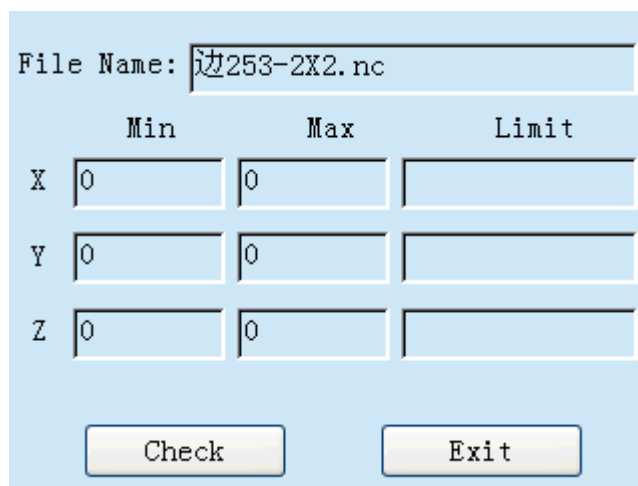
Fig 10-7

When entering the interface, users can see the names of current loaded file. Before starting the inspection, users should check the upper and lower limit of workbench's travel range of different axis. Pressing "OK" to start inspection, at this time, the progress bar shows the inspection schedule. The progress bar will disappear after checking, and display the max. /min. coordinates of X, Y, Z axis in current loaded file. If the min. value of one axis is less than its lower limit of workbench travel range, its

overrun column will show "Negative Overrun"; if the max. value of one axis is more than its upper limit of workbench travel range; its overrun column will show "Positive Overrun". If these two situations happen at the same time, the overrun column of this axis will show "Both Positive and Negative Overrun".

Fig 10-8

Chapter 11 Parameter Management

11.1 Modify Parameter

Press the button "Menu" in the main interface to enter main menu. And then enter 【Machine Tool Parameter】and【Machining Parameter】in the main interface to set parameters. Password is needed when entering "Machine Tool Parameter".

Select "Machine Tool Parameter" in the main menu, password input window will pop up (see Fig 11-1):
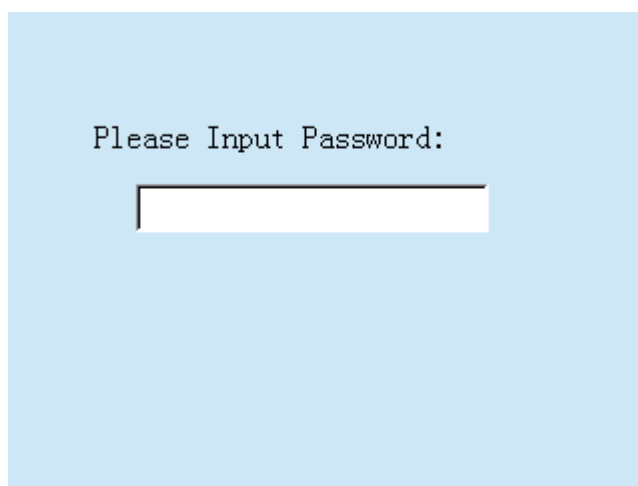

Please Input Password:

Fig 11-1 Enter Password

You can enter Machine Tool Parameter menu after entering the correct password. In the window, you can switch the selected parameters through Up/Down keys. Press button "OK" to enter Modify Parameter window (see Fig 11-2):

