WEIHONG® WATERJET MACHINE'S MOTION CONTROL SYSTEM

NCSTUDIO™-V9

Users' manual

Preface

Thank you for choosing our products.

This manual will help you acquaint the products of our company, from which you can get the information of the system configuration, installation and usage, etc.

Please carefully read the introduction before installation, which will help you in using our products, since this manual will offer you detailed installation process and functions of the system.

Because of continuous updating, the software and hardware you have bought may differ from some description in this manual, for which we apologize to you.

Pay attention to the followings:

1. Don't plug or pull out the cable connected with the computer when power on.

2. Don't plug or pull out the cable of manipulation boxes when power on.

3. In order to ensure safety and avoid disturbance, the shell of computer and engraving machine must be grounded.

4. If machine is not in use, please shut off the power supply.

5. Please unplug the power supply, if machine is not used for a long time.

6. The life time of the spindle motor bearing is inversely proportional to its rotating speed.

Fast Master Chapter

This chapter applies to the condition that all wires linking to electrical apparatus have been connected and the control card and Ncstudio™ system have been well installed.

➢ Check if the conjunction is correct. Test whether the communication signal is normal.

Join the terminal board to port JP1 of the card with DB37M/F cable, add 24V power supply to the terminal board and supply power for the machine signal system (proximity switch, etc). Examine signal input indicator LED on the terminal board: if the origin switch is normally closed, three LEDs of X0, Y0, Z0 should be alight. You can touch the origin switch through artificial imitation. (For travel switch, artificial pressure can be used to observe whether signals can be received. For photoelectrical switch, artificially obstruct the light to see if the signals can be gotten. For metal proximity switch, artificially touch it with metal to see if the signals can be gotten.) If the corresponding LED is out when being touched, it indicates the origin signals have been sent to the terminal board. If the connected origin switch is normally open, LED should be normally out, and by artificially touching the switch, LED will become light, which shows the origin signals have been received by terminal board. The same method can be taken to test other ports and to make sure the correctness of the wiring between terminal board and machine tool. This method can greatly shorten the debugging time.

➢ Debug Signals

Open the computer and run NcStudio™ software. Open [DIAG]→[I/OPort] window, and you will see many mapping InPorts and OutPorts. Green lamps indicate the signals are valid, while red lamps indicate the signals are invalid. Revise the polarity of the mapping InPorts according to the type of the origin switch, E-STOP buttons, and other buttons you have chosen, and so on. Polarity of the normally open InPort is "N", and that of the normally closed InPort is "P". After all modifications have been finished, exit from NcStudio software and restart it. Your modification will become effective. Supply power to the electrical box. Open the computer and run NcStudio™ software. Open [DIAG]→[I/OPort] window, you will see many mapping InPorts and OutPorts. Dots which are in front of signals of "program start", "program stop", "the reference point" of X, Y Z axis and "emergency stop" InPort should be in red which indicates all these signals are invalid. If in green, you should examine the circuitry in electrical box and the polarity of the signals. If there is no problem with the circuitry, you have to revise the corresponding polarity to make the dots in front of the signals in red.

Load processing program, open [AUTO] interface, press [Start] button, and observe if the color of the dots in front of "program start" changes. This dot should be in green when "start" is pressed and in red when "stop" is pressed. If no change occurs to this dot, please turn to step one to check if LED on terminal board functions normally and the wiring conjunction of DB37M/F & DB50M/F cable with the card and the terminal board is loosened. Imitate to touch the origin signals according to the first step. Check the reference point signals of X-axis, Y-axis and Z-axis. Once the origin signal is being touched, the signal dot should be in green.

➢ Set Parameters

Set [pulse equivalent] of manufacturer parameters. The smaller the pulse equivalent is, the higher the resolution will be. But if the pulse equivalent is too small, it will influence the maximum feed

speed. Generally speaking, users of model machine can set the pulse equivalent as 0.001mm/P (The corresponding maximum feed speed is 9600mm/min) or as 0.0005mm/P (The corresponding maximum feed speed is 4800mm/min); for users who are not critical about the accuracy, pulse equivalent can be set a little larger, such as 0.002mm/P (The corresponding maximum feed speed is 19200mm/min) or 0.005mm/P (The corresponding maximum feed speed is 48000mm/min). After the pulse equivalent has been confirmed, please compute the electronic gear ratio of servo driver in accordance with value of this pulse equivalent. Electronic gear ratio= encoder resolution × pulse equivalent × mechanical deceleration ratio/ pitch; thereinto, mechanical deceleration ratio= reducer rotation speed input/ rotation speed output=driven gear tooth number/ driving gear tooth number. Move the machine manually, make sure the correctness of moving direction of each axis, be noted NcStudio abides by right-hand coordinates system. That is to say, rightward movement of X-axis is the positive direction, upward movement of Z-axis is the positive direction, while the positive direction of Y-axis is moving far away from the operator. (If Y-axis movement is the workbench movement, the positive direction is the direction of machine tool moving towards the operator). If this direction is not right, you can revise [Axis Direction] parameter or relative servo driver parameter of manufacturer parameters. Set [table travel range] of manufacturer parameters on the basis of the actual machine tool dimension to make the software limit available. Set [mechanical reference point] of manufacturer parameters on the basis of sensor installation position of the three axes, execute [back to reference point] action, and carefully observe the moving direction of the axes when backing to the reference point. If moving direction is abnormal, you need to reset the parameters of [back to reference point] under manufacturer parameters.

➢ Load processing file and start machining (or perform manual machining)

Software Update Records

In order to provide convenience for user to know the state of our software update rapidly, we list here the update records of each software version. User can compare this record with the software version number that he has got, and upgrade the software promptly. If user wants any particular new function, please feel free to contact us.

New function:

➢ User can customize step-size under [Manual] interface.

➢ [System maintenance], user can maintain the system, and also can make installation package for a stable system with entire parameters.

➢ Under [Auto] mode → [Position] function window → [Information] screen, add statistics function for waterjet machining distance.

➢ Add encoder origin signal, servo driver alarm signal and servo driver enabled signal.

➢ Add automatic centering, user can record current machine coordinate after setting mechanical origin. After automatic center is set, the recorded value will be regarded as current workpiece coordinate origin.

➢ Strengthen the simulation ability and stability; [Back to the Reference Point] and [Save the

Reference Point] can be tested during simulation process; add [Travel Range] and [Position Limit] in [Simulation] window.

➢ Further enhanced function: Icons in [I/OPort] interface vary with different states; the interface has been user friendlier.

➢ Further enhanced function: a timer and a counter are appended for advanced users in [PLC] Window.

➢ New auto-reverse function, which can achieve reverse processing.

➢ New backup waterjet parameters, user can save or read setting of waterjet parameters.

➢ Add hardware time encryption to protect the interests of customers.

➢ Support G code format, PLT format, DXF format and ENG format.

Content

1 Summarization

The software of Ncstudio™, waterjet machines' motion control system, is independently developed by Shanghai Weihong Electronic Technology CO., Ltd. with copyright. It can directly support G code, DXF format, PLT code format and ENG code of JDPaint processing file that are created by CAD/CAM applications such as UG, MasterCAM, CASMate, ArtCAM, AutoCAD, CorelDraw, and so on.

Ncstudio™ is based on the operation system of Microsoft Windows and takes the great advantage of 32-bit computing and multi-tasks. At the same time, the interface of standard Windows style is user-friendly and dependable as well as easy to learn.

Apart from functions of automatic mode, manual mode, and backing to the reference point, this numeric control system also features the functions of simulation, processing time forecast, loading processing track in advance, dynamic tracking, breakpoints resuming (program block skip), and so on.

1.1   Software features

Ncstudio system includes the following functions:

- Process automatically. Completely support G code of ISO standard, HP PLT format, DXF format and JDPaint ENG format.

- Support manual function; user is allowed to control the machine tool not only by handheld device but also by input equipment of computer, such as keyboard and mouse.

- Support incremental feeding function; it is convenient for user to set accurate feed length, and step-size can be adjusted.

- Support special user data input function.   User can immediately execute the G code after direct on-line input of it.

- Support processing wizard. To execute functions such as contour milling and special lines, user only needs to input several parameters.

- Support single block mode. User can set the program being selected to be executed as [Single Block], which provides very good support for error diagnosis and failure recovery.

- Support advanced auto functions of breakpoint memory, block skip, etc.

- Support save/restore workpiece origin.

- Support backing to the reference point accurately.

- Support feed override on-line adjustment. During the machining process, user can adjust it at any time. The least value is "0" that indicates the processing is suspended, and the maximum ratio is 120%.

- Support new function of high-smooth speed connection algorithm. In a general numerical control system, connection speed between two G codes commonly is a fixed value (e.g., it may be "0" or a certain small fixed value). However, in new version of NC system, it adopts a particular adaptive prediction algorithm for processing speed. In accordance with speed value, direction and maximum acceleration, as well as the function of intelligent look-ahead, the algorithm adaptively decides the connection speed between the current instruction and the next instruction. It has greatly increased the processing efficiency (efficiency enhancement ranges from 30% to 300%) and improved the processing capability, eliminating the speed chatter marks left on the processing surface.

- Support particular function of analog display. Through simple operation, user can observe the processing result from different angles to understand the processing result more accurately and more intuitionally.

- Add new function of loading processing path in advance. After loading the processing file, system will automatically analyze and output the processing tracks. User can directly call out the tracks to view.

- Support simulation function. This function can rapidly simulate the machining process and will finish in a short time. Meanwhile, it can check if there is any mistake with the program and if the processing result can meet user's satisfaction. Moreover, it can count precisely the actual processing time.

- Support keyboard operation. To satisfy user's need, we offer a strong support to the keyboard operation.

- Support log function. The system has a strong log function which is very helpful for user to view the detailed processing information and system diagnosis.

- Add a built-in processing file manager. What user needs to do is to save the processing file to a specified directory; Ncstudio™ will administrate these files in a built-in file manager.

- Add a built-in processing file editor. User can introduce a processing file into file editor at any time to edit and modify it.

- Support displaying file processing information function. By simulation or actual processing, [file processing information] window can help user to count some important information, such as processing time, processing range, the amount of processed workpiece, etc.

- Particularly support [Parameters auto backup] function. In this window, system can automatically save parameters' settings information, which saves user a lot of time and trouble to set parameters time and time again.

- Support backup for 8 parameters' setting in [Parameters Back Up] window. User can save different parameters' setting of different processing files, and load them at any time.

- Particular [I/O Port] window can do map simulation, test and regulate polarity, which helps user well realize hardware detection, processing supervision, and fault diagnosis.

- Add PLC module, through which user can control I/O port.

- Support intelligent look-ahead; system will analyze situation in the forward and backward distance, in order to decide interpolation strategy and improve the integral smoothness of workpiece.

- The maximum length of machining file is 2 G byte.

- Support modification of abrasive ratio, the range is 0-100%.

- User can record current mechanic coordinate after setting mechanic origin, if [auto center] is on, the recorded value will be set as current workpiece coordinate origin.

- Support reverse cutting, using in repetitive cutting.

- Support "auto open/close control valve sequentially", "open/close valve delay".

- Support MPG (Manual pulse generator), user can determine whether to use MPG or not according to his needs.

- Support multi-language, currently, English and Chinese have been embedded in the

software, other languages will be added in demands.

■ Support backlash compensation, lead screw error compensation, quadrant error compensation, etc.

2   System Installation

It is very simple to install Ncstudio™ application. Advanced users or those who have been familiar with the application can refer to the fast master chapter.

2.1　The basic configuration of Ncstudio^{TM}

◆　**Host computer**

CPU:　　　　　　　　　　main frequency 1G or above.

Memory:　　　　　　　　512 M or above.

Hard disk:　　　　　　　20G or above.

Display adapter:　　　　1024*768 at least

Display:　　　　　　　　14" VGA or above

CD-ROM:　　　　　　　　4X or higher (optional)

Mainboard Expansion Slot:　More than 2 slots of PCI

◆　**Operation system**

Microsoft Windows 2000 Professional

Microsoft Windows XP Professional

2.2　Ncstudio^{TM} installation

If there is already an old version of Ncstudio^{TM}, please delete it, and then install the new version. Regarding how to delete the old version, please turn to chapter of uninstall procedure. Or user can directly install the new Ncstudio^{TM}, which will lead to overriding the formal data.

Ncstudio™ includes two parts: the software and the motion control card. Therefore, the setup of the system is also divided into two stages: the software setup and the motion control card setup.

Please complete the installation of the software before installing the motion control card. In this way, driver for the motion control card needn't be separately installed. Following instruction does not include installation of electrical equipments, about which please refer to "Manufacturers' Manual" for detailed information.

2.3　Ncstudio^{TM} software setup

Software installation can be divided into several steps as following:

Supply power to the computer, and the system will enter into the windows operation system automatically after start-up. Please install the operation system at first if you have not done so. After windows operation system has been installed, don't forget to exit from other running applications.

Put the setup CD of Ncstudio™ system into the CD-ROM. Double click "My computer" icon, then double click the CD-ROM icon. Find out the SETUP.EXE file and double click its icon . The first dialogue box that appears is the [Language of the theme] dialog, see Fig. 2-1. In V9 version, you can also switch over between English and Chinese in the system. If you click [Chinese], run Ncstudio™

system after installation, the system will take the Chinese interface as the default interface. For the same reason, if you click [English], the system will take the English interface as the default interface. If there is already Ncstudio™ on the computer, the new package will delete the old version and install the new version to update the software. At this time, an update prompt will appear on the screen, shown in Fig. 2-2.
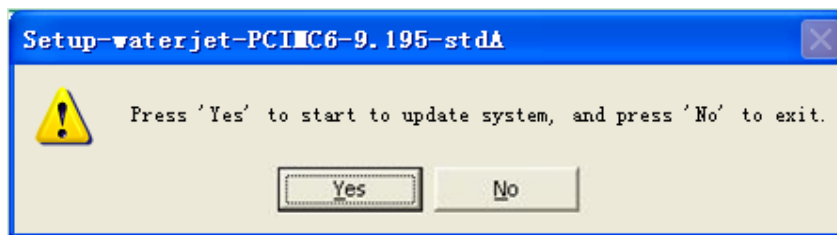
Fig. 2-1 Language selection dialog

Fig. 2-2 Update reminding dialog

Click [Yes (Y)]. In order to avoid disturbance, system will ask you to save parameters and delete the old version before going on further installation. As Fig.2-3, it is the prompt to save parameters before installation.
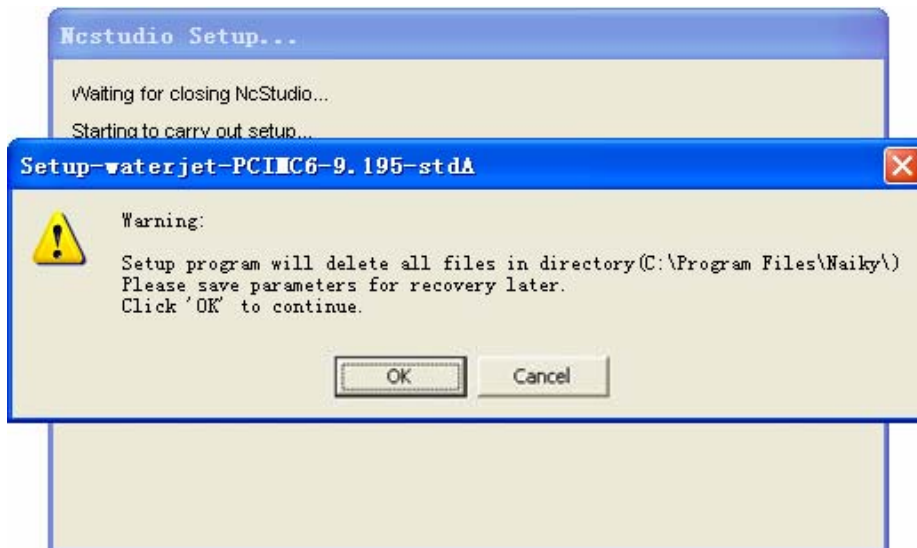
Fig. 2-3 Remind user to save parameters before installation

Click [Yes (Y)]. Installation begins. Ncstudio™ will be installed to the default directory, C:\Program Files\Naiky. Installation progress bar will show you the installation speed. See Fig.2-4. After installation, prompt will remind you whether to shut down the computer (We recommend closing the computer to facilitate the installation of the control card). See Fig.2-5.
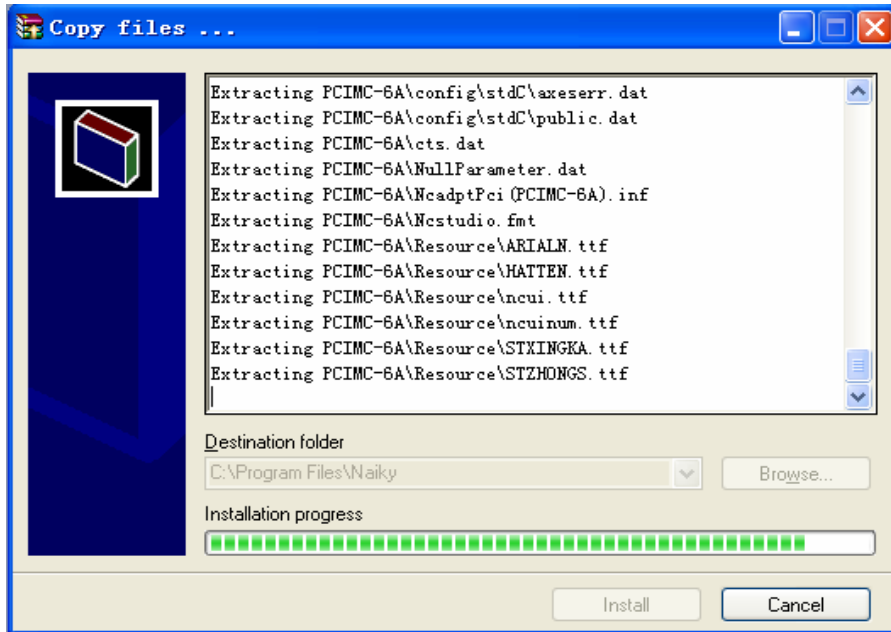
Fig. 2-4 Software installation interface

Fig. 2-5 Remind whether to shutdown the computer (Recommend: OK)

Here the software installation has been completed.

