

Water Jet Cutting Control Software(XLNC6.xZ)

Operating Instruction

Contents

Chapter 1 System Summarize

Chapter 2 Program Mode

Chapter 3 Emulation Mode

Chapter 4 JOG Mode

Chapter 5 Automatic Mode

Chapter 6 Set Parameters

Chapter 7 Diagnose Mode

Chapter 8 Quit System

Chapter 9 Common Error and Solution

Chapter 10 Common G Code

Chapter 11 Basic Operation Process

Chapter 1 System Summarize

§1.1 Software Characteristics

- It can control the stepper motor and AC servo motor
- The WINDOWS Version of Chinese language with good human-computer interface
- Supporting the international standard of ISO code and DXF files produced by Auto CAD with auto series connection.
- static and dynamic simulate function, check up automatically the phrase mistake for NC code.
- Easy to operate for manual-auto processing, pause, continue, back off processing, single segment, etc.
- double limit mode of soft limit and hard limit for safe operation.
- It has the function to recovery the coordinate after the breakpoint.
- With diagnosis and real time tips in Chinese language for various fault operation.

§1.2 Function of Software

1. The whole software interface of CNC system is designed according to the requirement of professional CNC system. It is convenient to operating.
2. Stand for DXF documents and NC code automatically generated by any programming software.

It needn't to define the high pressure, air valve, sand valve and oil pump in NC code. They are all automatically controlled by control software according to the shape.

The DXF document generated by AUTOCAD software needn't any other conversion software. It can be loaded directly by CNC control software, which will series connect the shape of DXF document.

3. Forward and backward processing function

It can cut for forward or backward optionally during processing without shutting off the high pressure, which avoids the influence on the processed shape. It can continue to process for Forward or backward after pause

4. The on/off precedence of high pressure, air valve, sand valve and the switch delay can be easily set by parameter.
5. uniform control function

The change of the speed has great effect on quality of the products processed by high-pressure water. For cutting line or arc, the control system can automatically adjust the cutting speed of each axis to make the Synthesis speed equal to the required processing speed, which can ensure the processing quality of the products.

6. Two modes of single stage processing and smooth processing

Under the mode of the single stage processing, the code of each line speed up and slow down separately. Under the mode of the smooth processing, it will connect several parts of adjacent small lines and arc for speed decreasing and increasing together, according to the change of the shape automatically. The

influence on the products caused by speed fluctuation will be decreased, which will make the contour of cutting products smoother and level up the processing quality.

7. Automatically tackle with the small arc speed

The users set the radius limit and the processing speed of the arc by themselves. During processing, the control system automatically estimates the size of the radius of the arc. If the circular is over the arc limit, it will use the fast processing speed. If the arc do not reach the arc limit, it will process according to the set low speed. It can both ensure the processing efficiency and the accuracy of the small arc.

8. Circulating Processing Function

When we process a product for a large number, we can set the circulation times. It can continue the whole procedure from the beginning to the end of processing until it finish all the times. Avoid of putting down bottom frequently to improve the processing efficiency.

9. Coordinate recovery function after power off

With machine tool original function, it can remember automatically the Coordinate of the last ten times. After power off it can also automatically recover the work Coordinate.

10. Cutting head adjustment function during processing

During processing, it is easy to adjust the position of cutting head manually to control the distance from the cutting head to the work piece.

11. It can separate the up and down speed parameter of the fast moving speed and processing speed to easily adjust up and down speed parameter of the processing which can be also supplied when the cutting speed is lower. It ensures the cutting quality of corner to the requirement.

12. Easy to pause and continue or process at any part

During the processing, it needn't return to the origin manually move the machine tool or the MDI motion machine at any place

13. Figure scaling and rotation function

It can scale the figure at any ratio and process ant any angle.

§1.3 PC Requirement

- As for P3 computer, it can only support the Windows 98 but the Windows 2000 and Windows XP.
- As for P4 computer, it can support Windows 98, Windows 2000 and Windows XP.

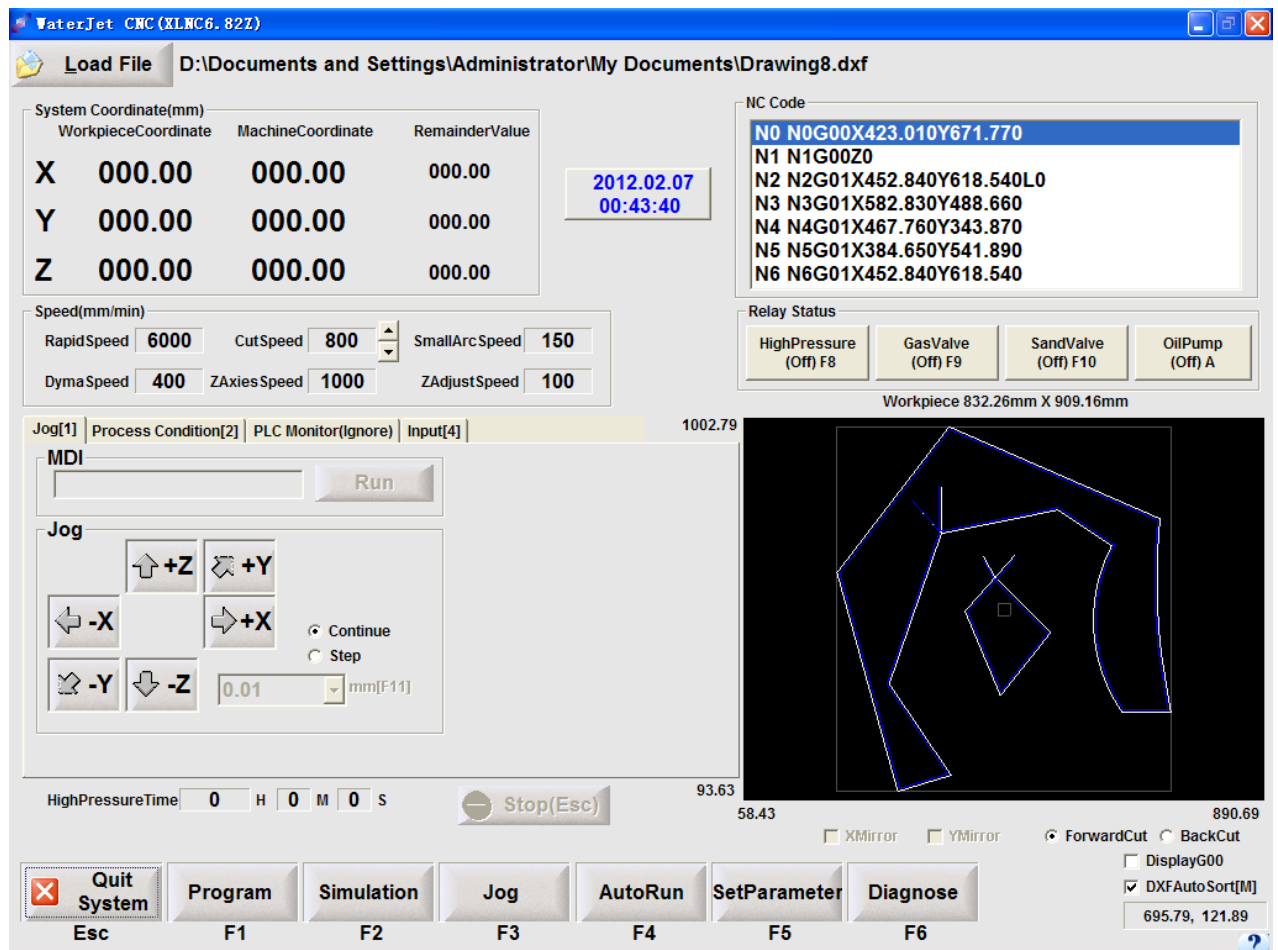
§1.4 Hardware Requirement

- As for P3 computer, the master frequency of CPU is larger than 400M; the memory capacity is above 128M; the hard disk is above 40G.
- As for P4 computer, the master frequency of CPU is larger than 1.5G; the

memory capacity is above 235M; the hard disk is above 80G.

The control system includes two kinds control card: ISA control card and PCI control card. The ISA control card can only support the Industrial Control Computer. PCI control card can support the Industrial Control Computer and usual PC.