Linear Pulse Equivalent

X 0.00100000   mm/p

Y 0.00100000   mm/p

Z 0.00100000   mm/p

Detail: Corresponding Actual Tool
       Travel for Per Pulse From
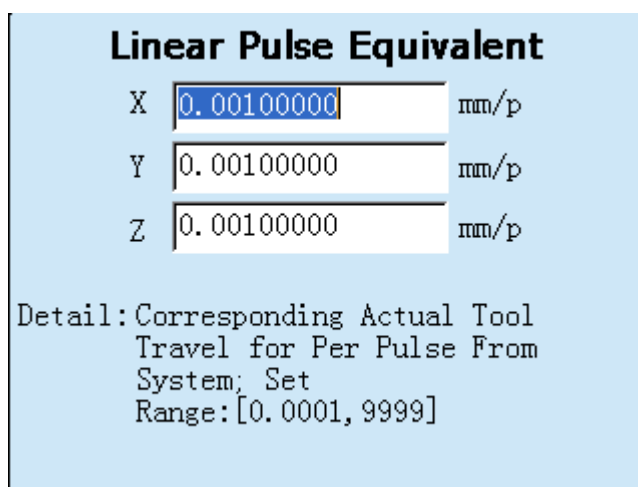   System; Set
       Range: [0.0001, 9999]

Fig 11-2 Pulse Equivalent

Under the interface of Modify Parameter, users can press numeric buttons 0 ~ 9 and

symbol keys key ". ", "-" to change parameters. If the parameter has only one item, after modification users can press button "OK" to save and exit the interface; If there are multiple parameters, press the button "OK" to save the current parameters and switch the cursor to next item. After the modification, users must press the button "OK" to save.

If the input parameter is not legal, or parameter is not in the scope of setting, it will popup dialog, as shown in the figure below:
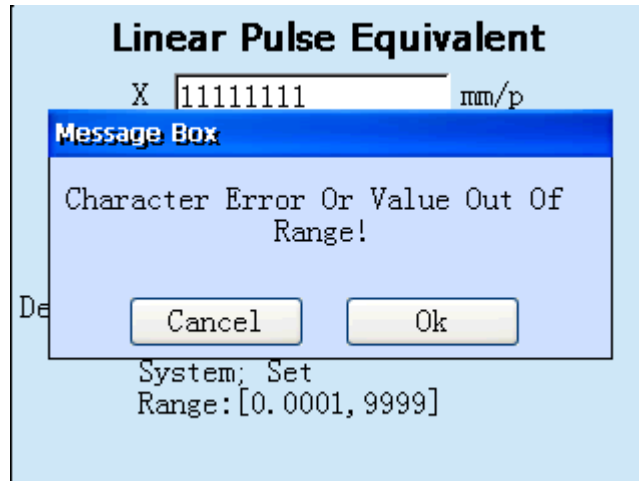

Fig 11-3

11.2 Parameter Backup

Select "System Parameter" under the main menu, and select "Parameter Backup" in the menu of "System Parameter", the pop-up window as follows:
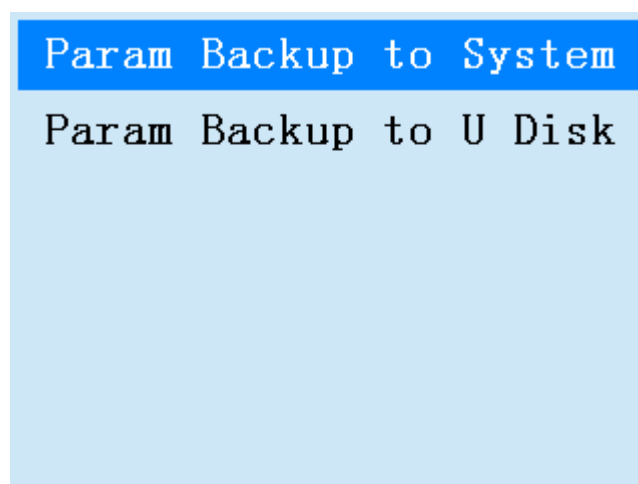