2.4 Install Ncstudio™ control card and other devices

Installing Ncstudio™ motion control card is divided into the following steps.

1. Turn off host computer's power supply, open the cover, and insert the motion control card into a suitable PCI expansion slot.

Note:

While installing the motion control card, press two flanks of the motion control card lightly, make sure the motion control card is firmly inserted into the slot and well contacted with computer baseboard, stable without fluttering, then tighten the locknut of the card and finally lid the cover. The installation of motion control card has been

completed.

2. Similar to the installation of the control card, please insert peripheral equipments into the corresponding slots or ports and make sure they are firmly fixed without fluttering.

3. The installation has been completed now. Restart the computer.

After the installation of motion control card and other peripheral equipments, please restart the computer. Windows operation system will report a new hardware has been found, and eject a hardware update guide dialogue. See Fig. 2-6. You should choose [No, not at this time] and click [Next] to go to next page. See Fig. 2-7. You should choose [Auto Install the Software] and click [Next]. See Fig. 2-8. Then the operation system will install the driver application for Ncstudio^TM software automatically. Up to this point, the whole installation of Ncstudio^TM software and the motion control card have been completed. To start Ncstudio^TM, click the icon on the desktop or click [Start] menu on the desktop →click [Applications] →click [Ncstudio^TM].

Fig. 2-6 Page one of hardware update guide

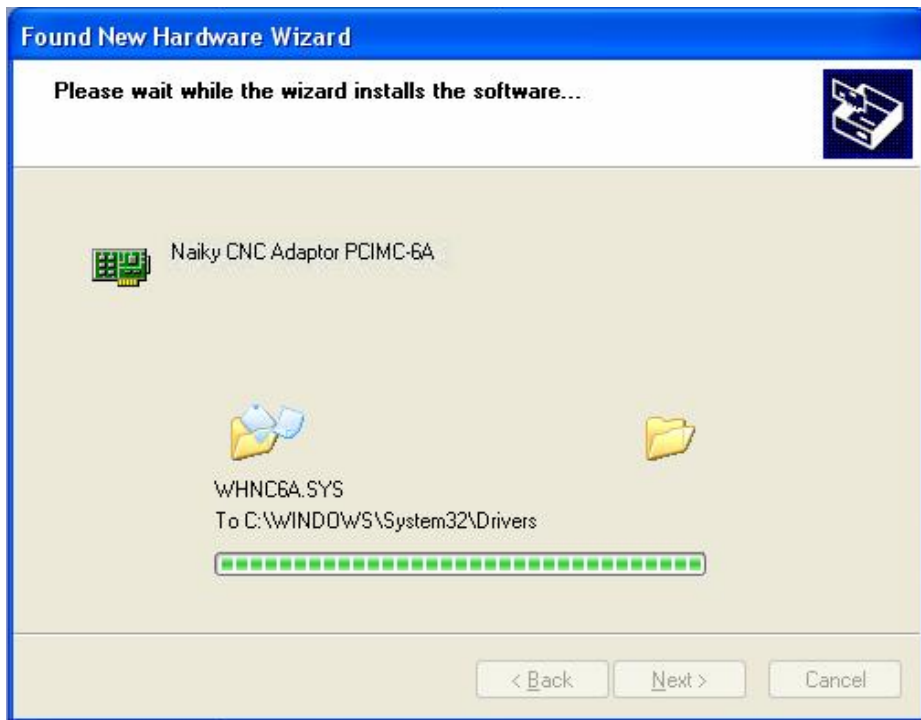Fig. 2-7 Page two of hardware update guide

Fig. 2-8 Page three of hardware update guide

Note:

Figures above may be different in different software version. They are just for reference.

2.5　Other installation problems

If you don't follow the above installation procedures and insert the motion control card before

software installation, the computer will remind you to install driver for motion control card installation. Please cancel the dialogue of installing driver, and directly install Ncstudio™ software, then restart computer. After restart, the operation system will request once again to install driver for the motion control card, at this time, you can set it according to the hardware guide of the last chapter. After all installation has been finished, you can use Ncstudio™ software.

2.6   Uninstall NcStudio™ system

Ncstudio is a green software which has the following advantages; easy installation, no need to write installation information into registration form, and no remnants files in HDD after deleted. Therefore, to uninstall Ncstudio™ system, user only needs to delete its document folder: "C:\Program Files\Naiky", Ncstudio in [Applications] of [Start] menu (as Fig. 2-9), and the shortcut menu on the desktop.

Fig. 2-9 [Start]→[Applications]→delete [NcStudio]

3 Basic Concepts of Ncstudio™

Ncstudio™ is a comparably complicated system which refers to many concepts, such as workpiece coordinates system, machine coordinates system, operation mode, operation state and so on; unavoidably you will use these concepts. Therefore, you should understand these concepts before using Ncstudio™.

3.1 Operation mode and state

3.1.1 Operation mode

Regarding machining operation, there are totally three modes as below.

◆ **Reference point mode ([back to the reference point])**

[Back to the Reference Point] mode is the process to unify the internal coordinates and the machine actual coordinates. Therefore, during activation process, it is essential to back to the reference point. Every time the system is started, the default operation mode is reference point mode. It is very convenient for user to implement [Back to the Reference Point] action.

◆ **Auto mode**

Under automatic operation mode, the machine tool generates motions through the procedure loaded in advance. Therefore, the processing procedure must be loaded in advance in this mode.

◆ **Manual mode**

Under manual mode, user can operate with manual equipments, such as manual pulse generator and manual control panel; or user can directly input commands to operate the machine. Manual mode is subdivided into the following modes: jog, stepping (also-called increment), handwheel, etc.

➢ Under [Jog] mode, the machine will start motioning once the manual button is pressed, and won't stop until the button is released.

➢ Under [Stepping] mode, press down the manual button and release it. Machine will move a specified distance. The distance is thought as step-size, which is divided into four grades: 0.01mm, 0.1mm, 1mm, 10mm. Moreover, user can customize step-size. Therefore, user can accurately control the displacement of the machine tool under [Stepping] mode.

➢ Under [Handwheel] mode, user can control the machine tool by handwheel. Each time user turns the handwheel a lattice, the machine will move forward a step distance.

3.1.2 Operation state

In terms of the motion mode of the machine tool, each operation mode can be divided into the following types of operation states (the operation mode and operation state together decide the state of a machine tool).

◆ **IDLE state**

Idle state is the most common state. Under this state, the machine has no output motion, but is always prepared to accept any new task.

◆ **ESTOP state**

This is an abnormal state. When there is a hardware fault or when user presses down the [Reset] button, system will enter ESTOP state and implement the predetermined protection actions, such as

turning off waterjet and cool pump. In this state the machine tool is locked and cannot carry out any new motion. After hardware fault or ESTOP state is obviated, system will automatically implement [Reset] action and make the machine tool return to the IDLE state.

◆ **Running state**

When the machine is implementing any action, system enters into running state.

◆ **Pause state**

When a machine tool is running, if user performs [Pause] command, or presses down [Pause] button, or system parses a M01 command (waiting instruction), system will enter into PAUSE state and wait for user's further instruction. To suspend current action and let the system enter into IDLE state, several methods can be effective. For example: perform 'Start' command, or press [Start] button, or perform 'Stop' command, or press [Stop] button, or press [Reset] button under [Auto mode].

◆ **LOCK state**

Lock state is an internal state which occurs when user switches over states. Normally, user may not meet with this state.

3.2 Machine coordinate system

Coordinate system is a terminology that is used to describe the motion of a machine tool. For the sake of unification, standard coordinate system adopts the right-hand rule. See Fig. 3-1.
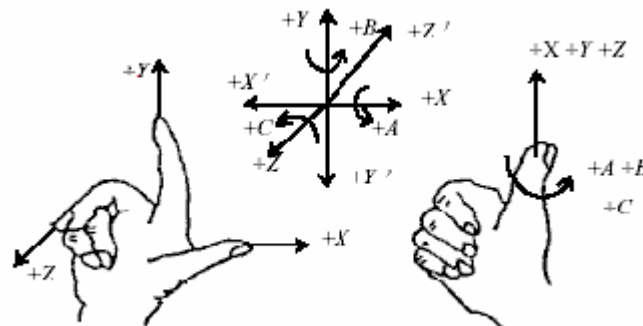
Fig. 3-1 Demonstration for right hand principle

For machine of milling types, the direction of machine tool axis is decided by both the type of machine tool and the layout of each component. The basic coordinate axes of milling machine are X-axis, Y-axis, and Z-axis:

——The direction of the Z-axis moving away from workpiece is the positive direction of Z-axis (Z+).

——X-axis is perpendicular to Z-axis and parallel to the clamped surface of workpiece. For the single column milling machine, if user faces the waterjet and looks in the column direction, right moving direction is the positive direction of X-axis (+ X).

——X-axis, Y-axis and the Z-axis together constitute the coordinate system that adheres to the right-hand rule.

3.2.1 Machine coordinate system

Machine coordinate system is a set of fixed right-hand coordinate system. Its coordinate origin is a fixed position that corresponds to the machine tool. Therefore, at any time, a certain point in space can be exclusively fixed by machine coordinate system.

The machine coordinate system requires the machine available of function of [back to origin], or this term will only appear in the software.

3.2.2 Workpiece coordinate system

It is more often to adopt workpiece coordinate system in machining various kinds of workpieces. Generally speaking, the machining position that we mention is a certain point relative to the workpiece, while the position where the workpiece is clamped is always variable corresponding to the machine origin, so it is necessary to introduce a set of more convenient coordinate system for workpiece machining. That is workpiece coordinate system. Workpiece coordinate system adheres to the right-hand rule as well. Its origin is fixed corresponding to a certain point on the workpiece, and is possibly floating corresponding to the reference point.

4 Operation Interfaces of Ncstudio™

Operation interfaces of Ncstudio™ are constituted by seven major sections. They are title bar, menu bar, numerical control state bar, information display bar, function window, status bar, and operation bar. Compared to the previous versions, this version has great changes in the operation interface.

> ➢ Picture display is more intuitional and detailed. Path and coordinate can be displayed under every operation mode. Therefore, the accuracy and instantaneity of machining view is ensured to the highest degree.

> ➢ The system provides powerful visual function, friendly interface, distinct style, and stylish interface patterns. User can freely select the theme of interface or design it by himself.

> ➢ Add button tip function. When user moves the mouse onto a button, a hovering tip will appear to tell user the button's function or its shortcut key.

> ➢ Add function of single shortcut key. Function transformation, parameters setting and machine stop/start can be finished by a single key, such as single letter, single number, or single function key, which has greatly shorten user's operation time.

> ➢ Add simulation operation panel. So many modes to be selected have greatly satisfied user's different requirements.

4.1 Integral interface
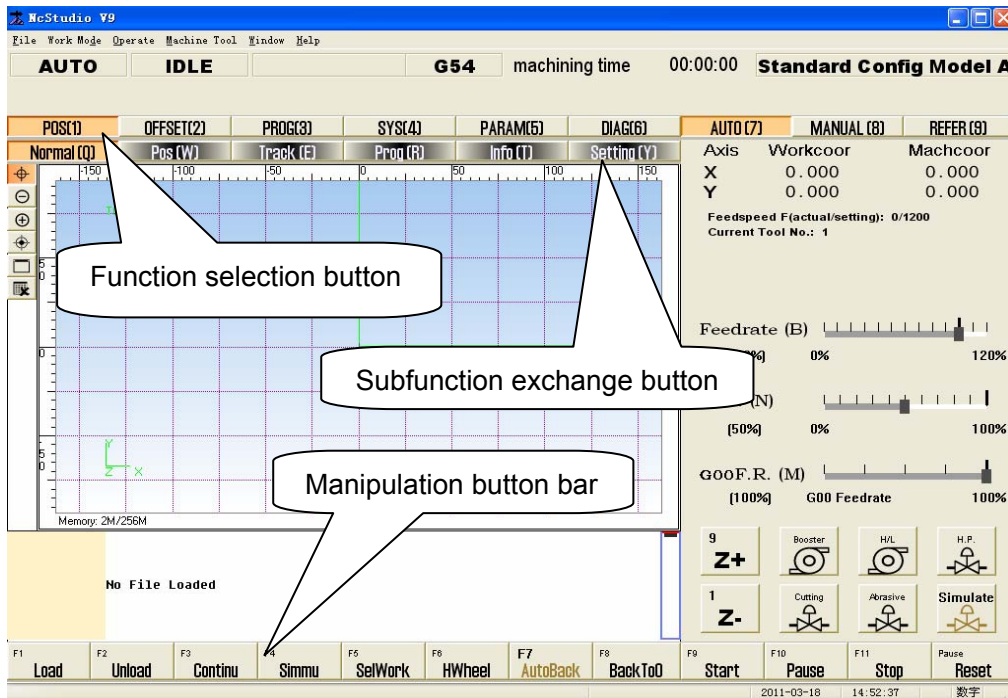
Fig. 4-1the Main operation interface

The main operation interface of NcStudio is shown as above in Fig. 4-1.

4.2 Title bar

The first column of Ncstudio operation interface is the title bar shown in Fig. 4-2. It is used to show the name of the application and the loaded processing file. The color of title bar is used to indicate whether the window is activated or not.

Fig. 4-2 Title bar

Note:

In Windows system, the concepts of active window and inactive window are very important. An active window refers to the window which accepts the input of keyboard at present. Anytime there is only one active window, and all the other windows are inactive.

Please pay attention to the color difference of the title bar between active window and inactive window. The color of an active title bar is blue, while the color of an inactive title bar is grey.

Click the icon or press "Alt + spacebar", and a system menu will pop up. This menu can control the position and the size of the window, such as "restore", "move", "close", "maximize", "minimize", etc. On the right side of the title bar, there are three control buttons, including button for restoring, button for maximizing and button for minimizing. These buttons are used for quickly setting the size of the window. Referring to the detailed operation manner, please turn to the on-line help in MS-Windows.

4.3 Menu bar

Under the title bar is the menu bar shown in Fig. 4-3.

File Work Mode Operate Machine Tool Window Help

Fig. 4-3 Menu bar

The menu bar includes several concealed pull-down menus; each pull-down menu is composed of several menu items; each menu item can realize a special function, action, or state that is relative to a certain procedure. To implement a function, an action, or to change the state setting, user only needs to select the relative menu item. Menu selection can also be realized by mouse or by keyboard.

On menu bar, there are menu of "file", "work mode", "operate", "machine tool", "window" and "help", through which user can implement conveniently the action of processing file, mode selection, machining operation, machining relative operation, windows exchange, getting help information, and so on.

◆ **Mouse operation**

Click the menu on the menu bar, and a pull-down menu will pop up. Click the target item.

◆ **Keyboard operation**

Press down 'Alt' key and hot letter key simultaneously. (For example, there is a line under 'F' in the "File" menu, so the combination key is "ALT+F"). After that, a relative pull-down submenu will pop up. Regarding the pull-down submenu, user can also use keyboard to choose target item. For instance, [File] → [Open and Load]. When the pull-down menu of [File] appears, directly press the letter 'O' to select the item.