§1.5 Shortcuts Key Operating Instruction



Picture1.1 Main Menu

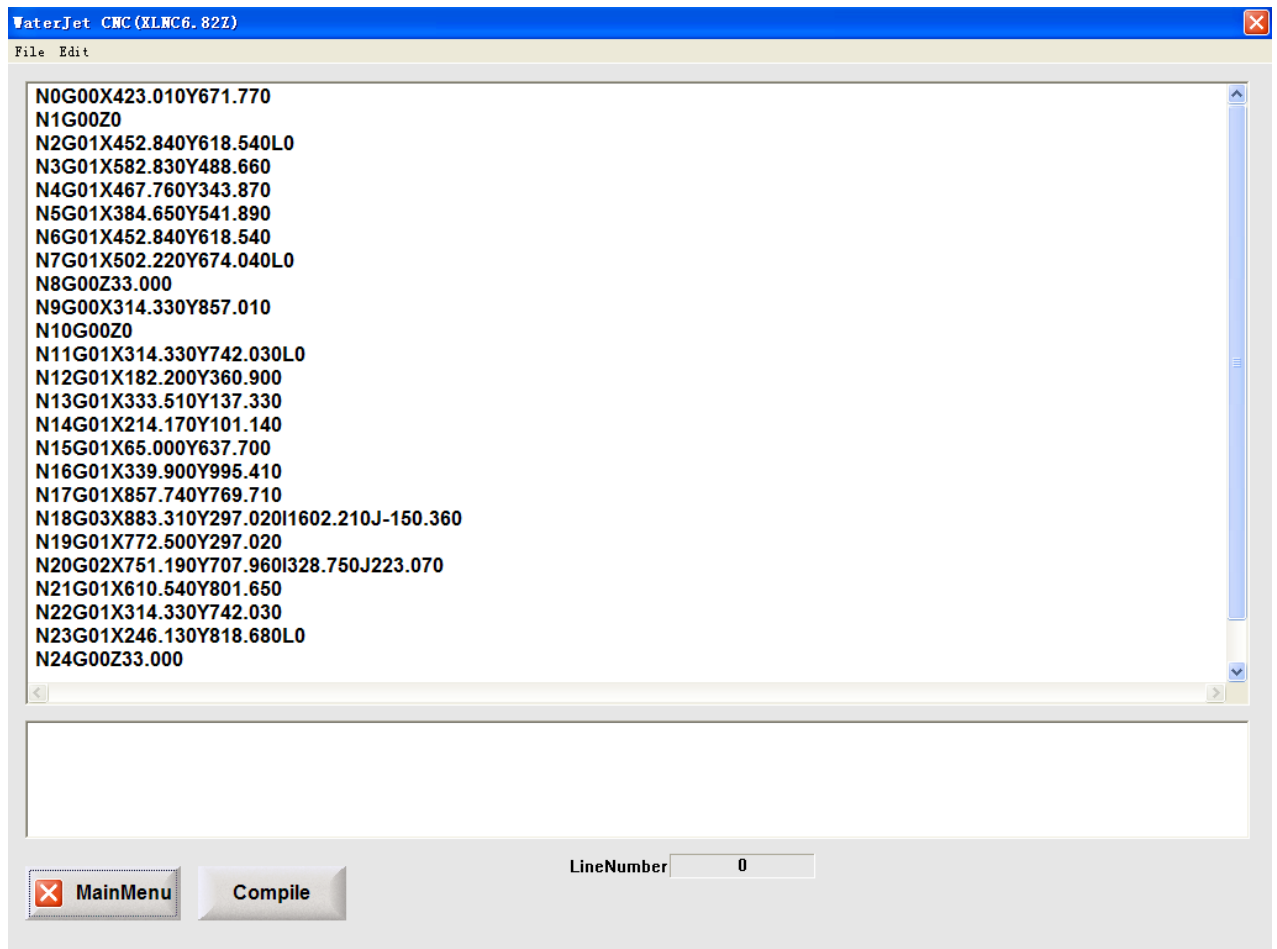
1. press ←、→ key to choose menu and press ENTER key or the hot key F1, F2 under the menu to start the correlation function.
2. When the NUM light on the keypad, press the ←: -X →: +X : +Y : -Y PgUp: +Z PgDn: -Z on the keypad.
3. Relay controls the F8: high pressure F9: gas valve; F10: sand valve
4. The letter in the parenthesis behind all the option means the shortcut key, for example put down M on the left-hand corner to elect the item.
5. Put up and down bottom can adjust the processing speed.
6. Put **P** to return zero point in work coordinate for X and Y axis, and return safe

height for Z axis.

7. Put down O to return the left-hand corner of the current NC code procedure.

8. **Esc** can control both returning and stopping.

Chapter 2 Program Mode



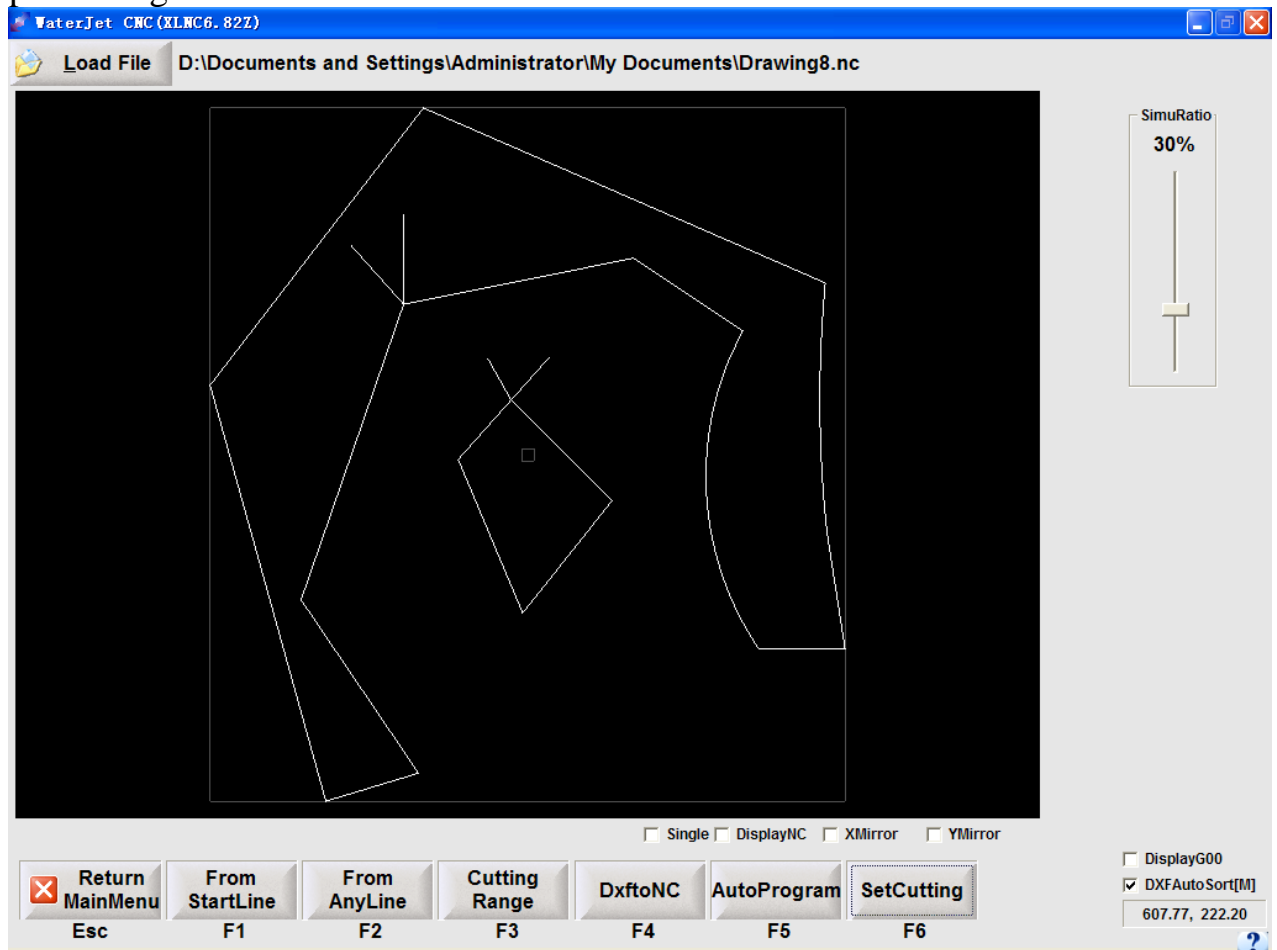
Picture 2.1

Press the F1 or select “edit” under the main menu. If the NC code is small, it will automatically open current NC code. As the picture 2.1 showed, we can edit, modify, reserve, and compile the NC code with this function. If there is error in the NC code, the information will appear in the box below. Select the fault message, then the line number of error will automatically enter the current edit box. If the NC code is over 100K, the notebook will open automatically.

Chapter 3 Emulation Mode

Under the main menu, press F2 to enter the emulation mode. It can take an emulation procession in the computer according to the programmed procedure. It can directly observe the cutting trace in processing. The emulation can be taken by NC file or DXF file.

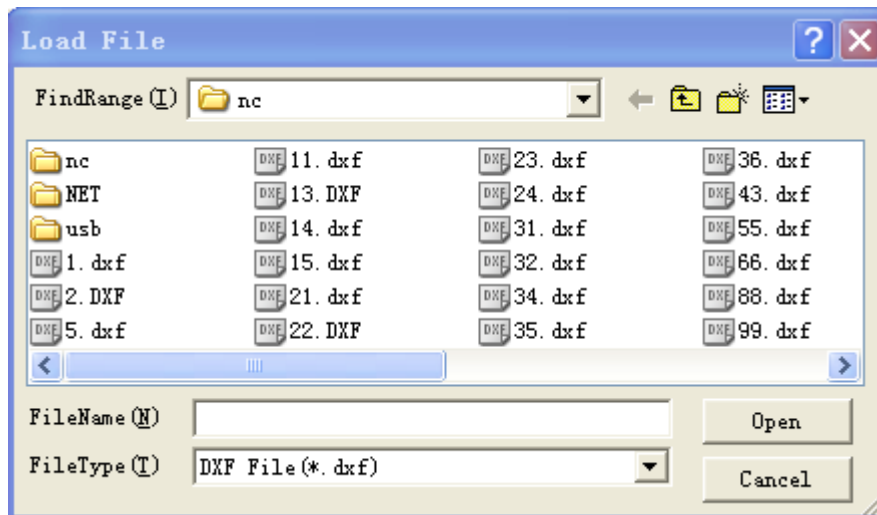
During emulation you can press Esc key at any time to break the emulation. As long as the drawing begins to be showed, the software will automatically accumulate the max and min value in NC code after emulation. Then the value will be displayed in the menu of 'procedure information' in order to knowing the processing information