Fig 11-4 Parameter Backup

After selecting the path of backup file, the window as following Fig 11-5 will pop up. This function is used to backup and save all the parameters already set for operators to

inquire in the future time. Input the file name which you have to backup in the window, and click **"OK"** button, then all the parameter value will be saved.
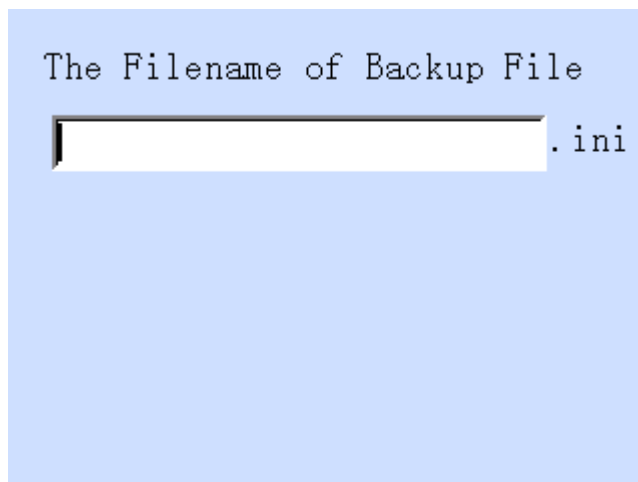

Fig 11-5

11.3 Parameter Recover

Select "System Parameter" under the main menu, and select "Parameter Recover" in the menu of "System Parameter", the pop-up window as follows:
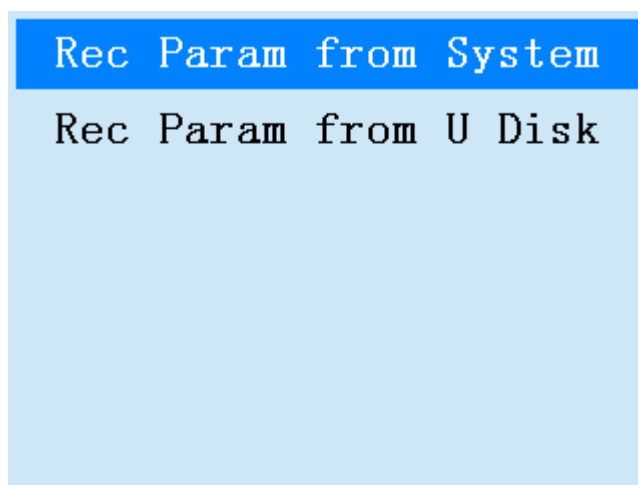

Fig 11-6 Parameter Recover

If users select "Rec Param from System", the window as following Fig 11-7 will pop up. This function can be used to return the parameters to the previous setting value. Select the file whose parameter is to be returned in the pop-up window, and click the **"OK"** button, then the system will restore the parameters to the previous setting value.

Fig 11-7 Parameter List

11.4 Modify Password

Select "System Parameter" under the main menu, and select "Modify Password" in the menu of "System Parameter", the pop-up window as follow. This function is used to modify user's password to effectively protect the private information of the users, thus to effectively protect the security of parameters setting.

Fig 11-8 Modify Password

In the password modification window, enter the old password and press "OK" to confirm the password; and switch the cursor to the new password edit box, enter the new password, press "OK" button to complete the setting.

This system has two parameter categories, i.e. **Mechanical Parameter** and **Machining Parameter**, as it involves a lot of parameters. It needs permission to modify and view one certain parameter.

11.5 Parameter List

11.5.1 Machine Tool Parameter

Parameter	Meaning and Function	Value Range	Effective Time
Movement after machining	Movement of the cutter after machining	0 (Stay still) 1 (Back to fixed point) 2 (Back to workpiece point)	Immediately
Height of cutter lifting when idling traveling	Height of cutter lifting when idling traveling	[1,1000]	Immediately
Pulse equivalent	Corresponding motion distance of machine tool for X/Y/Z axes to impulse sent by driver	[0,1]	Restart
Acceleration at turning	Max. acceleration of feeding motion in adjacent axis	[0.1-9999]	Immediately
Single axis acceleration	Change rate of feeding axis speed	[0.01-100000]	Immediately
Jerk	Change rate of feeding axis acceleration	[0.01-300000]	Immediately
Moving speed of returning to origin	In the process of backing to the mechanical origin, the feed speed of coarse positioning	[Take-off speed, max speed of each axis]	Immediately
Moving direction of returning to origin	Direction of rough positioning phase when back to mechanical origin	-1: X/Y Negative, Z Positive; 1: X/Y Positive, Z Negative	Immediately
Origin limit effective or not		0 (No): Ineffective 1 (Yes): Effective	Immediately
Permitted motion	Permitted motion direction of	0: Both positive and	Immediately