◆ **Shortcut Key operation**

In pull-down submenus, some items have shortcut keys on their right. For example, [File] → [Open and Load], its shortcut key is 'Ctrl + O'. That is to say, user can directly select the target item by the shortcut key, which can avoid the trouble to enter into multilayer menus.

Some menu options have three dots behind. For example, [File] → [Open and Load…], it indicates that a dialog will automatically pop up when user selects the menu option. If the item of the submenu shows in grey, it indicates the item is unavailable under current state.

In addition, right click on any position, a corresponding shortcut menu will pop up, then choose the relative item.

4.4 NC(numerical control) state bar

Under the menu bar is the NC state bar. Please see Fig. 4-4.

AUTO   IDLE   G54   machining time   00:00:00   **Standard Config Model A**

Fig. 4-4 The NC state bar

The embedded prompt boxes shown on the NC state bar are operation mode, operation state, other appended information, and configuration information. In the middle-right, there is processing

time. It gives user an intuitional way to get the current machine state and the processing time, and then decide the next operation.

4.5 Information prompt bar

Under the NC state bar is the prompt bar, shown in Fig. 4-5.
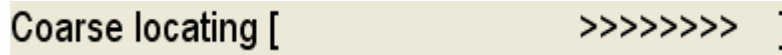
Coarse locating [>>>>>>>>]

Fig. 4-5 Information prompt bar

On the prompt bar, three kinds of information can be shown. They are normal prompt, warning prompt and error prompt information.

They are distinguished by color:

➢ In grey theme interface, if the color of the prompt bar is the same as that of the current interface and the inside characters show in black, the current prompt is a normal prompt; if the color of the prompt bar is yellow and the characters show in black, the current prompt is a warning prompt; if the color of the prompt bar is red and the characters show in white, the current prompt information is an error prompt.

➢ In black theme interface, if the color of the prompt bar is the same as that of the current interface and the inside characters show in white, the current prompt is a normal prompt; if the color of the prompt bar is yellow and the characters show in black, the current prompt is a warning prompt; if the color of the prompt bar is red and the characters show in white, the current prompt information is an error prompt.

Note:

The normal prompt mainly shows the information relating to the normal running of the system.

The warning prompt mainly shows the information that needs to remind user. For example, if the port hasn't been configured before operation, the prompt bar will show 'The port has not been configured', and so on.

The error prompt mainly shows that system errors occur and operation can't continue, such as, ESTOP warning, limit warning, file error, software error, and so on.

4.6 Function window

Function window is the main component of Ncstudio V9 interface. It is used to display different kinds of function screens, including six powerful function sections and button columns of screen manipulation, such as [POS], [OFFSET], [PROG],[SYS], [PARAM], and [DIAG]. Press down any button to enter into the corresponding function section. See Fig.4-6.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)

Fig. 4-6 Function selection button

Each function section has its relative function screens. For example, under [OFFSET] function section there are screens of [WorkCoor], [Offset], etc. Under [Program] function section there are screens of [HD List], [FD List], [ProcessWiz] and [History], etc. To exchange between function

screens, please press function exchange button to select the corresponding button. See Fig. 4-7.

Fig. 4-7 Function exchange button

Note:

Screens of [POS] function section vary with different modes (such as auto mode, manual mode, and reference point mode).

After starting Ncstudio, the default window is "Normal" function window.

4.7　Manipulation button column

Under the function window is the picture manipulation button column. See Fig. 4-8.

Fig. 4-8 The picture manipulation button column

What it usually shows is the shortcut buttons relating to the current function screen or the current mode. For instance, under the function screen of [PROG] → [HD List], the picture manipulation button column has buttons of "load", "edit", "delete", "new", "rename", and so on.

User can choose the picture manipulation button to do the relevant operations according to the content displayed on the function screen.

Note:

When the conditions are not satisfied, some buttons in the picture manipulation button column may be prohibited in order to prevent user's false operation.

4.8　Status bar

Status bar is at the bottom of the screen. See Fig. 4-9. At the right of the status bar is a prompt which is used to show user the current date, the current time, and the current state of the capital lock, the number lock, and the scroll lock.
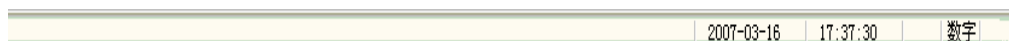
Fig. 4-9 The status bar

4.9　Operation panel

According to the differences in mode, operation panel can be divided into automatic mode, manual mode, and reference point mode. See Fig. 4-10. Above the operation panel there is a mini area for coordinate display where user can see the workpiece coordinate and the machine coordinate of the current cutter position, the current feed speed, the current spindle rotation speed, and the name of the current loaded file. Under the mini coordinate display area is the panel button area, inside which buttons are different under different modes.

AUTO (7)	MANUAL (8)	REFER (9)

Axis	Workcoor	Machcoor
X	0.000	0.000
Y	0.000	0.000

Feedspeed F(actual/setting): 0/1200
Current Tool No.: 1

Feedrate (B)
[100%]   0%   120%

S. C. (N)
[50%]   0%   100%

G00F.R. (M)
[100%]   G00 Feedrate   100%

9 Z+	Booster	H/L	H.P.
1 Z-	Cutting	Abrasive	Simulate

F9 Start	F10 Pause	F11 Stop	Pause Reset
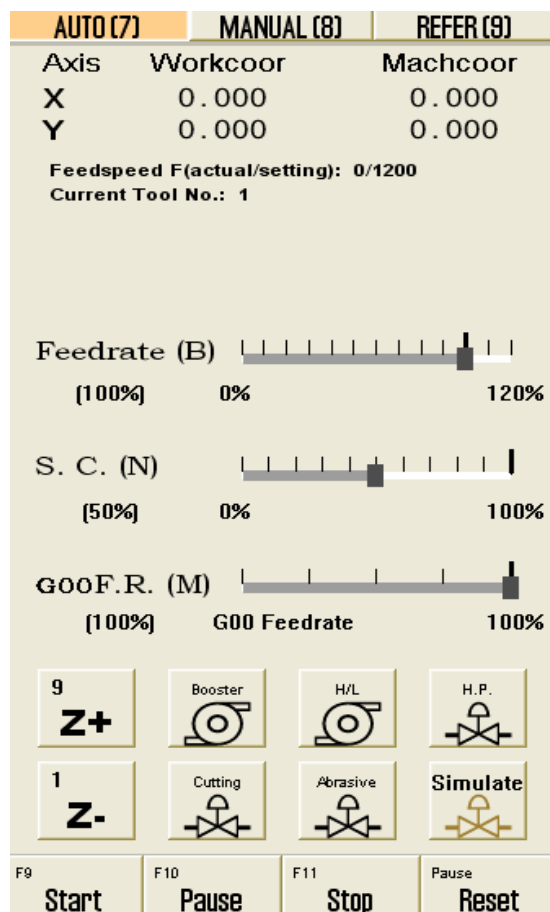
Fig. 4-10 Operation panel

5 Auto Mode of Ncstudio™

In auto mode, system can automatically analyze the loaded processing program to control the machine tool on machining the object workpiece. Advanced functions can also be fulfilled, such as breakpoint resume, selection processing, and handwheel guidance.

The picture of auto mode is shown in the Fig.5-1, including operation panel and function panel, as following.
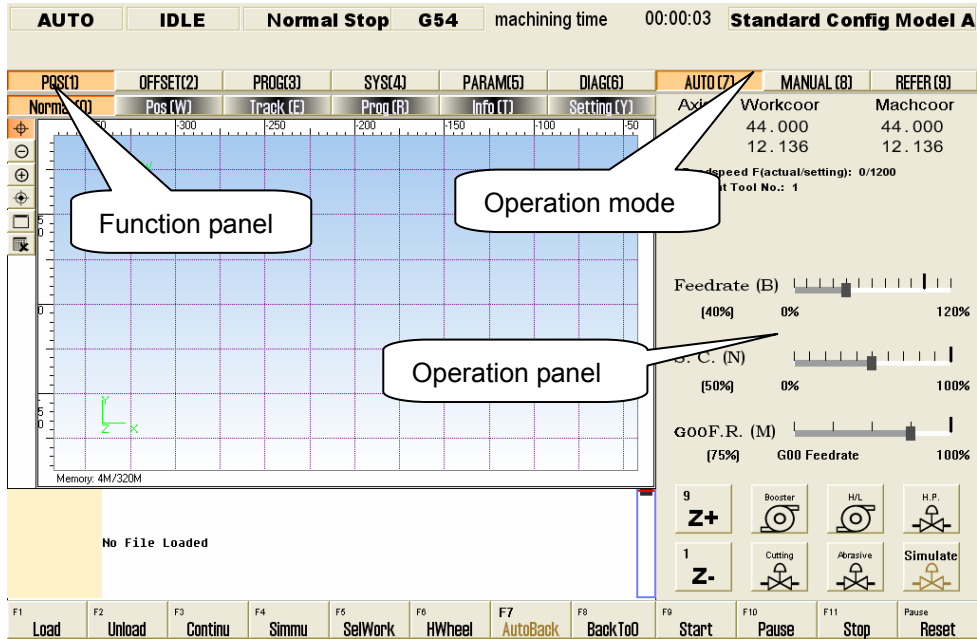
Fig. 5-1 Auto operation mode

5.1　Operation panel section

This operation panel is composed of several auto manipulation buttons, providing an alternant operation environment for manipulating machine tool automatically. See Fig. 5-2.
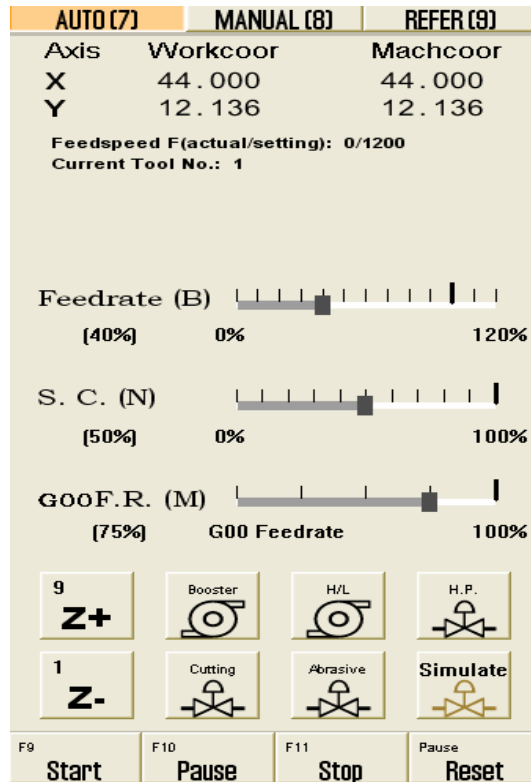
Fig. 5-2 Auto operation panel dialog

On the top of the panel, there is a coordinate display zone, which is used to display the current coordinate of X-axis and Y-axis, feedrate and the name of current file. Under the mini coordinate display area is the panel button area, inside which buttons are different under different modes.

Under auto mode, button zone includes the following contents:

◆ **[Feedrate] sliding button**

Fig. 5-3 Feed ratio sliding button dialog

User can manipulate the feed speed by controlling the feedrate. The formula is as following:

Current feed speed = rated feed value × current feedrate.

The least unit of feedrate scale is 10% (Each case indicates 10%). Current feedrate percentage is displayed beneath the word "Feedrate". Three ways can be used to revise the feedrate percentage. Click the sliding axle; drag the slide block; click the slide block to choose it, then press [PageUp], [PageDown] key; when [PageUp] or [PageDown] key is pressed once, the block will move one case, and the feedrate increases (or decreases) 10%. The range of feedrate percent is 0% ~ 120%.

◆ **[S.C] sliding button**

See Fig. 5-4.

Fig. 5-4 Sand valve control dialog

The main function of sand valve control is to control the start-up and shutdown of the abrasive, and the flow discharge of sand-jet.

◆ **[G00 F.R.] sliding button**

See Fig. 5-5.
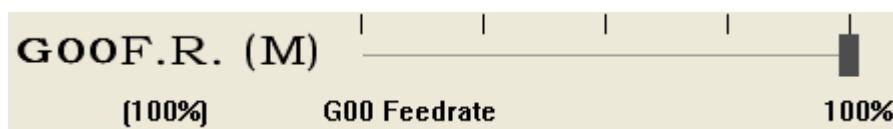
Fig. 5-5 G00 Federate sliding button dialog

User can manipulate the speed of G00 instruction by modifying G00 feedrate. If G00 feedrate is adjusted to 0%, G00 rate = feedrate percent, and the prompt will be displayed that the "G00F.R." is equal to "feedratio". The formula is as following:

Current feed speed under G00 instruct = rated feed speed × current machining speed.

When G00 rate is not equal to 0%, its calculating expression is as below:

Current feed speed under G00 instruct = G00 rate × dry running speed

The least unit of G00 rate on the sliding axle is 25% (Each case indicates 25%). Three ways can be used to revise feedrate percent. Click the sliding axle; drag the slide block; click the slide block to choose it, and then press [PageUp], [PageDown] key; each time press [PageUp] or [PageDown] key, the block will move one case, and the feedrate increases (decreases) 25%. The revised range of G00 ratio is 0%-100%.

◆ **[Z+], [Z-], [Booster], [H/L], [H.P.],[Cutting],[Abrasive],[Simulate]**

See Fig. 5-6.
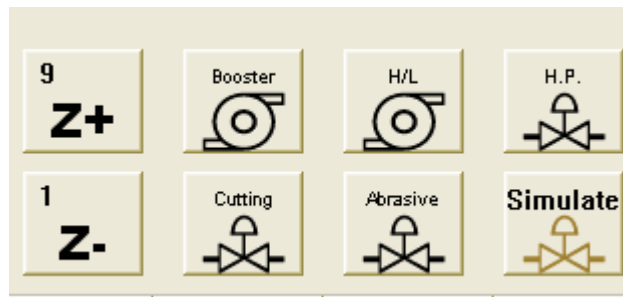
Fig. 5-6 Buttons of [booster], [cutting], [H/L], [abrasive], [H.P.], [simulate] on operating panel

These buttons are used to start or stop water pump, lubricating pump, high pressure valve, water valve, abrasive valve and simulation.

➢ Under grey theme, button's background is grey and the letters show in black. When user presses down the button, its background turns into green and the letters show in black. User can distinguish the button's state according to the color of button's background and letters. For example, press down button [Booster] (Water Valve), its background will turn into green, and press the button again, it will rise and its background will restore to grey.

➢ Under black theme, button's background is black and the letters show in white. When user presses down the button, its background turns into green and the letters show in white. User can distinguish the button's state according to the color of button's background and letters. For example, press down button [Booster] (Water Valve), its background will turn into green, and press the button again, it will rise and its background will restore to black.

◆ **[Start]**

See: ; Hotkey: F9, only available under AUTO mode.

➢ If there is no loaded processing file, the button is disabled;

➢ When user has loaded a processing file and operation of [back to the reference point] has been finished, the button is enabled. Press the button to start machining automatically;

➢ If user has loaded a processing file for the first time and hasn't executed related operations of the reference point ("directly setting" or "back to mechanical origin"), system will remind user to go back to the reference point first before operation.

The button has two functions:

➢ If a certain processing procedure has been loaded and backing to the reference point has been finished, and system's current state is "idle", press the button to carry out the procedure automatically from the first section. Once machining starts, system will enter into "Auto | Running" state. If system is under simulation, system will start simulation machining.

➢ If system is under "Auto / Pause" state, press the button, system will resume machining from the suspended position and enter into "Auto | Running" state. If system is under simulation, system will start simulation machining.

◆ **[Pause]**

See: F10 **Pause**

[Pause] button is valid under auto processing. Press the button, machine tool will pause and uplift the nozzle and then enters into "Auto | Pause" state. If user wants to continue machining, press button [Start] or choose the relative menu item.

If system is now under simulation state, press the button, simulation will be paused and system will enter into "Auto | Pause" state. If user wants to continue simulating, press button [Start] or choose the relative menu item.

Hotkey: F10, only available under AUTO mode.

Note:

System has two ways to enter into [Pause] state:

1. When system is executing single block function;

2. User selects [Pause] function during machining process.

◆ **[Stop]**

See: F11 **Stop**

This function is enabled under the auto mode or the input of effective G code by user. Press the button, the machine will stop and uplift the nozzle to stop all the machining tasks, then system enters into "Auto | Idle" state. This button is to break off the machining procedure normally during machining.

Hotkey: F11, only available under AUTO mode or user inputting effective G code state.