Picture 3.1

§3.1 Load the File Name

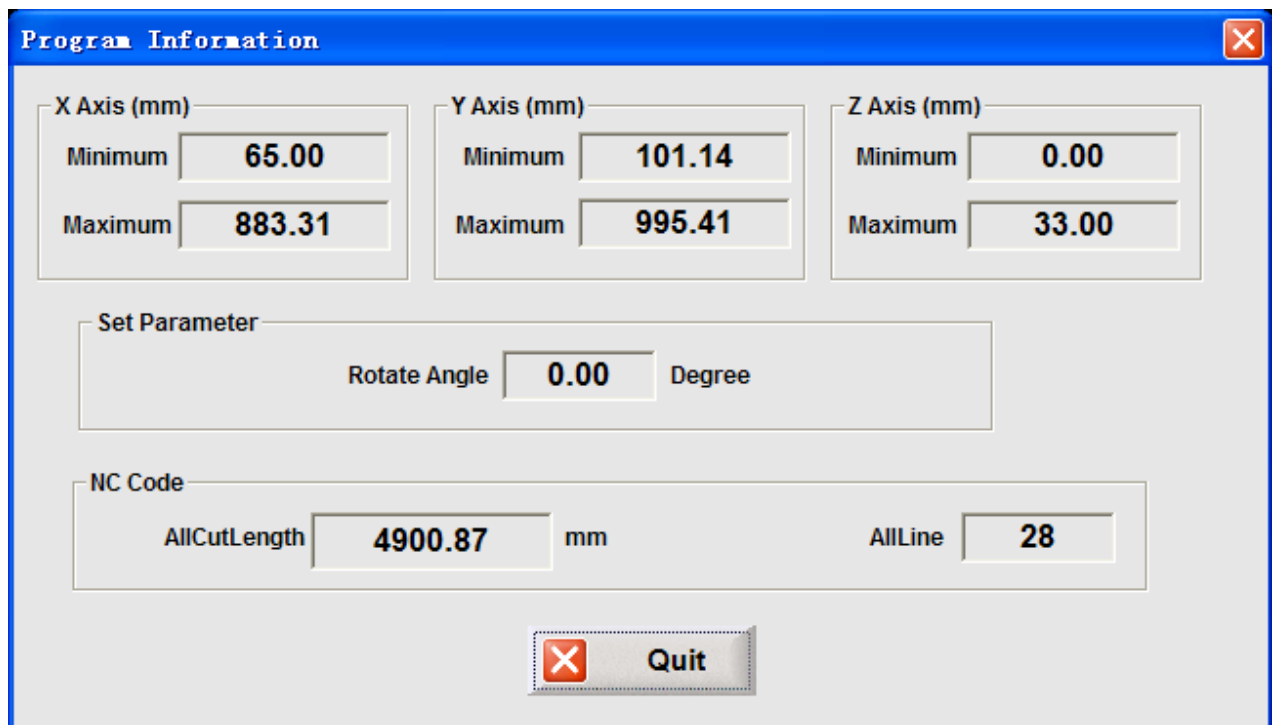
The function can be operated in main menu and emulation menu. Press shortcut key 'L' or click with mouse. Then the dialog of opening file will appear as picture 3.2 showed for selecting DXF and NC file. The emulation and procession are all the operation to such file name.



Picture 3.2

§3.2 Procedure Information

Select 'Procedure Information' after the emulation or auto procession of called procedure. At this time it displays the max and min value in size, processing ratio, angle of rotation, total cutting length, total lines in the procession of NC code for setting current point coordinate and setting limit easily. Please look picture 3.3 as reference.



Picture 3.3 Procedure Information

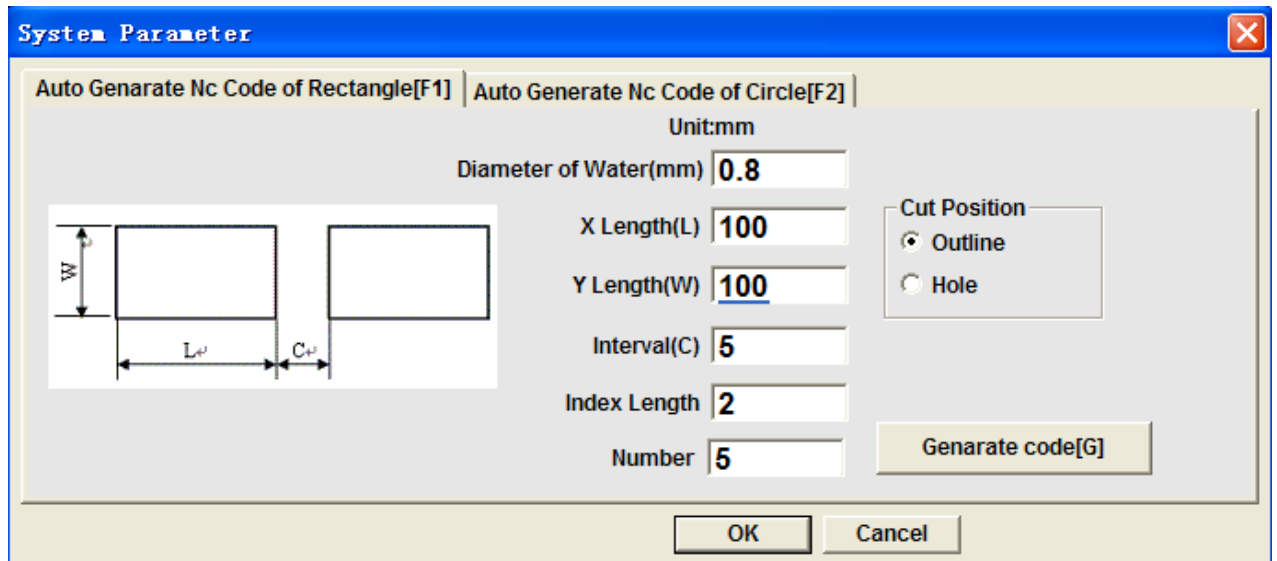
§3.3 Transfer Code

This function transfer the DXF files from AUTOCAD or other software into CNC procession NC code and automatically generate the NC code with same name

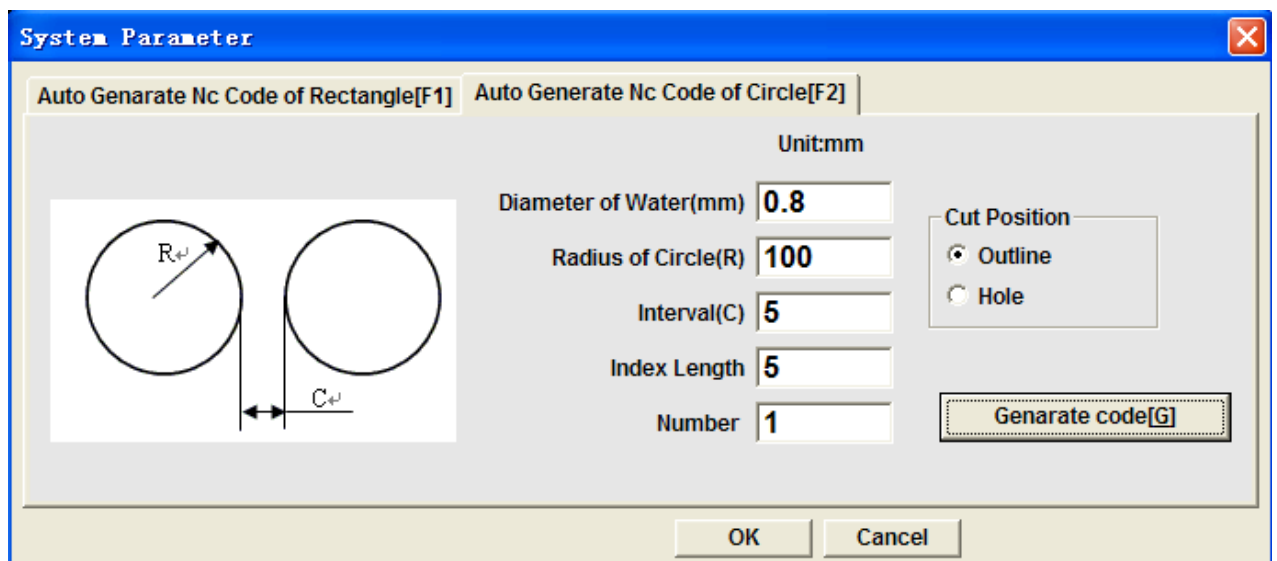
under the directory of DXF file. Note:

§3.4 Auto Program

The auto programming function includes auto rectangular forming and auto arc forming. Look at picture 3.4 and 3.5 separately. The processing code will be generated automatically after clicking the 'generate processing code' and inputting the file name. The generated processing code is with the offset for a radius of water column. As to the cutting position, to select 'outer outline' for shift towards outside and select 'inner hole' for shift towards inside.



Picture 3.4

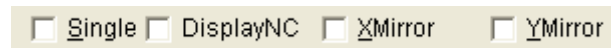


Picture 3.5

§3.5 Multi Option

The picture 3.6 can only be selected in emulation. The function of each option is below:

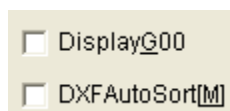
1. Single segment: the shortcut key is 'S'. If to select 'single segment' it will emulate from beginning or any segment. Emulate only one line of NC code for pressing one button. If it is curve the emulation will take place once for the whole curve. If not, the emulation will be continuous.
2. Procedure Display: the shortcut key is 'D'. If to select this the term the procession of emulation will display the processing code of each segment. However, the display of code will slow down the emulation.
3. X-axis mirror image: the shortcut key is 'X'. If to select this term the NC code will make mirror image based on X-axis. The original positive value of Y in NC changes into the negative and the original negative value changes into the positive. This option is also effective to auto process.
4. Y-axis mirror image: the shortcut key is 'Y'. If to select this term the NC code will make mirror image based on X-axis. The original positive value of X in NC code changes into negative and the original negative value changes into the positive.