direction when origin limit	X/Y/Z axes when limiting origin	negative motion 1: Only positive motion -1: Only negative motion	
Default rotation speed of main shaft		[0, max. rotation speed of main shaft]	Immediately
Start-up delay of main shaft	Time required for main shaft to start from rest to the rotation speed set in the parameter	[0.5,300]	Immediately
Max. speed of each axis	Setting the max. speed of the main shaft	[0, 300000]	Immediately
Thickness of tool setting block		[0, 500]mm	Immediately
Tool setting speed	Z axis punching rate during fixed tool setting	[Take-off speed -1000]	Immediately
Workbench configuration	Standard configuration or turntable configuration	0: Standard configuration 1: Turntable configuration	Restart
Radius of reference circle		[0,999999]	Immediately
Speed of reference circle		[Take-off speed to max. speed of each axis]	Immediately
Max. speed of arc machining		[Take-off speed, reference circle speed]	Immediately
Rotation axis selection		0：X axis for rotation axis 1：Y axis for rotation axis	Immediately
The max. speed of rotation axis		[0，999999rpm]	Immediately
The max. acceleration of rotation axis		[0，999999deg]	Immediately

Effective level of origin		0: Low level effective 1: High level effective	Restart
Effective level of positive limit	Positive limiting effective for X/Y/Z axes or not	0: Low level effective 1: High level effective	Restart
Effective level of negative limit	Negative limiting effective for X/Y/Z axes or not	0: Low level effective 1: High level effective	Restart
Effective level of emergency stop		0: Low level effective 1: High level effective	Restart

11.5.2 Machining Parameter

Parameter	Meaning and Function	Value Range	Effective Time
Idling Speed	Speed of cutter when idling	[Take-off speed to max. speed of each axis]	Immediately
Safety height	System consider horizontal motion at this height safe	[5,500]	Immediately
Default feeding speed	Default feeding speed of system	[Take-off speed to max. speed of each axis]	Immediately
Adopt default feeding speed	Speed specified in the file ineffective when adopting default feeding speed	1 Adopt default feeding speed 0 Adopt specified speed in the file	Immediately
Diameter of rotary workpiece	Diameter of machining workpiece	[0-99999]	Immediately
Cutter lifting capacity of Z axis when pausing	Height of cutter lifting upward along Z direction when cutter pausing	[1,1000]	Immediately
Cutter change notification effective	Whether system prompts cutter change when there are cutter	0 (No): Ineffective 1 (Yes): Effective	Immediately

	change phrases in machining Explanations		
Adopt default main shaft speed or not	Specified feeding speed in program file ineffective when adopting default main shaft rotation speed of system	0 (No): Ineffective 1 (Yes): Effective	Immediately
Movement after machining	Movement of the cutter after machining	0 (Stay still) 1 (Back to fixed point) 2 (Back to workpiece point)	Immediately
Use ENG file to select tool for machining	Machining according to the specified tool	0：Not select tool for machining 1：Select tool for machining	Immediately
Back to mechanical origin before machining	Set if go back to mechanical origin before each machining	0 (No): Ineffective 1 (Yes): Effective	Immediately
Whether lift to safety height or not when pausing		0（No）：Lift to specified height 1：（Yes）：Lift to safety height	Immediately

Chapter 12 Auxiliary Management

12.1 System Diagnosis

System diagnosis includes functions like Key Detection, System Log, Version Information, IO Status, and other functions.