◆ **[Reset]**

See: Pause **Reset**

[Reset] button is enabled under every situation. Press the button, the machine will stop the current machining task (if it is machining) and restore from warning (if it is in E-stop status) state to "Idle" state.

Using [Reset] button to stop is different from using [Stop] button. When [Reset] button is pressed,

the state of water valve and high pressure will not be changed, and there is no deceleration or nozzle uplift.

Hotkey: F12, available under any mode.

5.2   Function panel section

Under auto mode, the function panel buttons are as following:

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)

Fig. 5-7 Position function screen under AUTO mode

5.2.1 POS function section

Under AUTO mode, [POS] subfunction section includes: Normal, Pos, Track, Prog, Info, and Setting. See Fig 5-8 below:

Normal (Q)	Pos (W)	Track (E)	Prog (R)	Info (T)	Setting (Y)

Fig. 5-8 Position subfunction buttons under AUTO mode

◆   **Normal function screen**

See Fig.5-9.
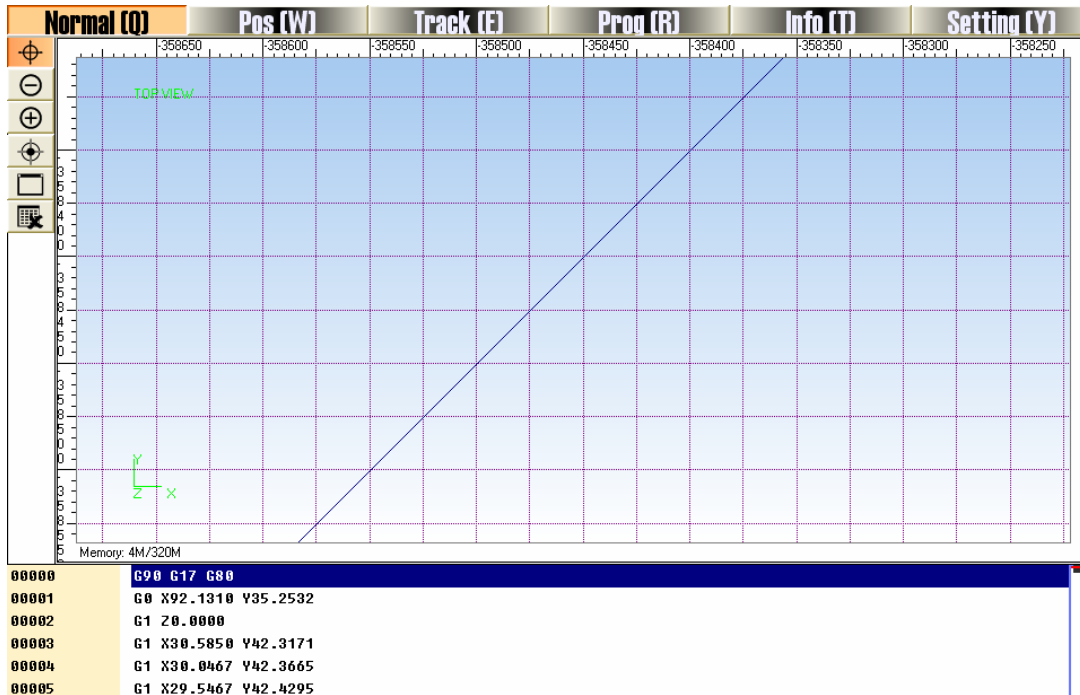
Fig. 5-9 Normal function screen

In the process track window, the path of water jet is displayed in real time. User can check water jet's track intuitively and make sure the machining procedure executed accurately. Besides, under this two-dimension track mode, user is offered abundant operating methods to look over the process figure from different position and scaling, such as zoom in, zoom out, etc.

◆   **[Icon button]**

On the left of the process track window, there are icon buttons, see Fig.5-10

Fig. 5-10 Icon buttons on the left of the process track window

V9 Ncstudio adds a tip function for each of these icon buttons. Move mouse onto the button, a tip will display. It tells user the button's function.

◆ **[Move]**

Click the icon( ) on the left of the process track window, move cursor to the display window, the cursor will show as , press down the mouse's left key, the cursor turns into , drag the path, move it to the suitable position that you want, and release the mouse.

User can also implement this function by keyboard. After pressing down the [Move] icon , press [enter] key, then press the direction key to move the whole path. When the mini-keyboard is unlocked, the mini-keyboard can also be used as direction key.

> Note:
> The operation is very convenient. User can greatly raise his working efficiency if he practices a lot.

◆ **[Zoom In]** and **[Zoom Out]**

Click the [Zoom in] or [Zoom out] button on the left of the process track window, then move the cursor into the display area. When the cursor shows , click the left key of the mouse, the figure will be enlarged; when the mouse shows , click the left key of the mouse, the figure will be diminished.

Shortcut: on mini-keyboard, [-] key can zoom out the figure and [+] key can zoom in the figure, no matter whether the mini-board is locked or not.

◆ **[Show Current Processing Point]**

Click [Show Current Processing Point] on the left of the process track window. The current processing point will be displayed in the center of the display window automatically.

Shortcut key: [End] key on the keyboard.

◆ **[Adjust to Window Size]**

Click the button [Adjust to Window Size] on the left of the process track window. The process track window will adjust to displaying all the paths and make path center in its midpoint. With this method, user can see the whole figure and needn't drag the scroll bar.

Shortcut key: [/] key on the keyboard.

◆ **[Clear (Delete)]**

After a long time of machining, the simulation figure will become very complex and the temporary folder used to record processing path will become more and more bulkiness. It will spend a lot of time to redraw, move, or revolve the simulation figure. At this time, user needs to clean the tracking window. The actual processing track is cleared after user pressing the button. Click the button again, the simulation figure in the processing track window will be deleted.

Shortcut key: [Delete] key on the keyboard.

◆ **Shortcut menu**

Right click in the processing track window, and a shortcut menu will pop out. Click the target item to implement the corresponding function. See Fig.5-11.

Fig. 5-11 Quick menu

The function of these quick menus will be illustrated separately as following (the same functions mentioned in the icon button will be omitted).

◆ **[Clear]**

It has the same function with button (Clear/Delete), which deletes the actual processing track. Hotkey: [Delete] key on keyboard; or, right click to eject the QuickMenu, and press letter [T] key.

◆ **[Center]**

The function is used to set the machining path at the center of the window. Hotkey: [Home] key on keyboard; or, right click to eject the QuickMenu, and press letter [N] key.

◆ **[Fit to Window]**

It has the same function with button (Adjust to Window Size). Hotkey: [/] on mini-keyboard; or, right click to eject the QuickMenu, and press letter [F] key.

◆ **[Show Current Point]**

It has the same function with button (Show Current Processing Point). Hotkey: [End] key on mini-keyboard; or, right click to eject the QuickMenu, and press letter [P] key.

◆ **[Customize Track View]**

Right click to eject QuickMenu, choose [Customize Track View], a "Custom Parameter" dialog will appear. See Fig. 5-12 below. User can choose the track color, and decide whether to show the travel range, draw the gridding, and so on. The detailed information will be introduced as below.
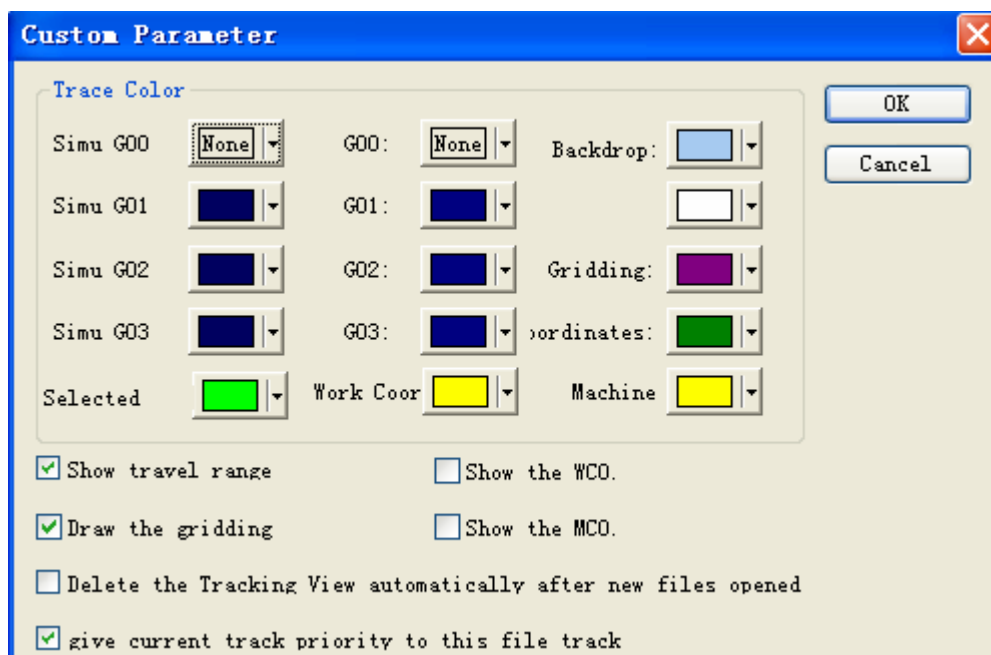
Fig. 5-12 Customized parameters

Look at the above dialog. On [Trace Color] board, user can set the color of backdrop, gridding, coordinate axis, the selected track, workpiece coordinate origin and machine origin. In V9 Ncstudio, a new function of loading path in advance has been added, so the path colors are divided into simulation path color and real path color. [Simu G00/G01/G02/G03] is corresponding to simulation path color, while [G00/G01/G02/G03] is corresponding to real path color.

◆ **[Show travel range]**

This option is used to decide whether to show the simulation worktable in process track window. If user selects this item, the simulation worktable will be displayed; if not, the simulation worktable won't be displayed.

◆ **[Draw the gridding]**

This option is used to decide whether to show the gridding in process track window. Once user selects this item, the gridding will be displayed. Otherwise it won't be displayed.

◆ **[Delete the Tracking View automatically after new files opened]**

This option is used to decide whether to delete the real path automatically after a new processing file is opened. Once user selects this item, the actual path will be deleted automatically. Otherwise it won't be in effective.

◆ **[Give current track priority to this file track]**

After user loading a processing file and starting machining, the machining path will be all displayed in process track window. But functions of [Center] and [Adjust to Window Size] are only effective to the current path. Select this item, simulation path will be set as the current path, or the real machining path will be the current path.

◆ **[Show the WCO]**

It decides whether to show the workpiece origin in process track window. Selecting this item means the workpiece origin will be displayed, otherwise it won't be displayed.

◆ **[Show the MCO]**

It decides whether to show the machine origin in process track window. Selecting this item means the machine origin will be displayed, otherwise it won't be displayed.

On the lower part of the Normal Function Screen, there are operation buttons, machine status display, and running state of processing file which are relative to the current operating mode.

Under [AUTO] mode, the lower part of Normal Function Screen displays the loaded processing file. When auto machining begins, the current executed instruction will be emphasized with deep blue. See Fig. 5-13. The frame in front of the instruction is used to mark the current instruction's position relative to the whole processing file.

00000	N3 G43 H03 Z10. S15000 M03
00001	N4 G00 X43.564 Y-27.737 Z10.
00002	N5 Z0.2
00003	N6 G01 Z-0.95 F105
00004	N7 G02 X41.364 Y-25.537 J2.2 F350
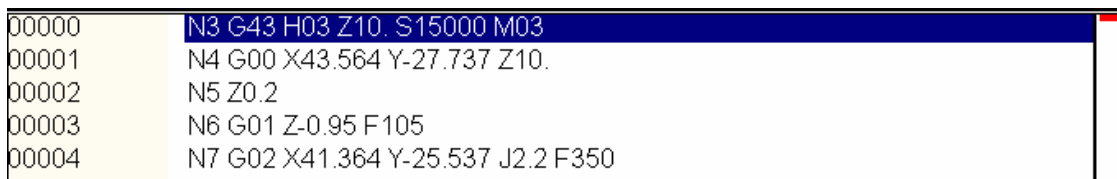
Fig. 5-13 The lower part of the normal function screen under AUTO mode

◆ **POS function screen**

See Fig. 5-14. This screen is used to look over the position of current machining, including machine coordinate, workpiece coordinate, and remaining distance. When current G code is executed, the absolute distance between current position and destination's position is called remaining distance, only available under AUTO mode.

POS[1]	OFFSET[2]	PROG[3]	SYS[4]	PARAM[5]	DIAG[6]
Normal (Q)	Pos [W]	Track (E)	Prog (R)	Info (T)	Setting (Y)

[Position Display]

In this window, you can clear see the instant position when processing.

Axis	WorkCoor(G54)
X	−364049.133
Y	−364049.133

MachCoor	Remain
−364049.133	0.000
−364049.133	0.000

Fig. 5-14 Pos function screen

◆ **Track function screen**

It is used for displaying machining paths. Buttons on the left of the screen is totally the same as those of Normal Function Screen. Please refer to Normal Function Screen for detailed information.

◆ **Program screen**

This screen is mainly used for displaying the loaded file. The left column is used for marking the current instruction's position in the whole processing file. Please refer to Fig. 5-15. [Edit Loaded File] button in the lower part of the screen is only available when there is a file loaded. Press the button, system will eject NcEditor, in which user can edit the file.

POS[1]	OFFSET[2]	PROG[3]	SYS[4]	PARAM[5]	DIAG[6]
Normal (Q)	Pos [W]	Track (E)	Prog (R)	Info (T)	Setting (Y)

```
00000    G90 G17 G80
00001    G0 X92.1310 Y35.2532
00002    G1 Z0.0000
00003    G1 X30.5850 Y42.3171
00004    G1 X30.0467 Y42.3665
00005    G1 X29.5467 Y42.4295
00006    G1 X29.0494 Y42.5109
00007    G1 X28.5890 Y42.6049
00008    G1 X27.6848 Y42.8519
00009    G1 X26.9153 Y43.1443
00010    G1 X25.5284 Y43.9670
00011    G1 X24.5350 Y44.9780
00012    G1 X24.1456 Y45.5867
00013    G1 X23.8285 Y46.2724
00014    G1 X23.6012 Y46.9940
00015    G1 X23.4442 Y47.8184
00016    G1 X23.3741 Y48.6560
00017    G1 X23.3824 Y49.6161
00018    G1 X23.4165 Y50.0961
00019    G1 X23.4723 Y50.6093
00020    G1 X23.5460 Y51.1202
00021    G1 X23.6428 Y51.6654
00022    G1 X22.2491 Y92.5248
```

Edit Loaded File (J)

Fig. 5-15 Program function screen

◆ **Info function screen**

See Fig.5-16. Information Function Screen is mainly used for displaying various information about machining, such as the maximal processing range, cutting time, workpieces count, hole count,

current file line, file length, file path, and incising length. Incising time is the total time of water jet causing deformation to the workpiece. Workpiece count means the times of completing the file and pressing its button will clean the count number. Hole count means the number of the holes to complete punching marching and pressing its button will clean the count number. Incising length records the whole length of waterjet incision path. All the information is clear at a glance.
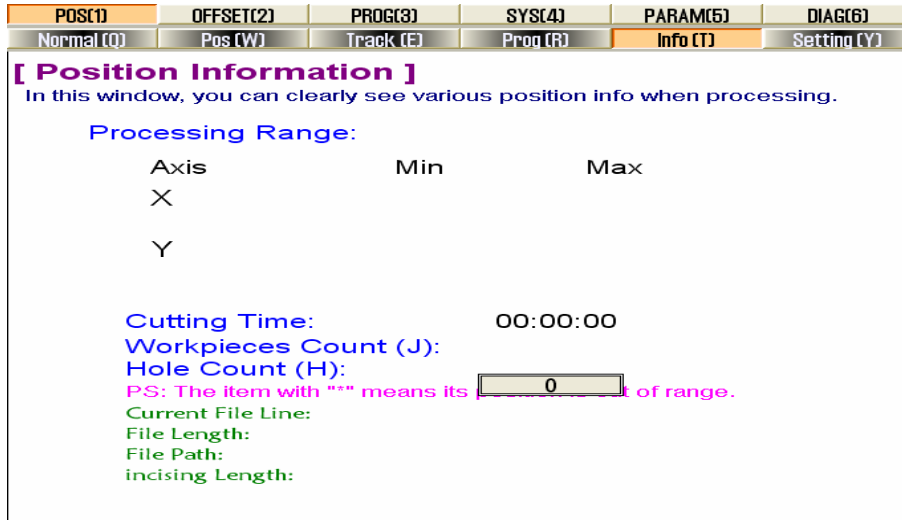
Fig. 5-16 Information function screen