Picture 3.6

The picture 3.8 can be used for emulation function or under the main menu. The function is as following:

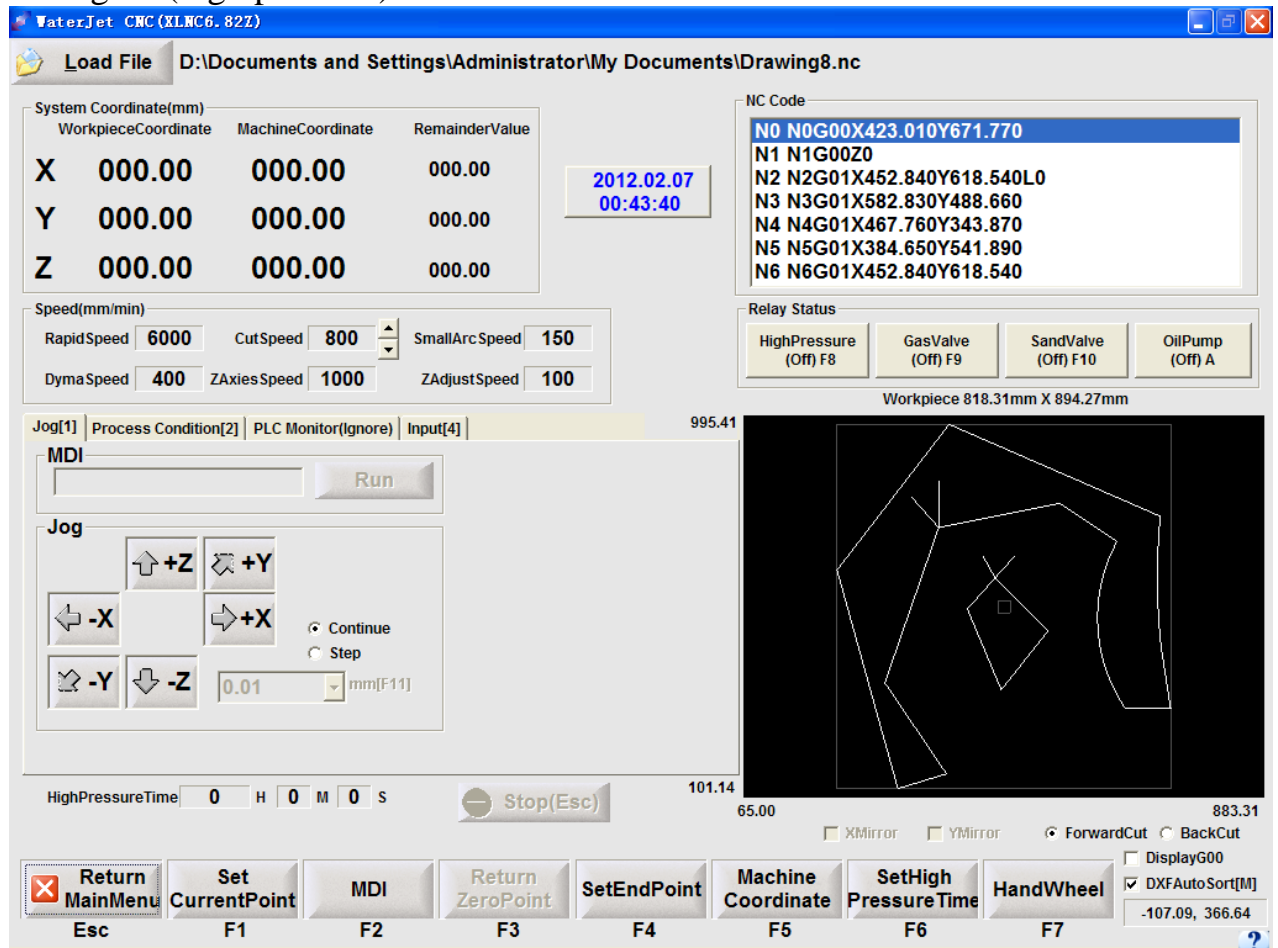
5. Rapid running line display: the shortcut key is 'G'. If to select this term, G00 code will be displayed by yellow line. To select this for seeing the processing order of the drawing in DXF file or NC code clearly. If not the G00 line will not be displayed in emulated drawing.
6. DXF auto series: the shortcut key is 'M'. If to select this term control software will take auto series connection to the drawing of DXF file. That is to say it will automatically and orderly find the next segment connecting with it from the beginning. Due to the feeding line and exiting line usually in a close drawing, for easily automatic series connection when save the DXF file please save the feeding line first, click the first line of the close drawing (this operation is for ensuring the direction of the close drawing), then select other lines of the close drawing, at last select the exiting line. Note: to draw the feeding line from the outer to the inner. If not to select this term it will process according to the order of DXF file.



Picture 3.7

Chapter 4 JOG Mode

Press the F3 key or select the JOG function under the main menu to enter picture 4.1. There are six functions, such as setting current position, MDI, returning to the origin, setting the position of end point, setting the coordinate of machine tool, setting HP(high pressure) time.



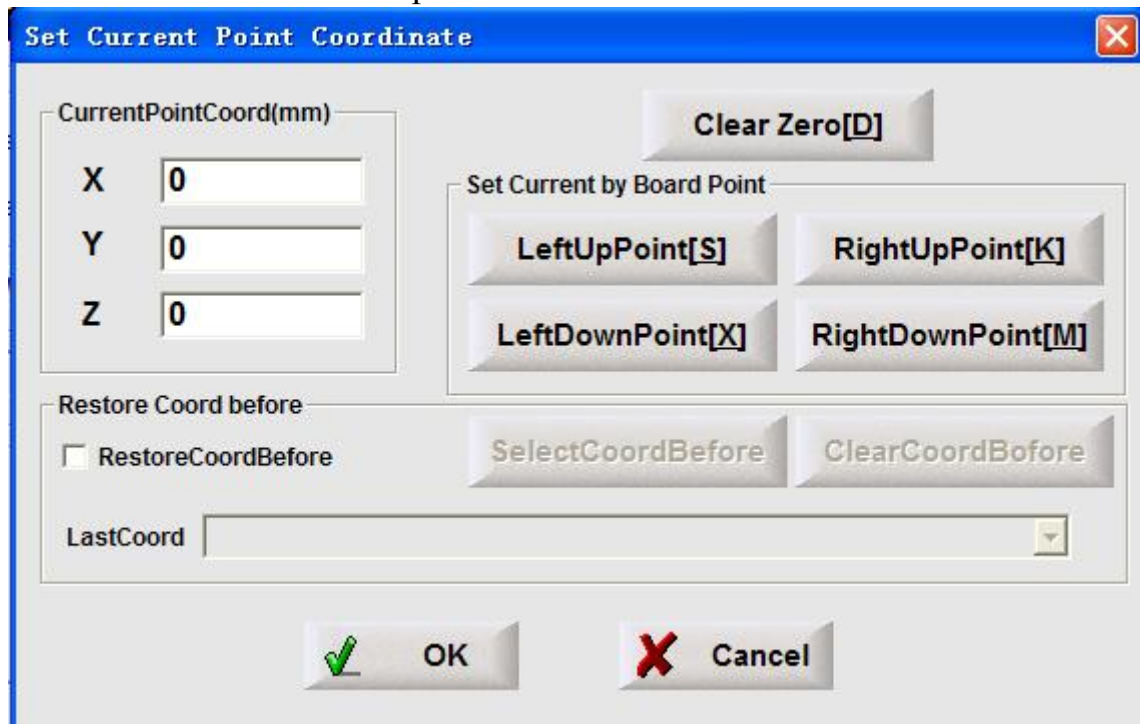
Picture 4.1

§4.1 Current Position

This function is used to set the work coordinate, saying, programming coordinate. Generally, at first, move the cutting tool to the programming original point. Check up the X, Y axis. Open the 'current setting'. Then the dialog as picture 4.2 will appear. There are seven ways to set current position as following:

1. Directly input the current coordinate
2. Select 'reset' to reset X-axis and Y-axis automatically;
3. Set the current position to the left bottom corner of program;
4. Set the current position to the left upper corner of program;
5. Set the current position to the right bottom, corner of program;
6. Set the current position to the right upper corner of program;
7. Select 'recover pre-coordinate' and select 'select pre-coordinate', then the

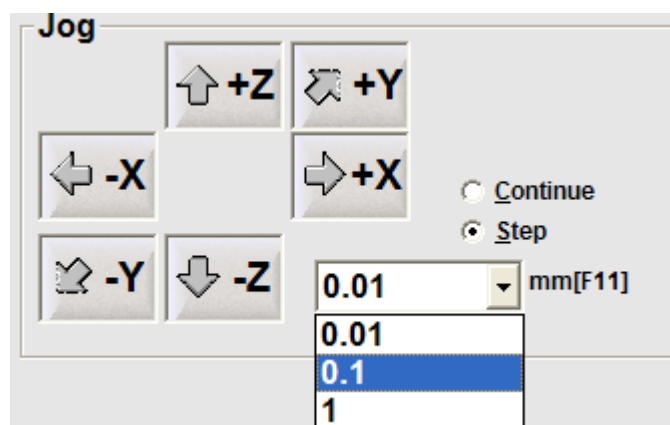
software will automatically take the first item in the 'pre-coordinate' list, saying, the last working coordinate as the current position. This function is for automatically recovering the last coordinate. There are 10 times work coordinates in the 'pre-coordinate' list. The software will reset all the last 10 times coordinates if to select 'reset pre-coordinate'.



Picture 4.2

§4.2 JOG Function

The JOG button is as the picture 4.3 showed. As long as the JOG button is on, the machine tool can be operated to move. Select 'continue' and press the correspondent jog key or select the correspondent button use mouse, then the machine tool will move continually. Due to the high speed of jog, select 'single step' to accurately move. At this time we can press the correspondent button once from 0.01mm, 0.1mm or 1mm of the list, then the machine tool will move the selected length.

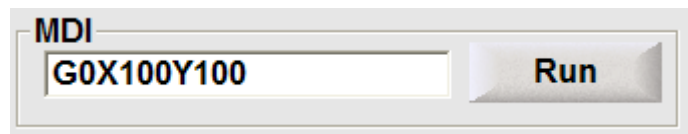


Picture 4.3

§4.3 MDI Function

Enter a line NC code to process under the MDI mode but processing the outline described by more than one program. This function is the most suitable for manual simple localization. As picture 4.4 showed, after entering the G code press the 'enter' or click the 'run' for running.