After entering the main menu, please select system parameters, and then enter the system parameter menu. As shown in the figure below:
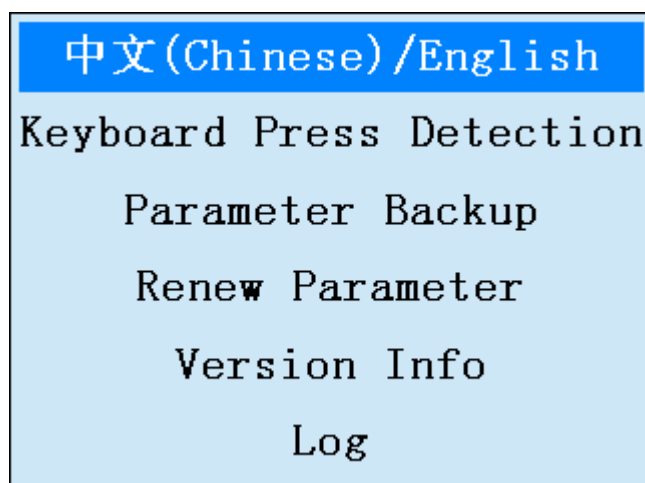
Fig 12-1 System Parameter

12.1.1 Key Detection

Through up and down key users can select Key Detection to enter this function. Just like the page of Key Detection shown in the following Figure 12-2. Users can press the key "CANCEL" twice to quit.
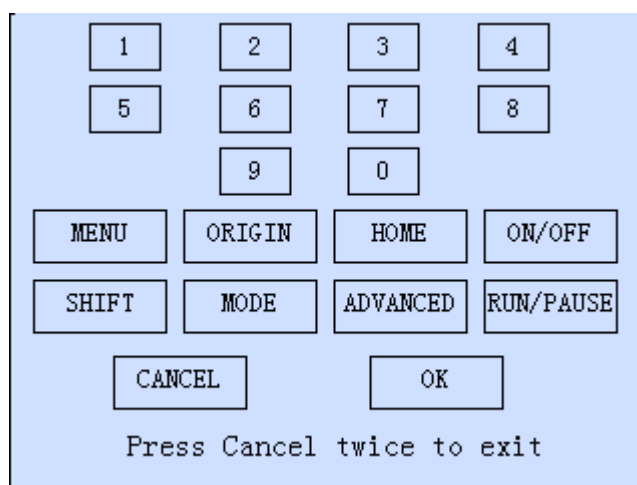
Fig 12-2 Key Detection

12.1.2 System Log

System log page, as shown in the figure below:

Fig 12-3 System Log Page

System record log information including:

(1) Information of automatically start/end machining;

(2) The change of workpiece coordinates;

(3) The system alarm information;

(4) The file processing complete information;

(5) Other system information.

Note:

Users need to remove system log periodically. Click "Delete" button can be directly empty the log information. Too big system log will easily cause system running slowly.

12.1.3 System Version

System version information page, as shown in the figure below:
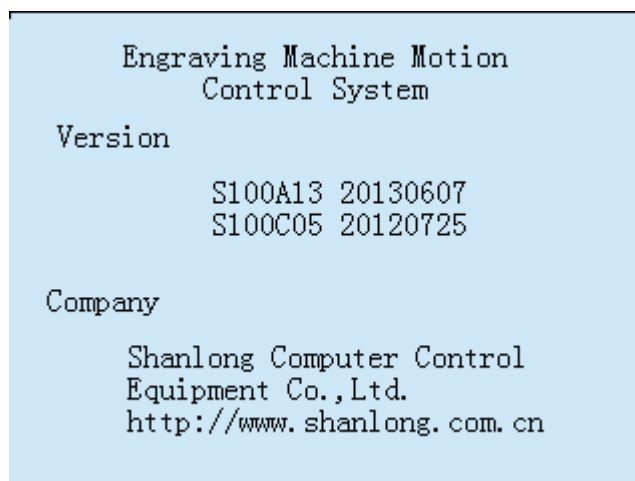
Fig 12-4 System Version

12.1.4 IO Status

Below is the IO status page. It shows the current status of system IO. User can obtain system IO status through this page.

Fig 12-5 IO Status Page

12.2 Software Update

12.2.1 Chinese/English Interface

The function can change the system language; support switching between English/Chinese. (Currently not support)

12.2.2 System Software Update

Click to enter "Software Update" module.