◆ **Setting function screen**

Setting function screen is mainly used for setting parameters in common use in AUTO mode, such as dry running speed, processing speed, speed mode specified, cycle times of program, interval of time between machining file cycle and quality-efficiency regulation. See Fig. 5-17.
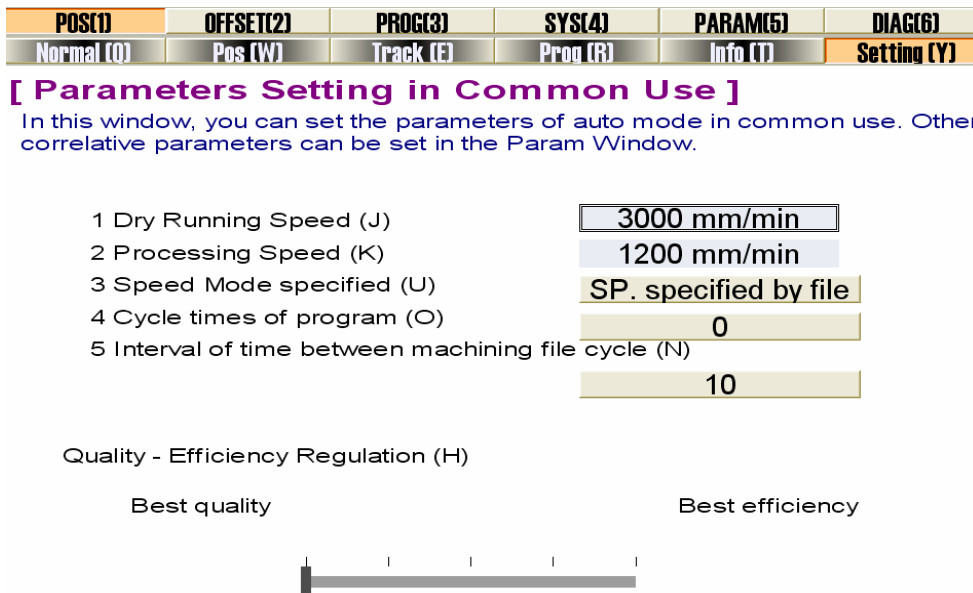
Fig. 5-17 Setting function screen

◆ **Dry running speed**

Dry running speed refers to the running speed under G00 command, and its range is 1~99999.999. If it is out of the range, system will prompt user. Press the button, system will eject an

Input Box, shown in Fig 5-18. It has functions of [copy], [paste], [expression calculation], and even supports hotkey. User will find it is very convenient, especially when disposing multi-digit number or calculating data.
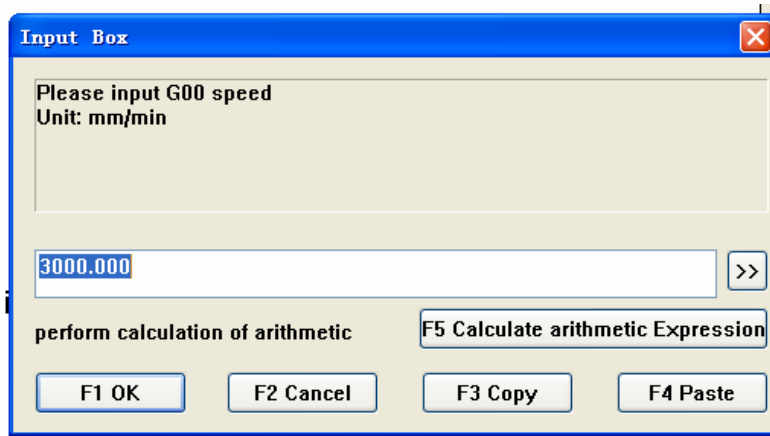
Fig. 5-18 Input box

◆ **[Processing Speed] (feed speed)**

Processing speed refers to the moving speed under G01, G02 and G03 command. Its range is 1~ dry running speed. If it is out of the range, system will prompt user.

◆ **[Speed Mode Specified]**

[Speed mode specified] refers to whether to ignore the feed speed specified by the processing file and use the default feed speed in the system or the specified speed in scale. Press the button, to eject an input dialog. User can modify the parameter according to the prompt. Please see Fig.5-19.
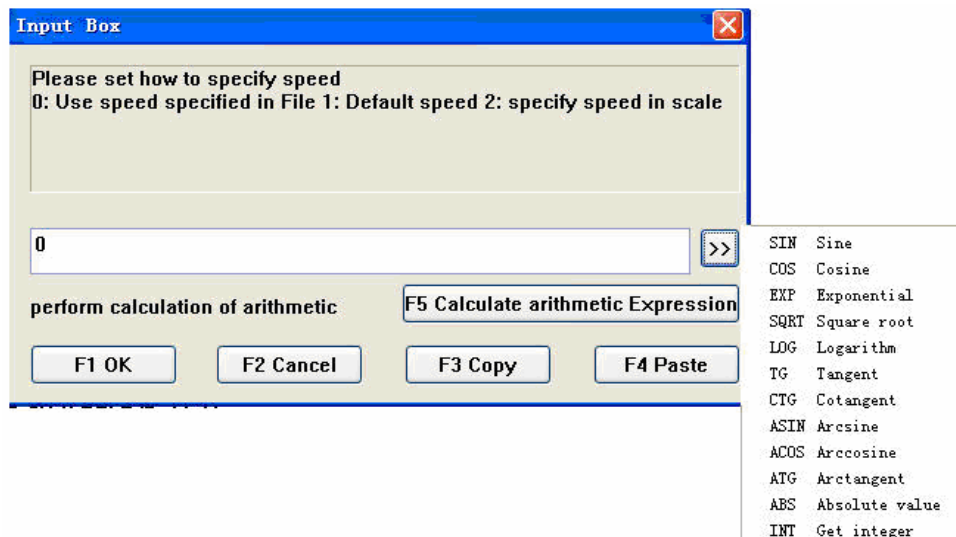
Fig. 5-19 Speed mode specified input dialogue

◆ **[Cycle Times of File Processing Task]**

This parameter is to specify the cycle times of execution of loading files.

◆ **[Interval of Time between Machining Time Cycle]**

This parameter is to specify the interval time between the end of circle processing and the beginning of next circle processing.

◆ **[[Quality-efficiency Regulation]**

The relationship between quality and efficiency is relative, but there is no concrete ratio; when the processing efficiency is high, the processing quality will be relatively low, and vice versa.

◆ **Manipulation button bar**

Under AUTO mode, all picture manipulation button columns in POS Function Screen are the same, as shown in Fig 5-20.

F1 Load	F2 Unload	F3 Continu	F4 Simmu	F5 SelWork	F6 HWheel	F7 AutoBack	F8 BackToO

Fig. 5-20 Manipulation button bar in POS function screen under AUTO mode

◆ **[Load]**

Hotkey: F1, it is only available under AUTO mode. Pressing down the button will eject "Open and Load" dialog, choose the target file, click [Open] button, and system will load it into parsing engine for machining.

◆ **[Unload]**

Hotkey: F2, it is only available under AUTO mode. Its function is to uninstall the loaded file from Ncstudio system.

◆ **[Continu]**

Hotkey: F3, it is only available under AUTO mode. This button is only available when power-off or E-stop occurs, or when user presses [Stop] button during machining process. If user can insure the accuracy of workpiece coordinate, user can press this button to continue machining. If not, please go back to the reference point first and then restart the machining. This function can also be applied to simulation mode.

◆ **[Simmu]**

Hotkey: F4, it is only available under AUTO mode. Simulation function provides user with a fast but lifelike simulation environment. After loading a processing file, press [Simmu] button and then press [Start] button on the operation panel, system will carry out simulation automatically with high speed from the first section. Simulation is similar to demonstration function of CNC system, but more advanced. To run a machining file under simulation mode, system will not drive the machine tool to do the relative actions, but only show the water-jet trace with high speed in the processing trace window. By simulation, user can know machine's movement in advance, avoiding damage (to the machine tool) which is caused by mistake in writing procedure, and can get other additional information, which user can look over "Info" function screen under POS function section.

◆ **[SelWork]**

Hotkey: F5, it is only available under AUTO mode, used for program jump execution. Press down the button to eject an "Execute (Advanced Options)" dialog box, as shown in Fig. 5-21. Choose start

position and end position in current machining program, then click "OK" button, the machine tool will carry out the selected block as requirement. When user selects "File Start" or "File End", "Specific Position" will be disabled. This function can also be used for simulation.
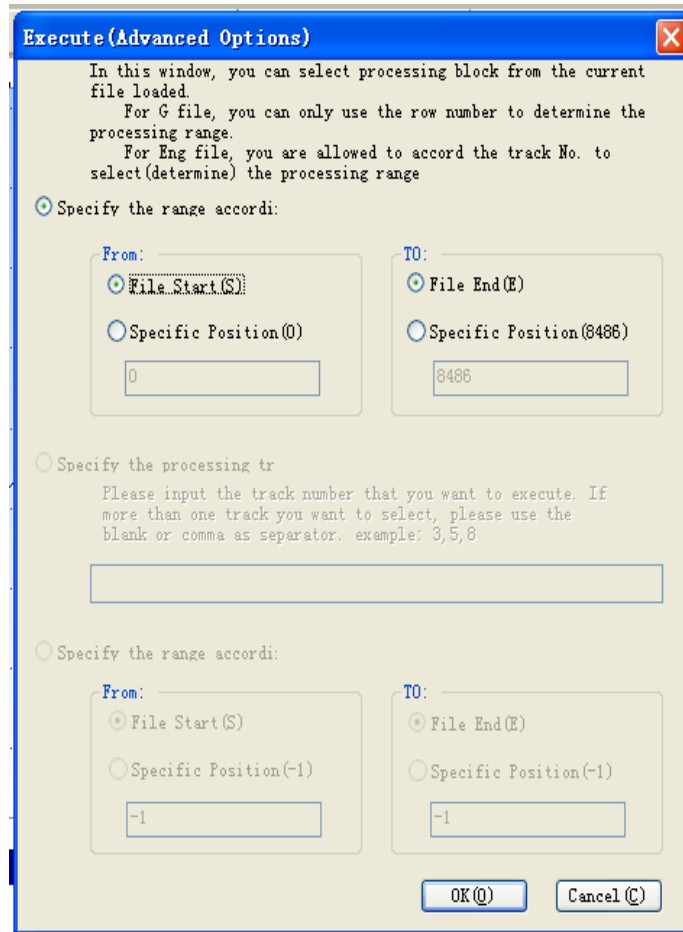
Fig. 5-21 Select program block to execute

◆ **[HWheel]**

Hotkey: F6, it is available only under AUTO mode. Press button [HWheel], user can control machine tool only by MPG; and then all of the buttons on operating panels are invalid.

◆ **[AutoBack]**

Hotkey: F7, it is only available under AUTO mode. After the button is pressed, a dialog "Auto Back" will pop up, and the machining program will be executed in reversed order. If user wants to change to "Auto Forward" or "Stop" state, please press the corresponding button on "Auto Back" dialog; if user closes the dialog, auto back state will be automatically substituted by auto forward state. See Fig.5-22.
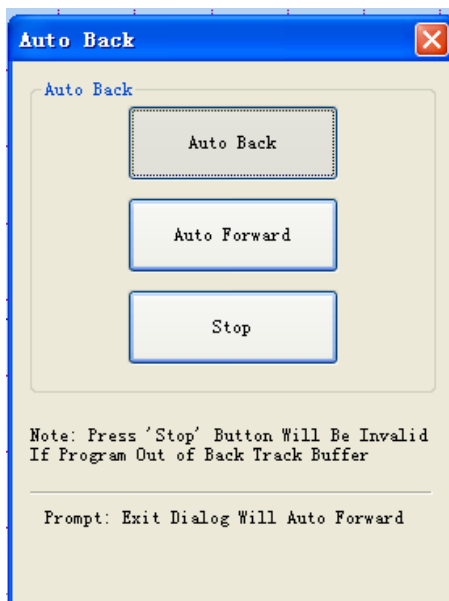
Fig. 5-22 Auto back dialogue

◆ **[Back To 0]**

Hotkey: F8, it is only available under AUTO mode. Press the button, and system will control water jet to go back to the workpiece origin. Please pay attention to the difference among workpiece origin, fixed point and machine origin (the reference point).

5.2.2 OFFSET function section

These screens are mainly used to deal with the related operations of workpiece coordinate offset. Offset function section includes [WorkCoor], [OffSet], and [S. and L.] function screen.

◆ **WorkCoor function screen**

See Fig.5-23. In this screen, user can set the current workpiece coordinate. Click the button on the screen to eject a dialog "Input Box", input coordinate value and click [OK] button, system will adjust the current workpiece coordinate to the input value automatically, and prompt user the moving distance and direction of the water jet. But it won't cause any actual movement of the current position.

Fig. 5-23 Coordinate function screen in OFFSET function section

"Input Box" is as Fig.5-24.
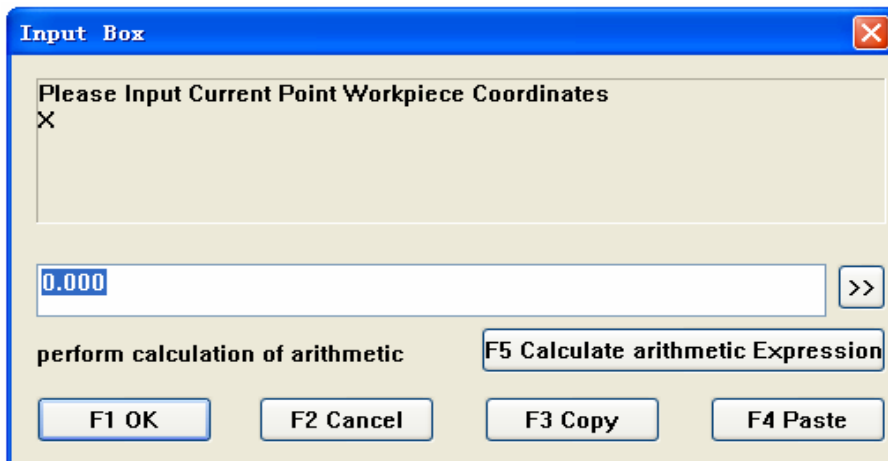
Fig. 5-24 "Input Box" for workpiece coordinate adjustment

◆ **Offset function screen**

Offset function screen is as shown in Fig. 5-25. It is used to set outer offset and the offset value between workpiece coordinate and machine coordinate for G54、G55、G56、G57、G58、G59.
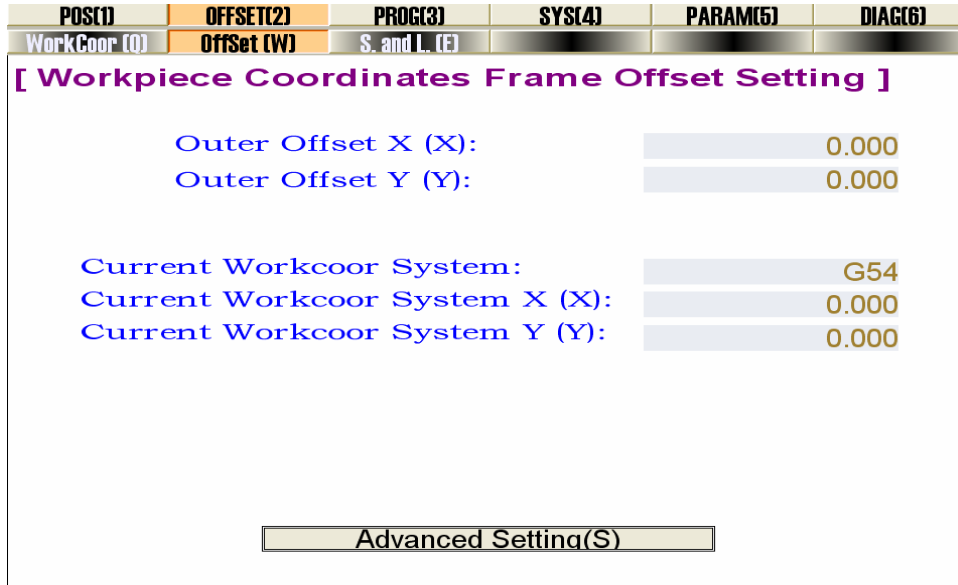
Fig. 5-25 Offset function screen in OFFSET function section

Final offset value between workpiece coordinate and machine coordinate = Outer offset value + G instruction offset value (including G54、G55、G56、G57、G58、G59) + water jet offset value