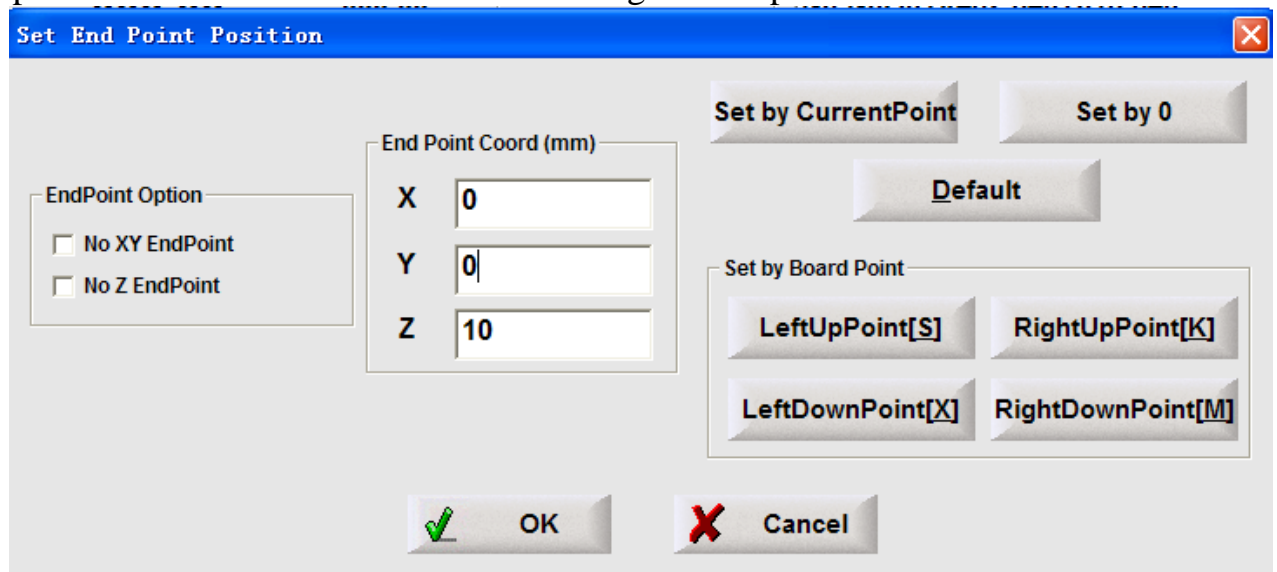
For example, move the machine tool to the (0, 0) and input G00X0Y0.



Picture 4.4 MDI Function

§4.4 set end point position

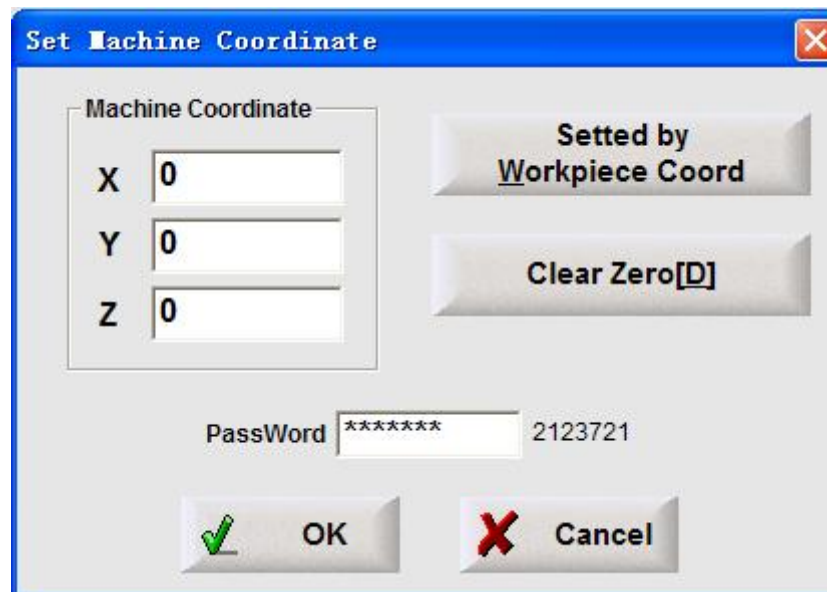
This function is for setting the stop position of cutting head after finishing all the processing code. If to select 'XY without over point', the X-axis and Y-axis position of over point will be decided by the processing code. The method of setting end point is as the same as the one of selecting current position.



Picture 4.5

§4.5 Set the Coordinate of Machine Tool

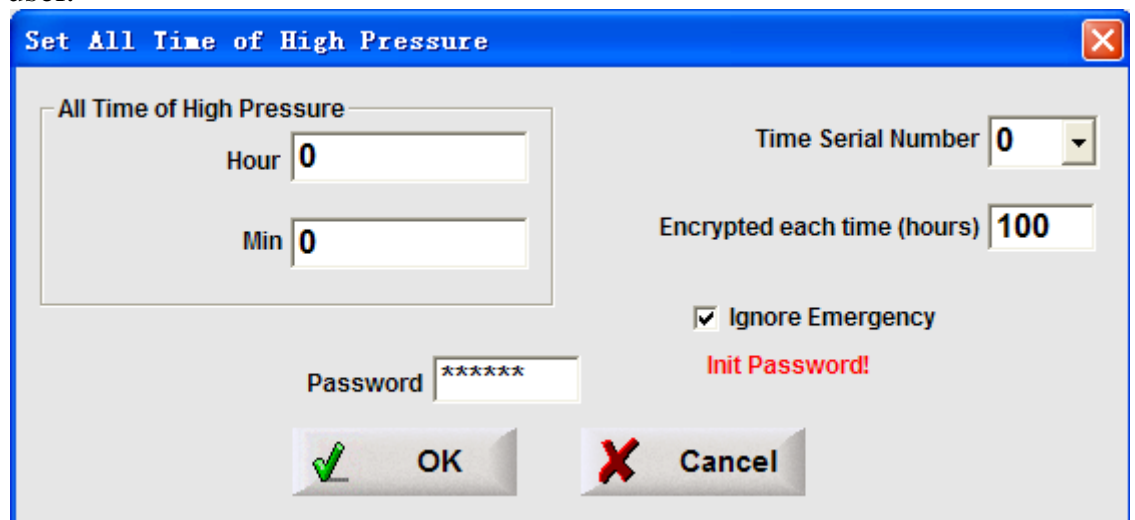
If the 'ignore the coordinate of machine tool' in the 'system parameter' is not selected, the dialog box will not be opened, saying, it is forbidden to set the coordinate of machine tool. If the 'ignore the origin of machine tool' is selected, the dialog can be opened as picture 4.6 showed. Due to the ignored origin, actually there is no coordinate system of machine tool. It is just looks as a reference to the components coordinate without influence on the whole procession. This function needs password. Password is "2123721".



Picture 4.6

§4.6 Set HP Time

As picture 4.7 showed, this function is just for modification of manufacturer instead of user.

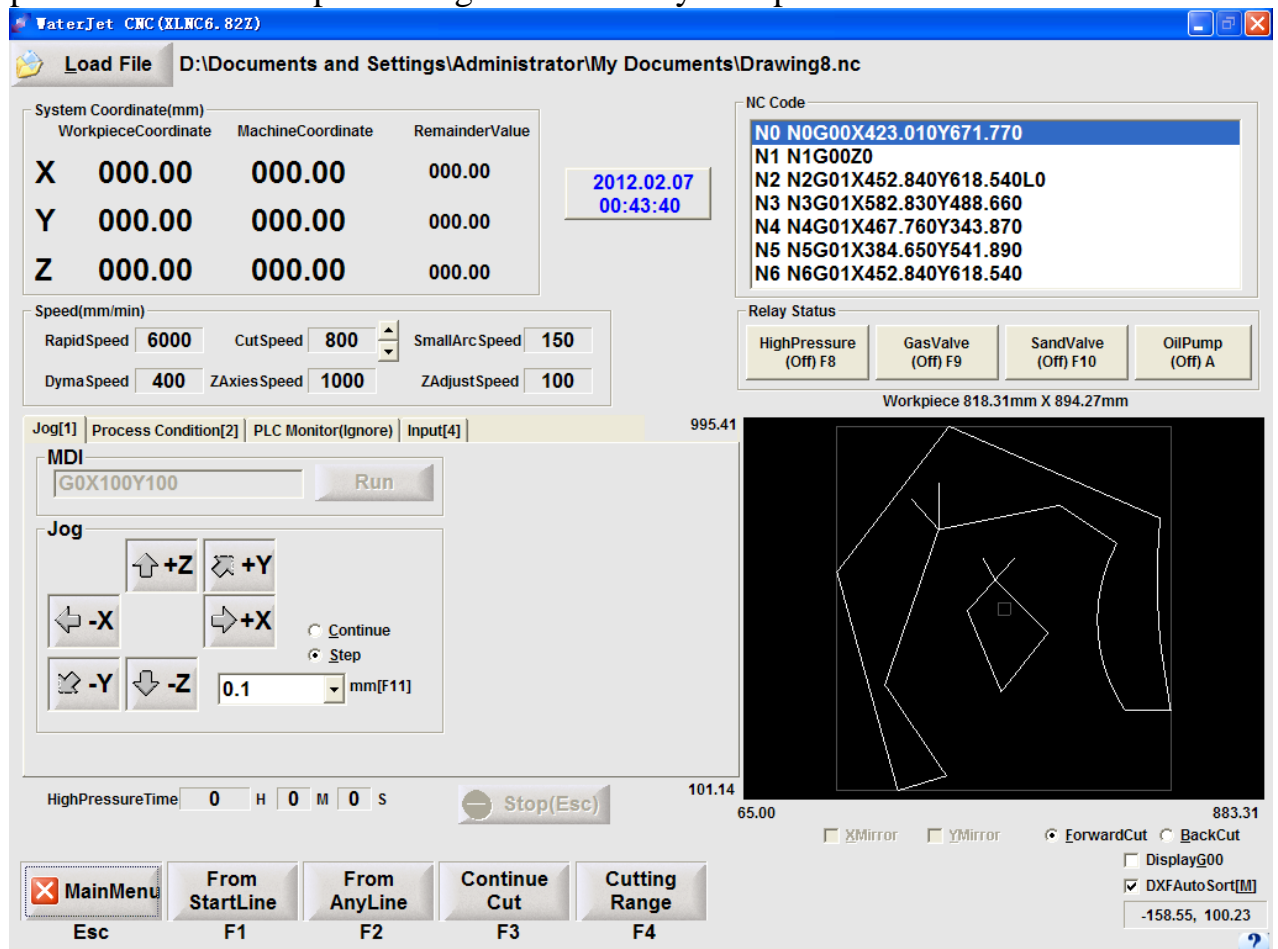


Picture 4.7

Chapter 5 Automatic Mode

Entry the auto function as picture 5.1 showed press the Esc key to stop during processing.

Under the stage processing, we can use the 'B' key, 'F' key, selecting 'forward processing' or 'backward processing' during processing. 'B' is for backward processing along old trace; 'F' is for forward processing along old trace. The stage process and smooth processing is set in the system parameter.