Firstly, the system will check whether there is an external hard driver device (U disk) for system update. If there is no device, the system will exit directly after the prompt.

If there is an external hard driver device (U disk), it will indicate whether to backup the system parameter as shown in figure 12-6. The system will pop-up backup parameter page to prompt to input backup file names. Click "OK" to enter "Select Update File Page" as shown in figure 12-7. After selecting the file, click "OK" to enter upgrade page. The system update is divided into two stages. The first stage is data copy. Do not pull out peripheral devices or power off in the process; otherwise the system update will fail and the system is still the old version again. The second stage is burning program. At this time, pull out upgrade peripheral devices like U disk will not generate errors. But we suggest not pulling out. In the process of updating, power off is prohibited. If users power off before system update finished, the system will be damaged and cannot be started and with black screen. If this phenomenon occurred, please return it to the factory for repairing.
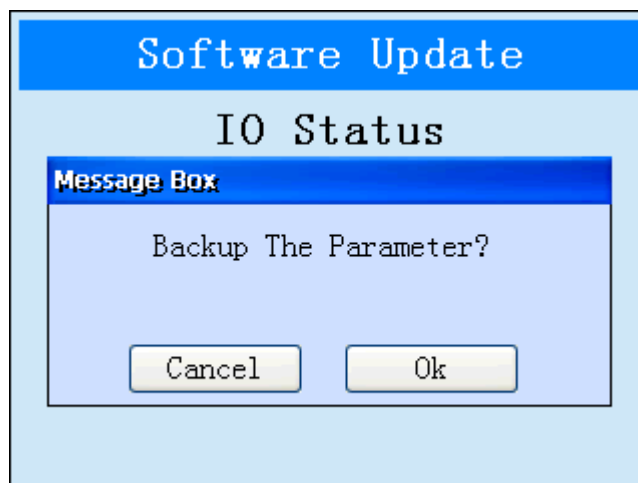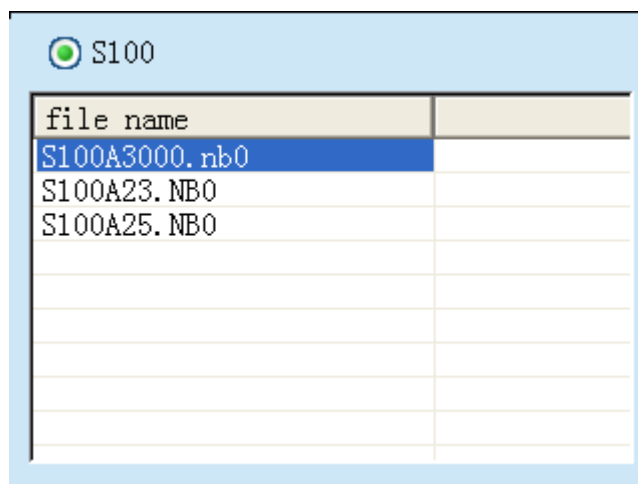
Fig 12-6 Prompt Backup Parameter
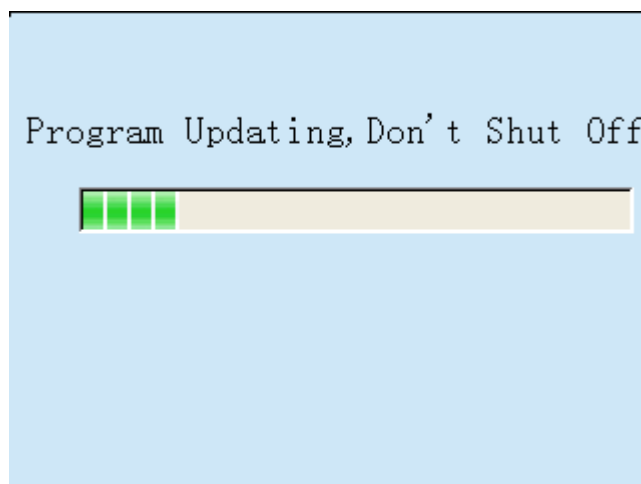
Fig 12-7 Select Upgrade File
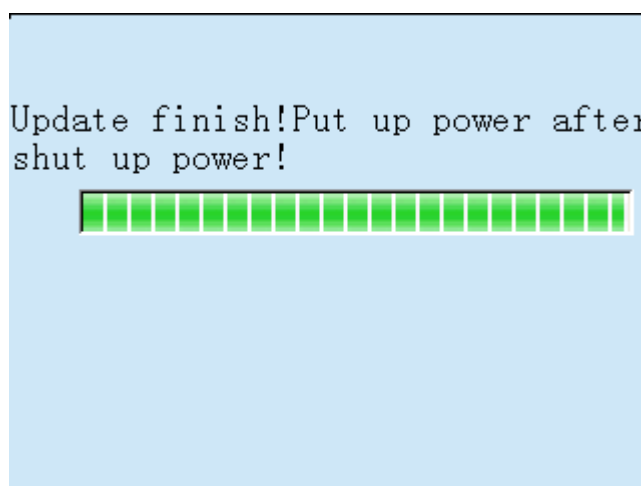
Fig 12-8 Upgrade Progress