Click the "Advanced Settings" in the dialogue, the offset setting can be specified. See Fig.5-26.
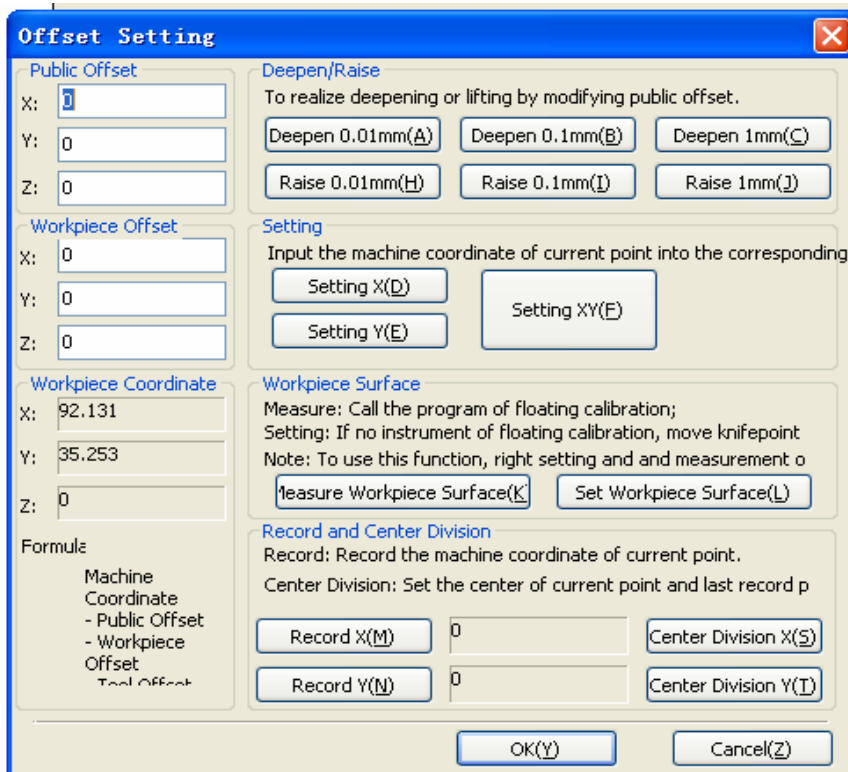
Fig. 5-26 Offset setting dialogue

◆ **S. and L. function screen**

The screen is as shown in Fig 5-27. User can save and load the offset value of workpiece coordinate in this screen. Each group is identified by group no., file name, saved time. When saving

or loading the record, press the corresponding button, system will eject a dialog box to prompt user.
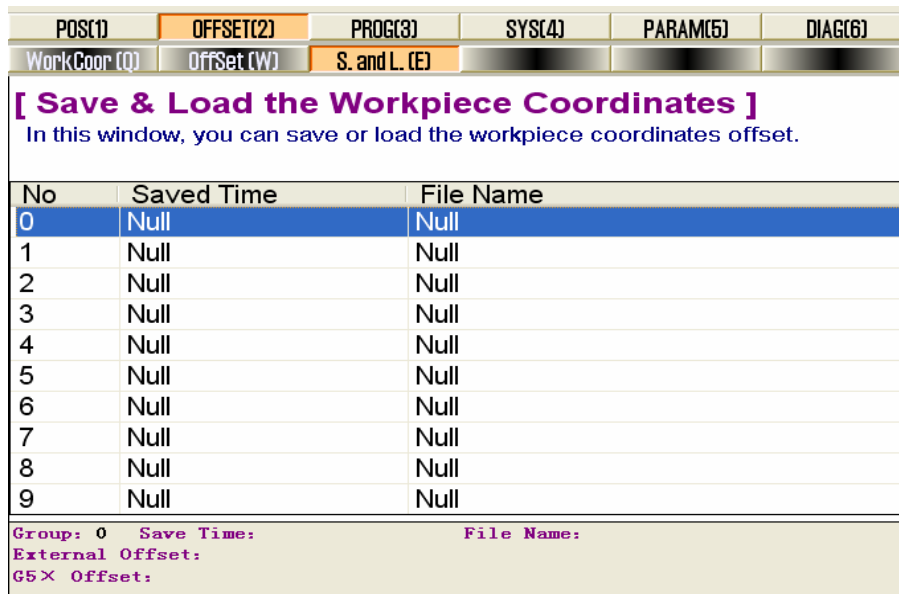
Fig. 5-27 Save and load function screen

◆ **Manipulation button bar**

The manipulation button in [OFFSET] function under AUTO mode is a little bit different, which will be introduced separately.

The manipulation button in [WorkCoor] and [OffSet] function screen is as Fig. 5-28.

Fig. 5-28 Manipulation button in [Offset] and [WorkCoor] function screen

◆ **[X Clear]**

Shortcut Key: F1. Click this button, and system will eject a dialog box to prompt whether to modify the offset to ensure that X-axis workpiece coordinate value of current point is zero.

◆ **[Y Clear]**

Shortcut Key: F2. Click this button, and system will eject a dialog box to prompt whether to modify the offset to ensure that Y-axis workpiece coordinate value of current point is zero.

◆ **[XY Clear]**

Shortcut Key: F4. Click this button, and system will eject a dialog box to prompt whether to modify the offset to ensure that workpiece coordinate values of both current Y-axis and X-axis of current point are Zero.

◆ **[Back To 0]**

Shortcut Key: F8. Press the button, and system will control water jet to go back to the workpiece origin automatically. Please pay attention to distinguishing among workpiece origin, fixed point and mechanic origin.

The picture manipulation buttons in [S. and L.] function screen is as shown in Fig. 5-29.

Fig. 5-29 Manipulation button in [S. and L.] function screen

◆ **[Save]**

Hotkey: F1, only available in the [S. and L.] function screen of OFFSET function section. It is used for saving offset value.

◆ **[Load]**

Hotkey: F2, only available in the [S. and L.] function screen of OFFSET function section. It is used for loading offset value.

5.2.3 PROG function section

These screens are mainly used to deal with the related machining operations. [PROG] function section includes [HD List], [FD List], [ProcessWiz], and [History] function screen, which will be introduced separately.

◆ **[HD List] function section**

As Fig. 5-30, in this screen, user can look up processing files under default file path on hard disk, and carry out a series of operations, such as load, edit, delete, rename, etc. Moreover, user can create a new file under default file path D:\ NCFILES and compile it.
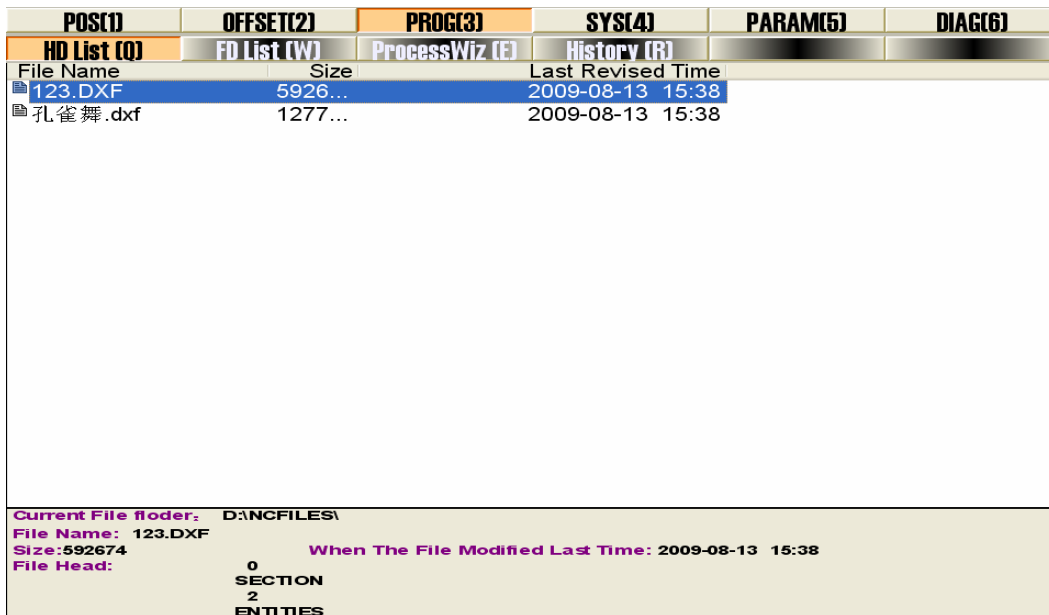
Fig. 5-30 [HD List] function screen

The upper part of the screen is a file list, which displays all of the files under the path "D:\ NCFILES". User can save the processing files under the path to make it more convenient to operate.

The lower part of the screen is a prompt frame. After user selecting a file in the list, its attributes will be shown in the prompt frame, such as current file folder, file name, size, and the first four lines in

the file.

> Note:
>
> User can save the machining files under the folder D:\ NCFILES, then the files will be loaded into the HD list automatically.

◆ **[FD List] function section**

As Fig.5-31, in this screen, user can load the files in floppy dick or in USB flash disk. For instance, user can execute a series of operations to a file in the floppy disk (Drive: "I"), such as [load], [copy to HD], [select drive], and [safely remove].

Fig. 5-31 [FD List] function screen

The upper part of the screen is a file list. It displays all of the files in floppy disk or in external memorizer. User can execute a series of operations to these processing files.

The lower part of the screen is a prompt frame. After user selecting a file in the list, its attributes will be shown in the prompt frame, such as current file folder, file name, size, and the first four lines of the file.

◆ **[ProcessWiz] function section**

Ncstudio offers three basic processing procedures, especially for the waterjet machine, including: [rectangle frame incision wizard], [round frame incision wizard] and [special line incision wizard]. User only needs to input some simple parameters to fulfill the manipulation. Since [ProcessWiz] function varies under different manipulation buttons, the picture will be shown in the following.

◆ **[History] function screen**

History function screen is as shown in Fig. 5-32. If user wants to load a recent fie, please turn to this screen, which records all the files loaded recently, and press [Load] button to load the selected file to begin processing.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
HD List (Q)	FD List (W)	ProcessWiz (E)	History (R)		

[History Record of Processing Files]

In this window, you can look over 8 processing files recently used, and can also execute further operation.

孔雀舞.dxf	Load
	Load
	Load
	Load
	Load
	Load
	Load
	Load

Fig. 5-32 File record history screen

◆ **Manipulation button bar**

The manipulation buttons in [PROG] function under auto mode are a little bit different, which will be introduced separately.

The picture of manipulation buttons in [HD List] function screen is as shown in Fig. 5-33.

F1 Load	F2 Edit	F3 Delete	F4 New	F5 Rename	F6	F7	F8

Fig. 5-33 Manipulation button in [HD List] screen of PROG function section

◆ **[Load]**

Hotkey: F1, only available in the [HD List] function screen of PROG function section. Select a file in the list, and then press the button, system will load the file into parsing engine automatically. At the same time, the loading progress bar will be displayed in the information list on the upper side of the interface. After the processing file is loaded, system is ready to start machining.

◆ **[Edit]**

Hotkey: F2, only available in the [HD List] function screen of PROG function section. Select a file in the list, and then press the button, the loading progress bar will be displayed in the information list on the upper side of the interface. After the file is loaded, system will automatically open NcEditor application for user to edit the file.

◆ **[Delete]**

Hotkey: F3, only available in the [HD List] function screen of PROG function section. Select a file in the list, and then press the button, a prompt dialog will pop up to remind user whether to delete the file selected. See Fig. 5-34.
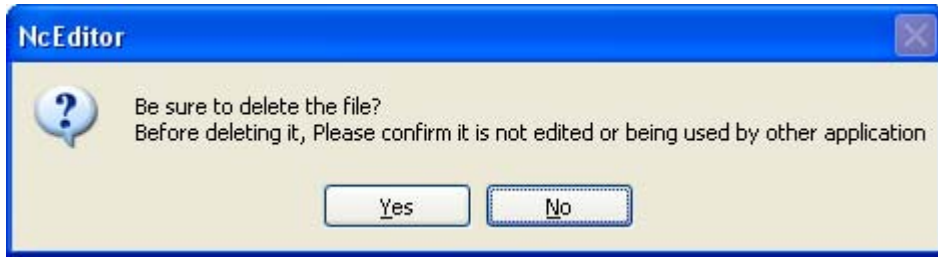
Fig. 5-34 Be sure to delete dialogue

Note:

When the processing file selected is in the state of loading, editing, or processing, system will forbid user to delete it.

◆ **[New]**

Hotkey: F4, only available in the [HD List] function screen of PROG function section. Press the button, and system will create a new file ".nc" (named: Untitle1.nc) under the path "D:\NCFILES". Select the file, Press [Rename] or [Edit] button, user can rename or edit it.

◆ **[Rename]**

Hotkey: F5, only available in the [HD List] function screen of PROG function section. Select a file in the list, and then press the button, system will eject a dialog to prompt user to input the file's name. Input the new name and click [OK] button to finish the operation. See Fig.5-35.
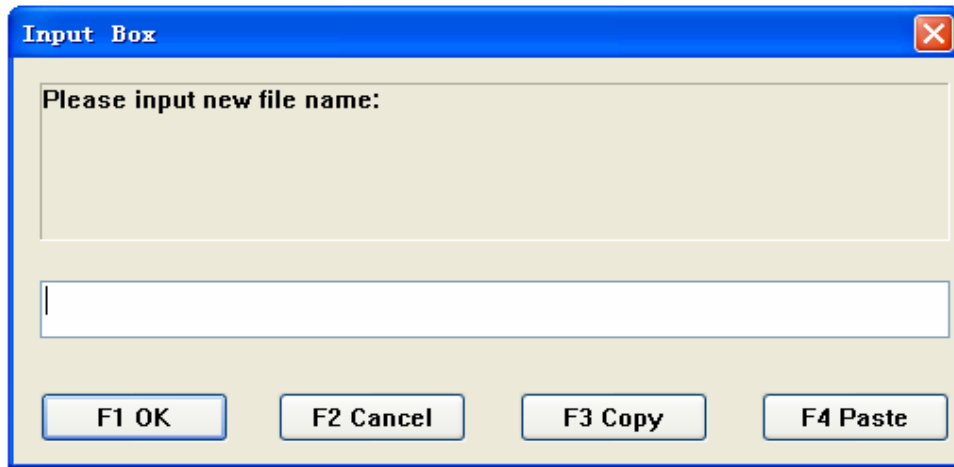
Fig. 5-35 Rename dialog in [HD List] of [PROG] function section

The picture of manipulation buttons in [FD List] function screen is as shown in Fig. 5-36.

Fig. 5-36 manipulation button in [FD List] function screen

◆ **[Load]**

Hotkey: F1, only available in the [FD List] function screen of PROG function section. Select a file in the list, and then press the button, system will load the file into parsing engine automatically. At the same time, the loading progress bar will be displayed in the information list on the upper side of the

interface. After the file is loaded, system is ready to start machining.

◆ **[Copy to HD]**

Hotkey: F2, only available in the [FD List] function screen of PROG function section. Select a file in the list, and then press the button, system will copy this file to the default path on the hard disk.

◆ **[Sel Drive]**

Hotkey: F3, only available in the [FD List] function screen of PROG function section. If there are several external memorizers, user can select one of them with this button.

◆ **[Safely Remove]**

Hotkey: F4, only available in the [FD List] function screen of PROG function section. After using the external memorizer, press the button, system will cut off the connection with external memorizer, and then the memorizer can be removed safely. However, if the current executed file or loaded file is in the external memorizer, the button is invalid.

The picture of manipulation buttons in [ProcessWiz] of [PROG] function section is as Fig 5-37.