Picture5.1 Automatic Procession

§5.1 Process from the beginning

Process from the first line of the processing mode file.

§5.2 Begin at any number

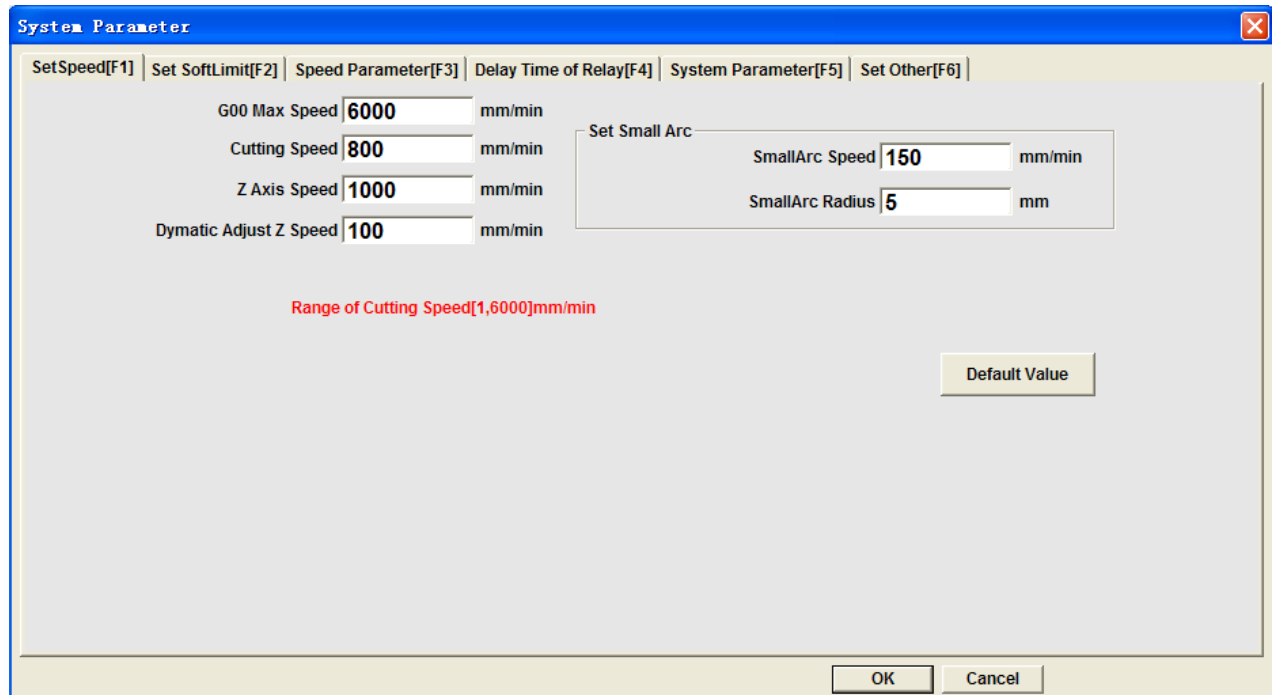
Begin at any segment number of the file. After entering this function, press Enter key. Then the talking blank appears. If the inputted segment number over the length of the file in the memory, the system will make instruction automatically.

§5.3 Continue

If it pauses after processing from the beginning or any step, to effect other orders, such as removing the surplus. After finishing, it will process according to this order. The procedure will return automatically to the paused position before to carry on processing.

Chapter 6 Set Parameters

Under the main menu, the dialog box will appear when we press F5 or select 'set parameter'. There are speed set, soft limit, setting parameter of speed increasing and decreasing, time delay of relay, system parameter, processing set in the talk blank. Press separately F1 to F6 to change or to select using the keypad.



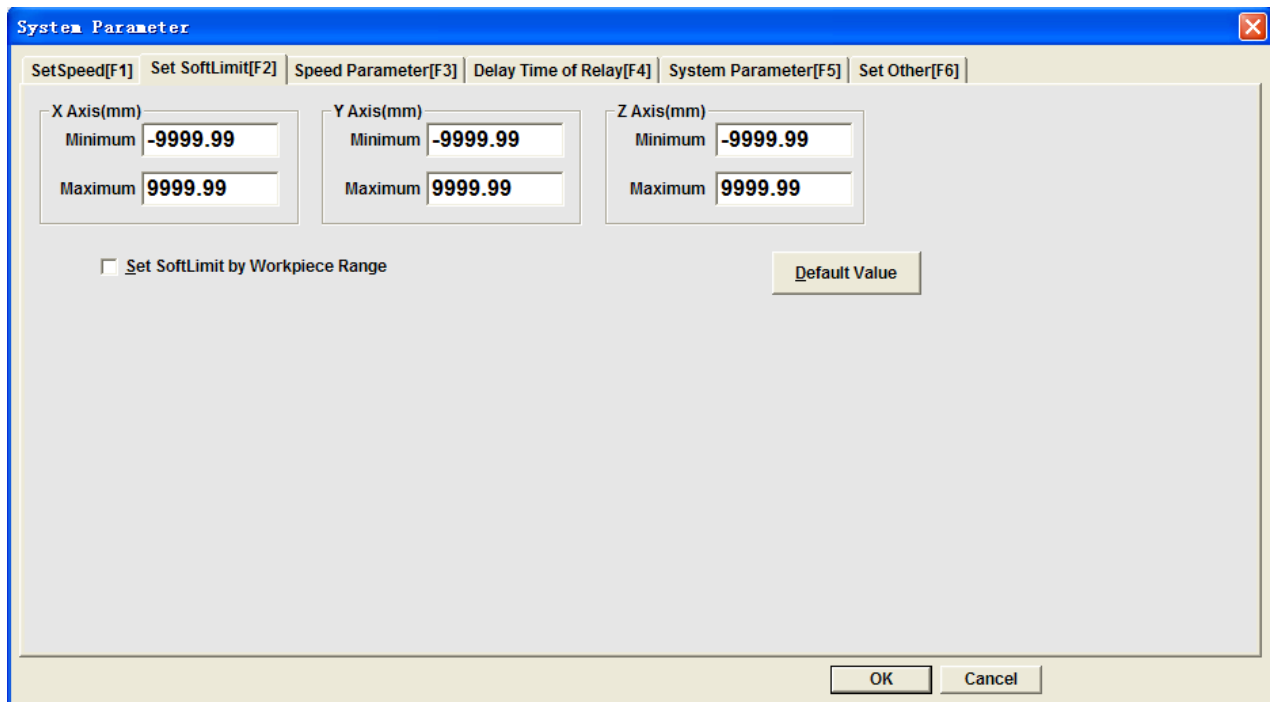
Picture 6.1

§6.1 Speed Setting

As the 6.1 picture showed, 'the max speed of G00' is G00 moving speed and the spot moving speed in the procedure. The cutting speed is line and small arc cutting speed which can not be over the highest speed of G00. The 'cutting speed and the radius limit of small arc' is to set processing speed of small arc. In the procedure, when the 'radius of arc' is smaller than the setting value of the 'radius limit of small arc' the system will process at 'the cutting speed of small arc' instead of 'the cutting speed'. Press letter 'D' or selecting 'default' to set all the parameter of this function as default automatically.

§6.2 Set Soft Limit

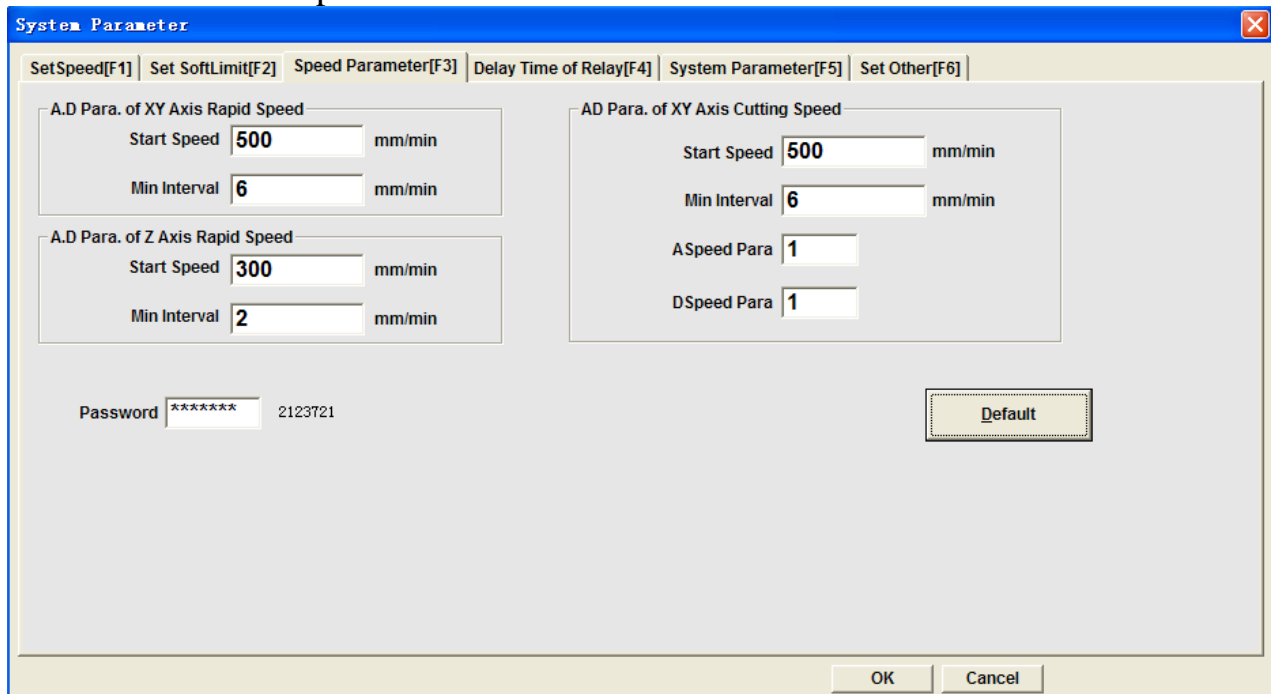
As picture 6.2 showed, set the max travel range of the machine tool. This parameter can be changed by user, but the min value must be smaller than the max value.