Fig 12-9 Update Finish

12.3 Upload Start Logo

Under the System Parameter menu, enter "Upload Start Logo". It will pop-up password dialog. Entering the correct password, users can select the list of start logo picture (as shown in figure 12-10)
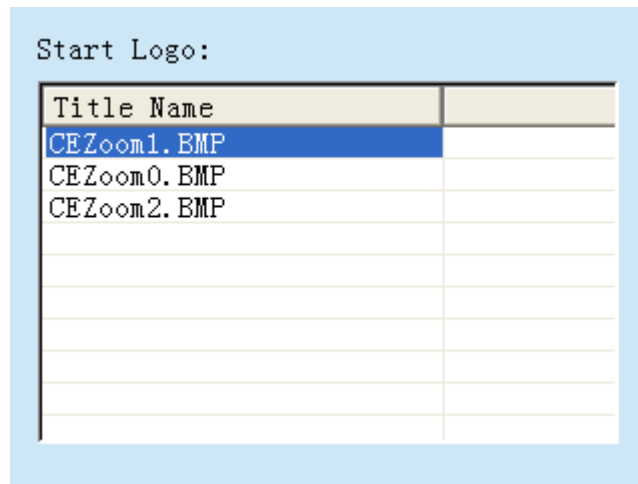
Fig 12-11

Select the bitmap Logo users want upload, and then press "OK" button to load and pop-up the progress dialog (figure 12-12)
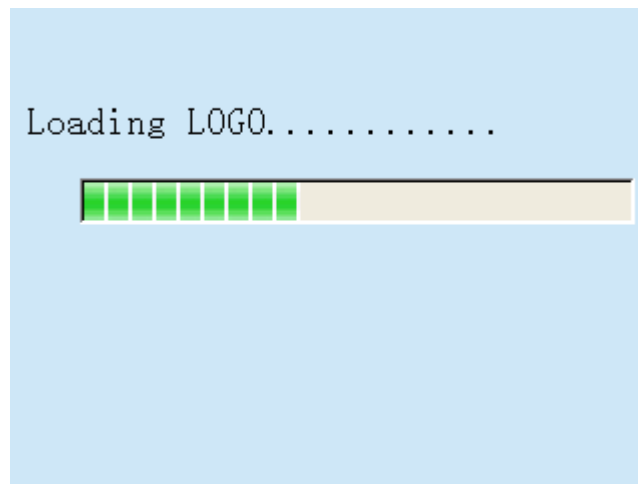

Fig 12-12

12.4 Instalment

There are two ways for manufacturers to encrypt the software: one is electricity time encryption. Under this encryption mode, the system began to calculate using time as long as you power on. The system can only use instalment function when the system is on expired. The other way is machining time encryption. Under this encryption mode, the using time only will be calculated when users start machining. When it expires, users can't machine any files but can still use other function. Following take electric time encryption as an example to illustrate the usage of installment function.