Fig. 5-37 Manipulation button column in [ProcessWiz] of [PROG] section

◆ **[RecFrm] incision wizard**

Hotkey: F1, only available in the [ProcessWiz] function screen of PROG function section. Press the button, system will change to [Rectangle Frame Incision Wizard] function screen, as Fig. 5-38. In the middle of the screen, there is a machining demonstration. After selecting "Inner Frame" or "Outer Frame" and infilling the corresponding parameters, press the "Start" button on Operation Panel to start the rectangle frame machining.
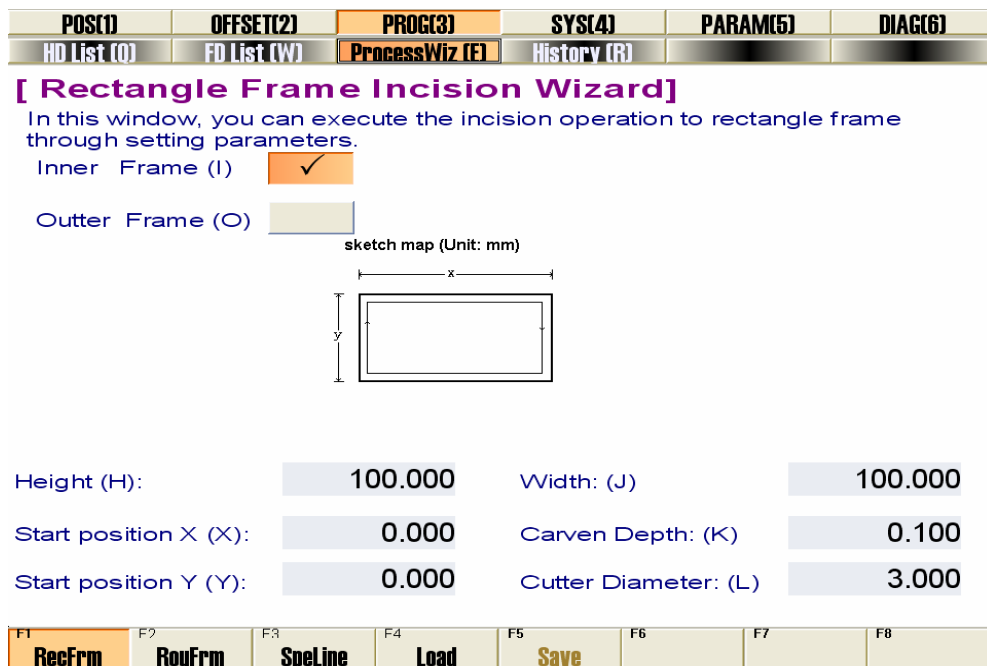
Fig. 5-38 Rectangle frame incision wizard

◆ **[RouFrm] incision wizard**

Hotkey: F2, only available in the [ProcessWiz] function screen of PROG function section. Press the button, system will change to [Round Frame Incision Wizard] function screen, as Fig 5-39. In the middle of the screen, there is a machining demonstration. After selecting "Inner Frame" or "Outer Frame" and infilling the corresponding parameters, press the "Start" button on Operation Panel to start the round frame machining.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
HD List (Q)	FD List (W)	ProcessWiz (E)	History (R)		

[Round Frame Incision Wizard]

In this window, you can execute the incision operation to round frame through setting parameters.

Inner Frame (I) ✓

Outer Frame (O)

sketch map (Unit: mm)

Carven Depth (J):	0.100		
Start position X (X):	0.000	Cutter Diameter (K):	3.000
Start position Y (Y):	0.000	Workpiece Diameter (L):	200.000

F1 RecFrm	F2 RouFrm	F3 SpeLine	F4 Load	F5 Save	F6	F7	F8

Fig. 5-39 Round frame incision wizard

◆ **[SpeLine]**

Hotkey: F3, only available in the [ProcessWiz] function screen of PROG function section. Press the button, and system will change to [Special Line] function Screen, as shown in Fig. 5-40. In the middle of the screen, there is a machining demonstration. After infilling the corresponding parameters, press the "Start" button on Operation Panel to start the special line machining.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
HD List (Q)	FD List (W)	ProcessWiz (E)	History (R)		

[Special Line]

In this window, you can execute the special line water-incision operation through setting parameters.

Back Point
②
End Point
Comp.Point ③
④ L
① c
start point

Rollback Distance (H):	20.000		
Compensation Distance (J):	0.000		
Start X (X):	92.131	End X (K):	100.000
Start Y (Y):	35.253	End Y (L):	100.000
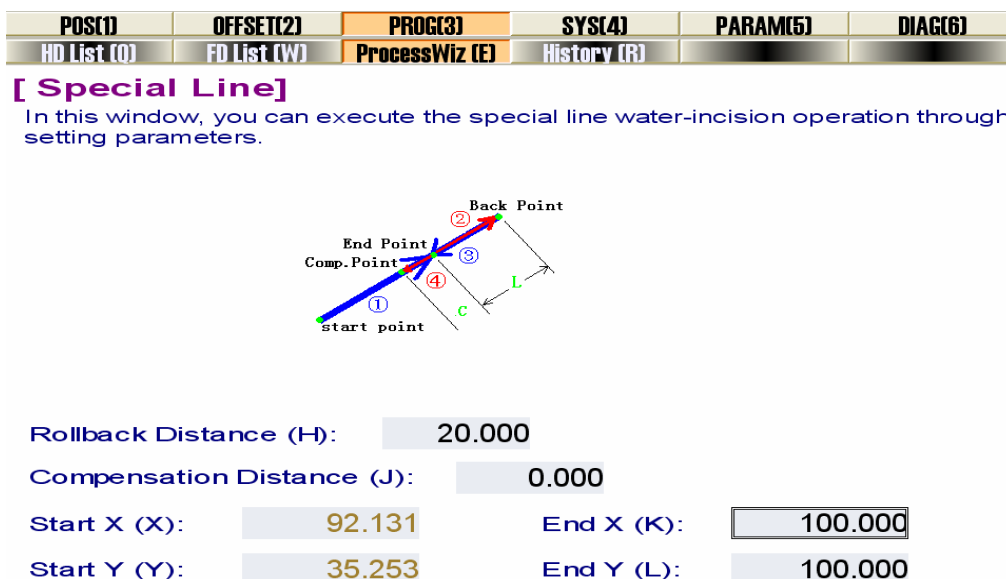
Fig. 5-40 Special line screen

◆ **[Load]**

Hotkey: F4, only available in the [ProcessWiz] function screen of PROG function section. Press the button to load the current process wizard file. Besides, User can choose a process wizard in advance and set the corresponding parameters, then press the button to load the processing file.

◆ **[Save]**

Hotkey: F5, only available in the [ProcessWiz] function screen of PROG function section. Press the button, and system will save the current process wizard and its parameters. If user usually uses a process wizard without changing the parameters, please press the button to save the wizard file, by which user can avoid repeated parameters setting after switching to [ProcessWiz] function screen again and start machining directly.

5.2.4 SYS function section

This function section is used for interrelated operations of configure selection, interface selection, language selection, and system information. [SYS] function section includes [Config], [Interface], [Lang], and [SysInfo] function screen.

◆ **[Config] function screen**

The screen is as shown in Fig. 5-41. It displays the configurations supported by Ncstudio. Press [Select] button to select the corresponding configuration according to actual situation.
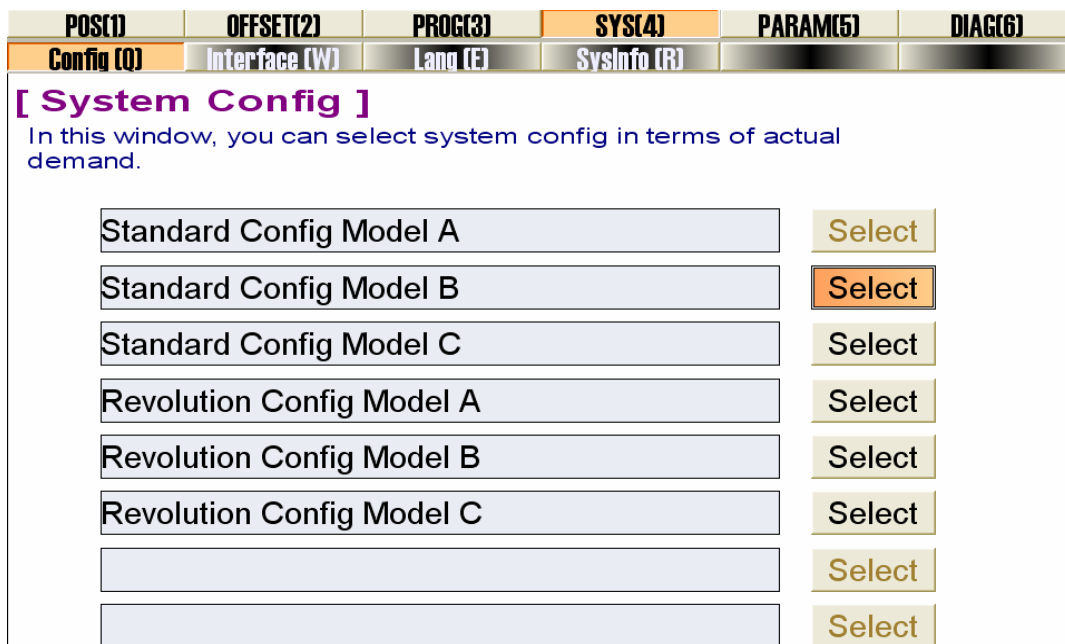
Fig. 5-41 Configuration function screen

➢ Standard Config Model A: it means that the output signals of all the pumps and valves are continuous.

➢ Standard Config Model B: it means that the output signal of oil pump is jog type while the output signals of other pumps and valves are continuous ones.

➢ Standard Config Model C: it means that oil pump and high pressure valve output jog signals

while other pumps and valves output continuous signals.

➢ Revolution Config Model A: it means that Y-axis is rotary axis and all of the pumps and valves are continuous signal outputs.

➢ Revolution Config Model B: it means that Y-axis is rotary axis and oil pump is jog signal output while other pumps and valves are continuous signal outputs.

➢ Revolution Config Model C: it means that Y-axis is rotary axis and oil pump and high pressure valve are jog signal outputs while other pumps and valves are continuous signal outputs.

◆ **[Interface] function screen**

As Fig. 5-42, user can select the theme of interface in this screen. Ncstudio supports two kinds of themes: Gray and Black. Press down [Select] button to choose one according to personal preference.

Fig. 5-42 Interface function screen

◆ **[Lang] function screen**

The screen is as shown in Fig. 5-43. User can select the theme of interface in this screen. Ncstudio supports two kinds of languages: Chinese and English. Press [Select] button to choose one kind of language according to actual situation.

Fig. 5-43 Language function screen

◆ **[SysInfo] function screen**

The screen is as shown in Fig. 5-44. User can look over the information about Ncstudio system. It includes version, company, hardware information, card no., remaining time and self-test information.

Fig. 5-44 SysInfo function screen

◆ **Manipulation button bar**

In [SYS] function section, only [SysInfo] function screen has picture of manipulation button, as Fig. 5-45.

Fig. 5-45 Manipulation button column under [SysInfo] function screen

◆ **[Register]**

Hotkey: F1, only available in the [SysInfo] function screen of [SYS] function section. Press the button, and system will eject a "REGISTER" dialog, as Fig.5-46. For new user, infill the register code and press [OK] to accomplish the registration.
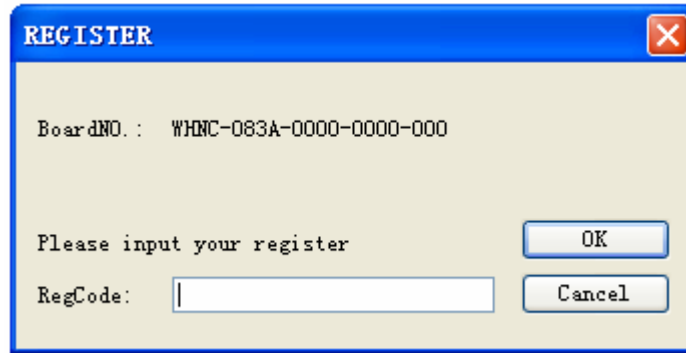
Fig. 5-46 Hardware register dialogue

5.2.5 PARAM function section

This section is used for parameters setting, parameters backup and parameters auto backup. [PARA] function section includes [Param], [WateJet], [Backup], and [AutoBackup] function screen.

◆ **[Param] function screen**

The screen is as shown in Fig. 5-47. There are three sub-screens in it. According to the grade of jurisdiction, user can access different sub-screens to set parameters.

Fig. 5-47 Manufacturers' parameter function screen

In V9 Ncstudio, user can look up a parameter in parameters list according to the type of parameters. If user is not clear about the type of the parameter, please refer to chapter 9, in which all parameters are introduced according to their serial number.

Parameters in V9 Ncstudio are numbered with 5 decimal digits. The first digit is main-class number (1~9). The second digit is sub-class number (0~9). Each sub-class has 1000 numbers (000~999). The type name of the parameters is offered according to the first and second digits.

The left buttons further classify the parameters according to their functions, which provides a very convenient way for user to look up the needed parameter.

◆ **[Operation] parameters**

In [Operation] parameters, it includes parameters which are related to the specific operations. Press the button to see the corresponding parameter list, including N7.1 (Manual), N7.2 (Auto), N7.4 (Reference Point) and N7.9 (other operations), etc.

◆ **[Axes] parameters**

[Axes] parameters include parameters which are related to the feed axes. Press the button, and you will see the corresponding parameter list, including N7.2 (Parameters that are related to axes under AUTO mode). Apart from N7.2, in manufacturers' parameters, it also includes N1.0 (Axis General), N1.2 (Compensations), and N1.3 (Velocity/ Acceleration Limits).

◆ **[Controller] parameters**

[Controller] parameters include parameters which are related to the connection and settings of external controllers. Press the button, and you will see the corresponding parameters list, including N5.2 (Handwheel), which is vacant in operators' parameters because only manufacturer and developer have the right to revise it.

◆ **[Program] parameters**

[Program] parameters include parameters which are related to machining program. Press the button, and you will see the corresponding parameter list, including N6.2 (G Code Select), N6.4 (Speed Control), and N6.5 (File Translation). Apart from these parameters, in manufacturers' parameters, it also includes N6.3 (Track Control).

◆ **[View] parameters**

[View] parameters include parameters which are related to interface appearance. Press the button, and you will see the corresponding parameters list, including N8.1 (Position View).

◆ **[Other] parameters**

In manufacturers' parameters, it includes N4200 (Red Light Prompt When Machining Finished), N75023 (Light Control Out Port Index), and N79400 (Auto Back to 0).

◆ **[All] parameters**

It lists all the parameters under current privilege. User can look over all the parameters' classification.

User can access different sub-screens to set parameters according to his privilege. Class and parameters available are shown as below. If user wants to look over and modify certain parameter, he needs to log-in the grade which has the right to look over and modify this parameter.

Table 1 Privilege and Parameters Available

Class	Parameters Available
Operator	Operators' parameters
Manufacturer	Operators' Parameters, Manufacturers' Parameters
Developer	Operators' Parameters, Manufacturers' Parameters, and Developers' Parameters

At the lower part of [Param] function screen is manipulation button column. There are three buttons including [Operator], [Manufacturer], and [Developer], with shortcut key F1, F2, and F3 respectively. Press a button, and you will access the corresponding parameters. For detailed information, please refer to chapter 9.

◆ **[WaterJet] function screen**

The screen is as shown in Fig. 5-48. These parameters are especially for waterjet machine, such as OPEN_HP_DELAYTIME, OPEN_CUTTING_DELAYTIME, and OPEN_ABRASIVE_DELAYTIME. User can reset them according to actual situation. For detailed information, please refer to chapter 9.