Picture 6.2

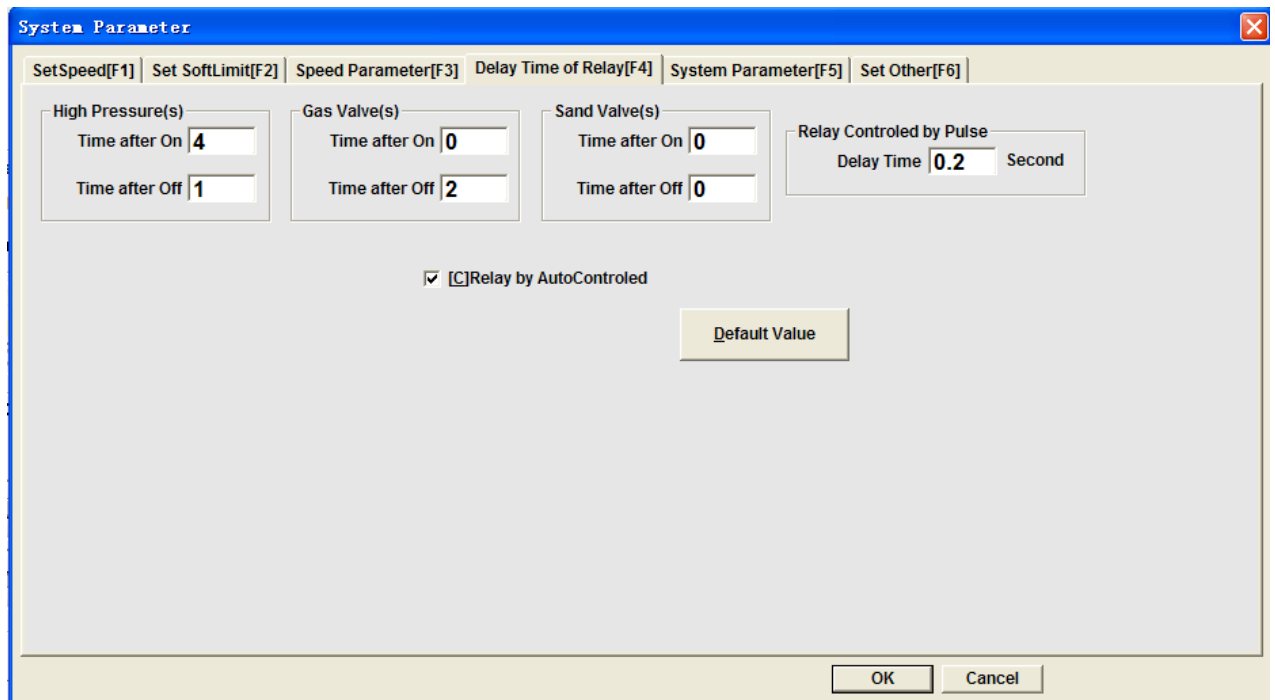
§6.3 Speed Parameters

As picture 6.3 showed, this function is used to adjust the ascend and descend speed feature of the motor. The start speed is the initial speed of the motor. The unit is mm/min. Then each sending of the pulse of ascend or descend parameter will raise the a min interval speed.



Picture 6.3

§6.4 Delay Time of Relay



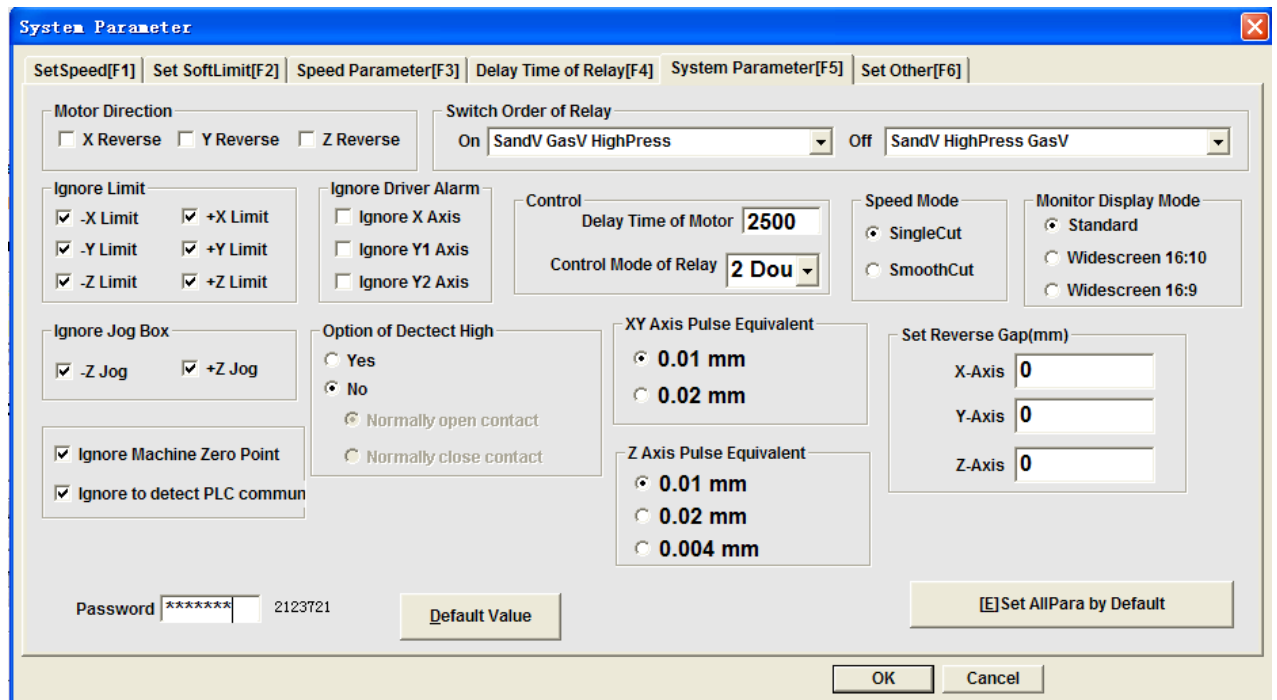
Picture 6.4

This function can be used to set the delay time of high pressure, gas valve, sand valve and oil pump after the on/off. The unit is second. The pulse relay is for immediate close after open and a moment delay. If it is set to 0, the high pressure and oil pump will not work due to short time. It is usually set to 0.2 second which is enough for work.

If the 'relay auto control' is not selected, the high pressure, gas valve, sand valve and oil pump will not be controlled during the auto processing. That operating is only used for debugging.

If the inflexion point delay is not 0, the procedure will have time delay when it comes across the inflexion point during auto processing, which is for split at the corner and suitable for using at the time of cutting steel sheet.

§6.5 System Parameter



Picture 6.5

Password is “2123721”, The parameters are defined as following:

1. The delaying time of “spot-moving” needn’t be rewritten. It is decided by Primary Frequency of the computer. The higher the Primary Frequency of computer is, the larger the value is. For example, the Primary Frequency is less than 600M and the relevant value may be 1200. If the Primary Frequency is around 1G , the value can be set 2000.
2. 0: close 1:puls 2 double: The controlling ways of relay may differ from factory to factory, so there are three ways to choose.

If the value is “0”, there are four relays to control the on-off of high pressure, the valve of high pressure gas, the valve of pressure sand and the on-off of the oil pump. When on, the four relays are closed. Otherwise they are open.

If the value is “1”, also has four relays to control the high voltage, the valve of high pressure gas, the valve of pressure sand and the on-off of oil pump. On the job, the four relays are closing . Otherwise the four relays are opening. And the on-off of high pressure and oil pump is controlled by plus. They always closed at the start, then open it. How much time they closed is decided by the value in the “on-off delay time”. Usually, the value is 0.2 second.

If the value is “2”, it has six relay to control the on-off of high pressure, the valve of high pressure gas, the valve of pressure sand and the on-off of oil pump. The on-off of high pressure and oil pump are controlled by four relays. The connection with PC is show in Fig. On the job, the valve of high pressure gas, the valve of pressure sand are closing . Otherwise the two relays are opening. And the on-off of high pressure and oil pump is controlled by plus. They always closed at the start, then open it. How much time they closed is decided by the value in the “on-off delay time”. Usually, the value is 0.2 scond.

3. If the process at the mode of “SingleCut”, the system read each line of NC code

and must ascend speed at the start ,then descend speed at the end. So it have the reverse machining function. And it suit for stepper motor and AC servo motor. At the mode of “SmoothCut”, the system do not have to ascend and descend speed with each segment. Because the software can know the next segment’s detail message, it can know the system have to descend speed or not. So can advance the efficiency and descend the roughness. And it only suit for AC servo motor. For example, to process a circle that contains four segment of arc. If it at the mode of “SingleCut”, the system have eight times to ascend and descend speed. If it at the mode of “SmoothCut”, the system only have two times to ascend and descend speed.

§6.6 Set processing

When processing multitudinous products, we can set circulation times. It will reprocess from beginning after finishing a procedure.