When start the system, if the system pop-up dialog box shown in figure 12-13, it means that the using time of the software is less than 48 hours. To avoid unnecessary
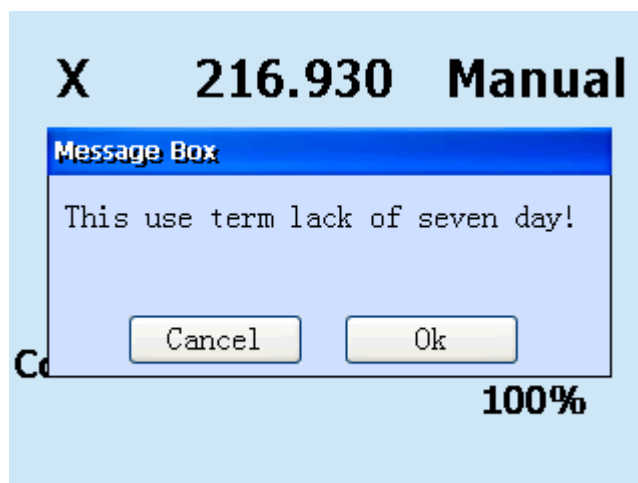
loss, please get in touch with manufacturer.

Fig 12-13

After getting a new password, users can open the main menu to enter the "System Parameter", and then click "Installment" to enter the installment interface (as shown in figure 12-14).
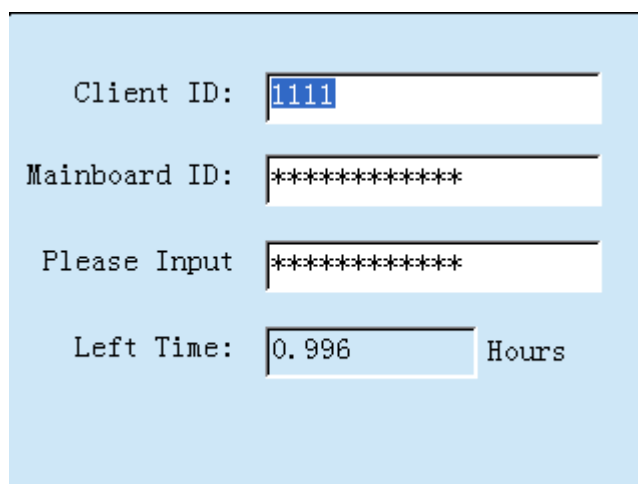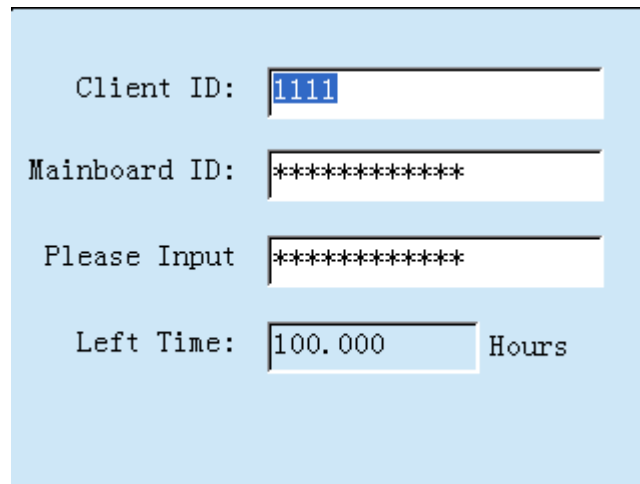

Fig 12-14

Under this interface, users can see the customer ID and the remaining time of software. Press "OK" to select "Please input password" item. Input the new password got from manufacturer, and then click "OK". The remaining time of using will be displayed on the screen. For example, figure 12-14 the remaining time is less than 1 hour, after inputting a new password (as shown in figure 12-15), the remaining time is 100 hours. It means that the new using time period has taken effect.

Fig 12-15

Tip: In the process, no matter which kinds of encryption methods is used, if the using time expires, the machining action will be forced to terminate.