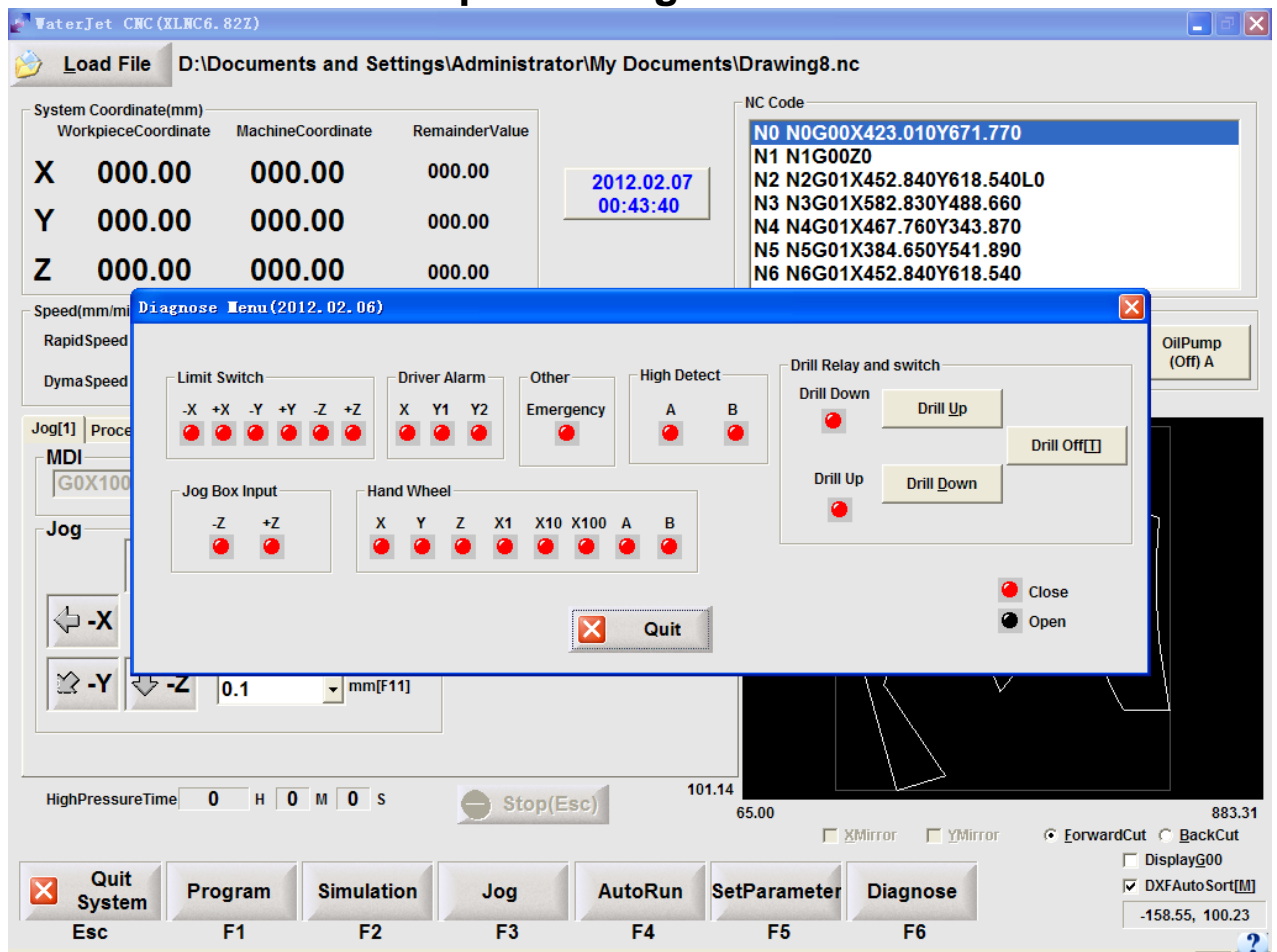
Set rotational angle for any shape rotation within 360°. The (0,0) is looked as basic point. The max and min figure of the shape will change automatically after rotation.

The screenshot shows the 'System Parameter' dialog box with the 'Set Other[F6]' tab selected. The dialog is divided into several sections:

- Set Loop Number:** Contains two input fields, 'AllLoopNumber' and 'CurrentLoopNumber', both set to '1'.
- Option of Rotate:** Includes a label 'Option of Rotate', a text field 'Rotate based on Point(0,0)', and a 'Rotate Angle' field set to '0' with the unit 'Degree'.
- Delay of Inflection Point(Second):** A single input field set to '0'.
- Gas drilling parameters(Second):** A section containing a checkbox 'Use gas drilling during automatic processing' which is currently unchecked. Below it are three input fields: 'Delay after gas drilling starts' (10), 'Fall time of gas drilling' (5), and 'Rise time of gas drilling' (5).
- Deviation Fields:** Two input fields for 'X-axis deviation relative to the tool(mm)' and 'Y-axis deviation relative to the tool(mm)', both set to '0'.
- Buttons:** A 'Default' button is located to the right of the deviation fields. At the bottom right are 'OK' and 'Cancel' buttons.

Picture 6.6

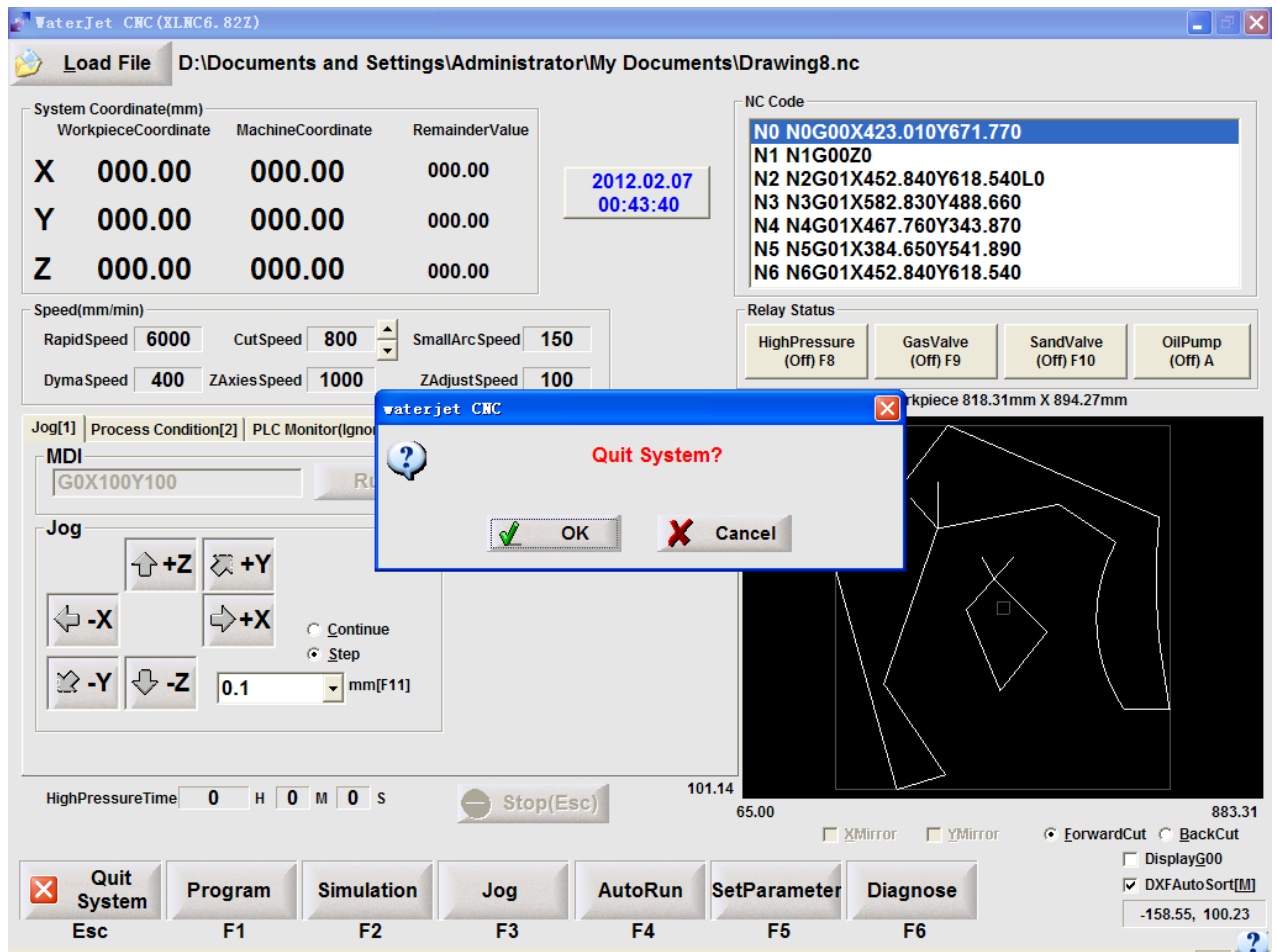
Chapter 7 Diagnose Mode



Picture 7.1 Diagnose Mode

This function (as picture 7.1 showed) can be used to check up the connection of all input switch. Enter this function, then press spot-moving switch or limit location switch. If the selected axis displays red box on the screen it means connection of wire. The white box means non-connection of wire.

Chapter 8 Quit System



Picture 9.1 Quit System

As picture 9.1 showed, press the 'Esc' key. The dialog box of quitting system will appear. Choose and press 'Yes' to exit system and save all the parameter automatically. While, choose and press 'No' to give up.

Chapter 9 Common Error and Solution

1. Can not enter the control system interface

Maybe the software dog is not inserted correctly, or the LPT is broken. If the reason can not be found, please reinstall the software, and enter the install disk for reinstallation.

2. Can not return to the origin or lose pulse after processing for a moment

The delay figure of the motor control may too small in the system parameter, which is related much with main frequency. Normally, it is between 1500...2000. The higher the speed of computer, the larger the required figure. If the alternating current stepper drive has the accumulation display of command pulse, we may check by the drive whether the pulse was lost ever returning to the origin

Chapter 10 Common G Code

- 1、 G00 rapidly set point

Format: G00 X* Y*

Note: *is value input(back is the same as it).

Moving from the current point to the coordinate point inputted by * for max speed.

- 2、 G01 Line Interpolation

Format: G01 X* Y*

- 3、 G02 Clockwise Arc

Format: G02 X* Y* I* J* F*

X* and Y* are the coordinate of the end point. I* and J* are the coordinate of the circle center to the start point of arc

- 4、 G03 AntiClockwise Arc

Formate: G03 X* Y* I* J* F*

X*、 Y* are the coordinate of the end point, I*、 J* are the coordinate of the circle center to the start point of arc.

Chapter 11 Basic Operation Procedures

- 1、 Emulation the program file that needs processing under the emulation menu. Observe and make sure the graphics is correct. And then, make sure whether the maximum and the minimum of the two axis in NC code are correct. Finally, return to the main menu.
- 2、 After installing the part, move the axis of the machine to zero point of the lower-left corner. Enter the 'CurPoint' menu under 'Hand' function menu, and set the coordinate to zero, and then return to the main menu.
- 3、 After entering 'Auto' menu, choose and press 'Start' menu to start Cutting.

Notice: If the machine already processed a part, and have set the coordinate to zero, you should leap over step 2, saying, to auto process directly after emulation.