Fig. 5-48 Waterjet parameters list

◆ **[Backup] function screen**

The screen is as shown in Fig. 5-49. User can save and restore the parameters' setting in this screen, avoiding the trouble of setting parameters repeatedly. After modifying the parameter, please switch to this screen, press [Save] button, system will eject an "Input Box", input the name, and click [OK]. Ncstudio supplies 8 backup items. After the parameters have been saved, user can click [Load] button to load them momentarily in terms of the item name on the left.
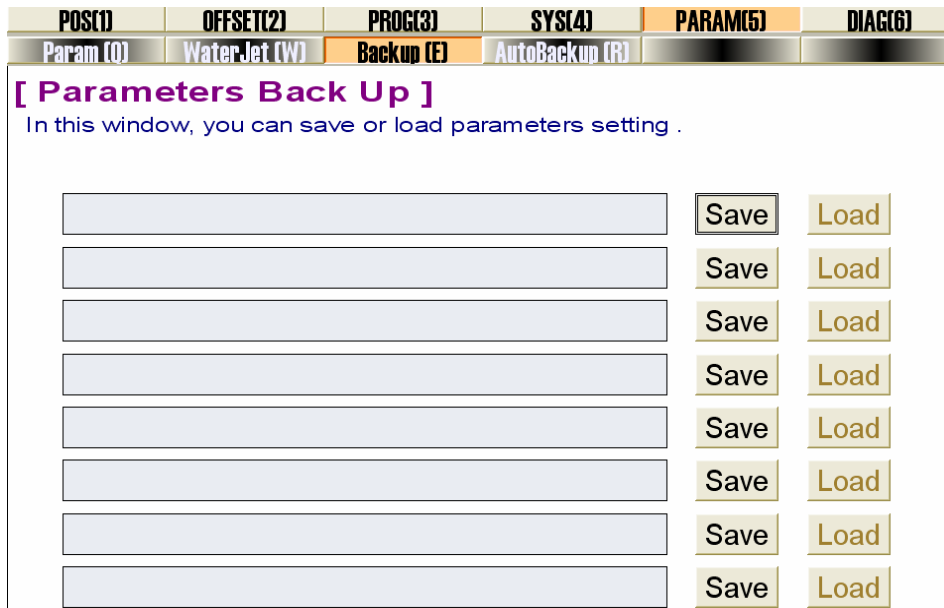
Fig. 5-49 Parameters backup screen

◆ **[Auto Backup] function screen**

The screen is as shown in Fig 5-50. Ncstudio supports function of parameters auto backup. If user forgets to save the parameters setting, please switch to this screen. Press [Load] button to restore a parameter setting of any time (from ex-factory date to the last time).
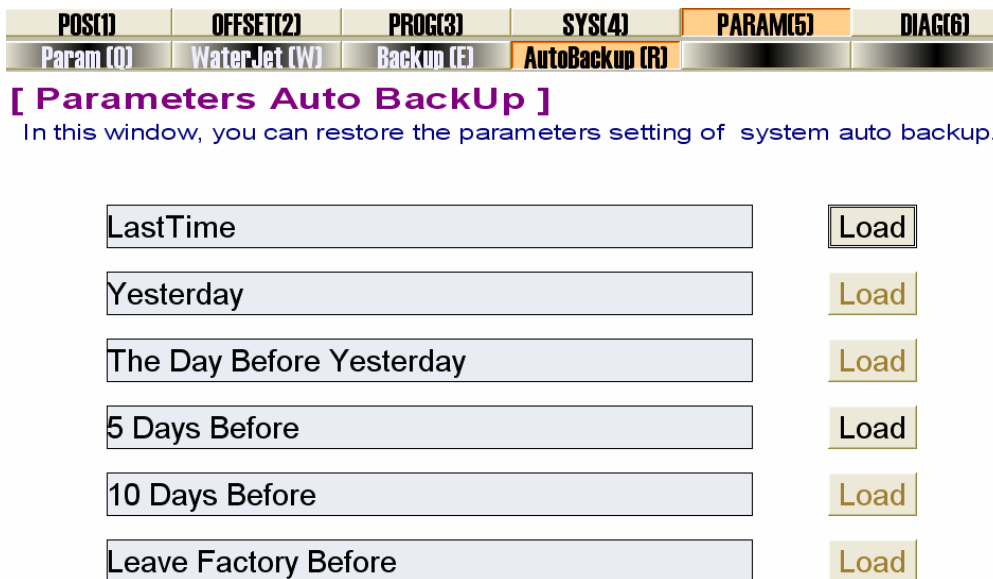
Fig. 5-50 Parameters auto backup screen

5.2.6 DIAG function section

The screens under [DIAG] function section are the same under each mode. This function section is mainly used for log record, hardware port test and polarity modification, and PLC test, etc.

[DIAG] function section includes [Log], [I/O Port], [PLC] and [OTHERS] function screen.

◆ **[Log] function screen**

The screen is as shown in Fig. 5-51. This screen records user's important operations and system events. User can browse not only current log information but also history log information.

Fig. 5-51 Log function screen

◆ **[I/O Port] function screen**

The screen is as shown in Fig. 5-52. It is used for displaying information about control card and system current input/ output map. It is very helpful for system supervision and fault diagnoses.

Fig. 5-52 I/O port function screen

◆ **[PLC] function screen**

The screen is as shown in Fig. 5-53. It is used for PLC debug and especially serves system debug persons and advanced users. For detailed information, please refer to V9 Programme Manual.

Fig. 5-53 PLC function screen

◆ **Manipulation button bar**

The manipulation button in [DIAG] function is a little bit different, which will be introduced separately.

The screen of manipulation button bar in [Log] function is shown as Fig.5-54.

Fig. 5-54 Manipulation button in [Log] function screen

◆ **[Refresh]**

Hotkey: F1, only available in the [Log] function screen of [DIAG] function section.

Pressing the button will refresh the log list and keep it synchronous with the system.

◆ **[Clean]**

Hotkey: F2, only available in the [Log] function screen of [DIAG] function section.

Pressing the button will clean all the current log information.

◆ **[Info], [Warning], [Error], [System], [Nowadays]**

Hotkey: F3, F4, F5, F6, F7, only available in the [Log] function screen of [DIAG] function section.

The default state of these buttons is checked, which means the corresponding information will be displayed, such as normal information, warning information, error information, and current information. If user needn't some type of information, click the related button to make it rise. When [Nowadays] button rises, system will display all the log records.

Manipulation button bar under [I/O Port] function is shown as Fig.5-55.

Fig. 5-55 Manipulation button column in [I/O Port] Screen

◆ **[Test On/ Test Off]**

Hotkey in turn: F1, F2, only available in [I/O Port] function screen. They are mainly used for simulating signal in simulation test. Press one of the two buttons, the indicator in front of the selected port will transform between green and red. Green light indicates port test is on; red light means port test is off.

> Note:
>
> Indicator lamps are different under simulation mode and actual situation. See information below:
>
> Under simulation mode: Green lamp🔲, red lamp🔲.
>
> Under actual mode: green lamp🟢, red lamp🔴.

◆ **[CanITst]**

Hotkey: F3, only available in [I/O Port] function screen. Pressing the button will cancel simulation signal and simulation test, and replace simulation signals with actual hardware signals.

◆ **[CanIAll]**

Hotkey: F4, only available in [I/O Port] function screen. [CanIAll] is similar to [CanITst] button, but [CanITst] button can only cancel one single port's simulation while [CanIAll] button can abolish all ports' simulation and replace them with actual hardware signals.

◆ **[ConvtPol]**

Hotkey: F5, only available in [I/O Port] function screen. It is used to modify the polarity "N" (Negative) or "P" (Positive) of port.

◆ **[DispAll]**

Hotkey: F6, only available in [I/O Port] function screen. Ports not used are hidden in this screen. Press down the button to see all the I/O ports; click the button again to let the button rise, and ports not used will be hidden again.

◆ **[PortAttr]**

Hotkey: F7, only available in [I/O Port] function screen. It is used to revise the attribute of the selected port. Press the button, and system will eject a dialog, as shown below. User can set sampling interval, decide whether to filter and enable, and revise port's name and description.
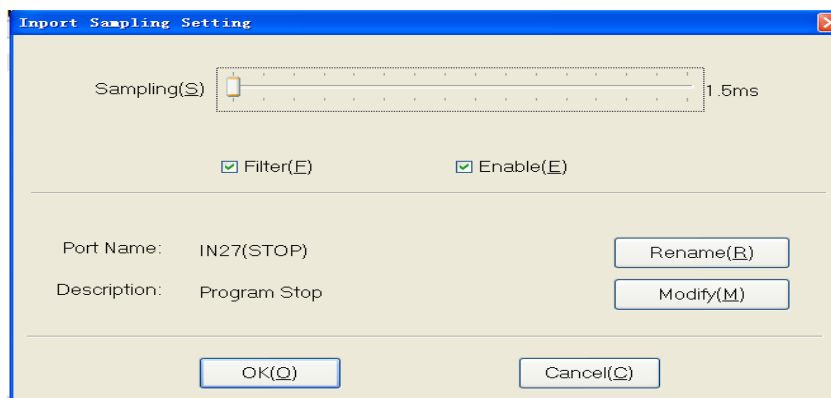
Fig. 5-56 Port attribute setting dialogue

6 Manual Mode of Ncstudio™

Under manual mode, user can control the machine by manual operation facility, such as manual pulse generator, manual operation panel. Manual mode is subdivided into: series, stepping (also-called increment), handwheel and so on.

Manual mode is shown in the Fig.6-1. Similar to auto mode, it includes two panels: operation panel and function panel. These two panels will be introduced separately in detail as below.
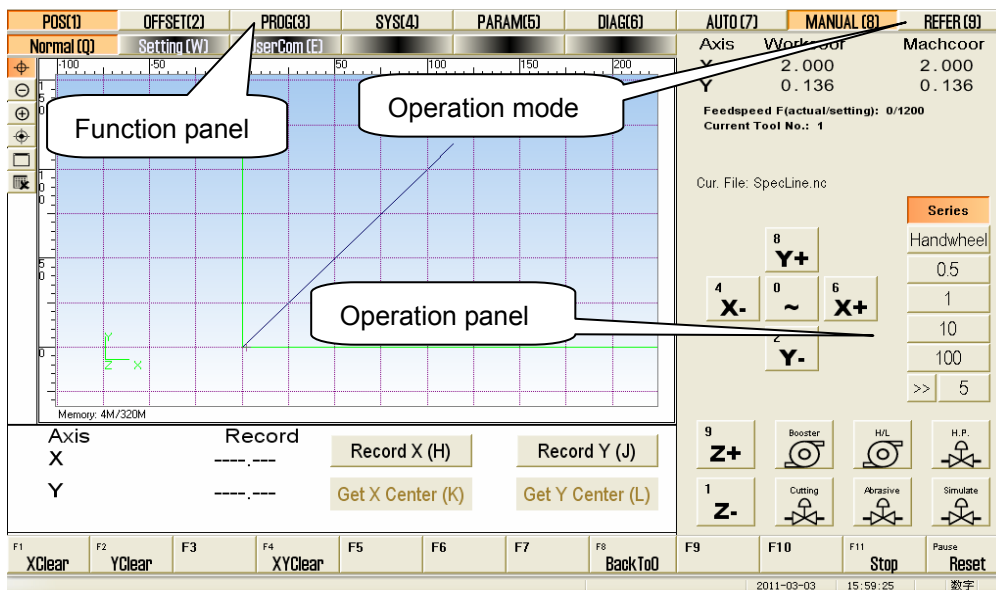

Fig. 6-1 Manual operation mode screen

6.1   Operation panel section

As Fig.6-2, this panel provides an alternant operating environment for manipulating machine tool manually, consisting of many manual operation command buttons:
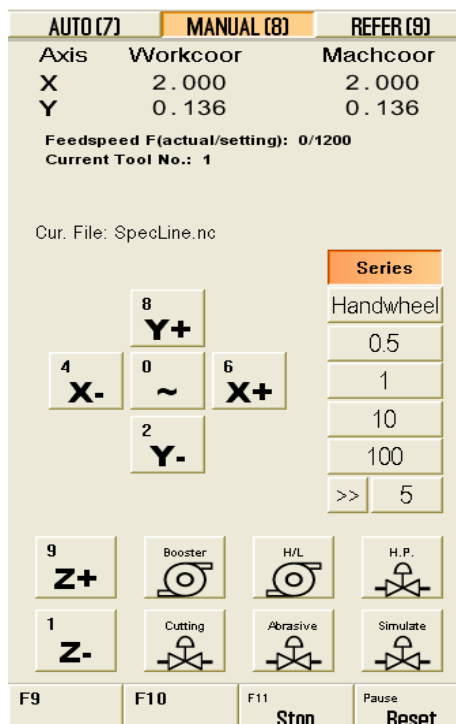
Fig. 6-2 Manual operation panel

The upper part of the panel is a coordinate display section, which shows the current coordinate of

the water jet, current feed speed, current tool No. and the name of the current file.

Under the mini coordinate display area is the panel button area, inside which buttons are different under different modes. Under the manual mode, the panel button section includes these buttons:

◆ **[X+], [X-] manual button, shown as:**

The buttons respectively correspond to moving direction of - X and + X axis. Hotkey: number key 4 and 6 in mini-keyboard.

◆ **[Y+], [Y-] manual button, shown as:**

The buttons respectively correspond to moving direction of - Y and + Y axis. Hotkey: number key 2 and 8 in mini-keyboard.

◆ **[~] manual button, shown as:**

Acceleration button; only available under "Series" state; it corresponds to number key "0" in the mini-keyboard; it is used together with X, Y direction keys to accelerate the moving speed of water jet; its combination key is "0" plus direction number key , such as "0"+"8" (accelerate along +Y direction).

◆ **[Series], [Handwheel], [0.5], [1], [10], [100], [>>] manual buttons, see Fig.6-3.**
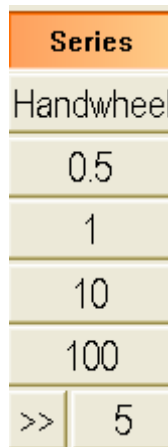
Fig. 6-3 Manual moving buttons

There are totally three manners to manipulate the machine under manual mode. They are series manner, handwheel manner (MPG), and stepping manner.

Series manner: pressing down the button, user can manually make the machine tool move continuously. Under the MANUAL mode (or the function screen is current active window), clicking the direction key in the panel, or pressing down the corresponding shortcut key will make the machine tool move continuously. Loosening it will stop the moving.

Note:
Manual screen must be active, which can be distinguished by the color of the title bar.

> When manual screen is active, number lock will not be taken into consideration, number key is active all the time.

Handwheel (MPG): to support handwheel (MPG—Manual Pulse Generator), we add the button [Handwheel]. Firstly, after installing a handwheel, press down the button, and then user can manipulate the machine tool by handwheel. At this time, the X, Y direction keys are invalid.

Stepping Manner (0.5, 1, 10, 100, >>): It is also called step-manner or increment-manner. This manner can control the machine's feed distance more accurately, which is different from jog-manner. Every time you triggering the X, Y direction key, the machine will move a specified step-size. Press down one button of [0.5], [1], [10], [100], the increment stepping mode will be activated. And increment stepping length equals to selected step-length. For instance, if [0.5] button is selected, increment stepping length will be 0.5 mm. Moreover, V9 NcStudio also supports custom step-size. Press down [>>] button, system will eject an "Input Box" dialog, input a step-size that you want, click [OK], value of the custom step-size will be displayed behind [>>] button. For example, press the [>>] button and input "500.0" into the "Input Box", and then click [OK], "500"will be displayed behind [>>] button. When [500] button is pressed, every time triggering the direction key X and Y will make the machine tool move 500 mm accordingly.

◆ **[Stop], [Reset]**

These two functions have already been introduced in AUTO mode, which will be omitted here.

6.2　Function panel section

Note: as shown in the picture, the function button **POS(1)** of manual mode is different from that in the auto mode, while the others are the same as the buttons in auto mode.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Normal (Q)	Setting (W)	UserCom (E)			

Under MANUAL mode, **POS(1)** subfunction section includes: Normal, Setting, and UserCom.

◆ **Normal function screen**

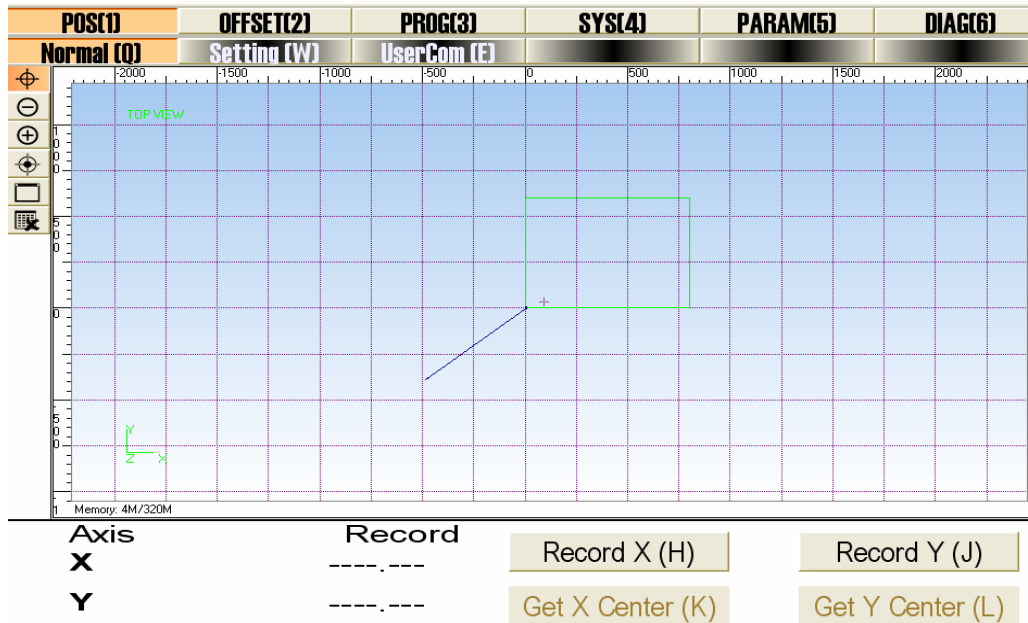Normal function screen under MANUAL mode is as following, see Fig.6-4.

Fig. 6-4 Normal function screen under manual mode

◆ **[Record X]**

Press the button ⬚ Record X (H) , system will record X-coordinate of the current position.

◆ **[Record Y]**

Press the button ⬚ Record Y (J) , system will record Y-coordinate of the current position.

Buttons Get X Center (K) and Get Y Center (L) are corresponding to the completion of X centering and Y centering.

◆ **Setting function screen**

Setting function screen under MANUAL mode is as Fig 6-5. In this screen, user can set the parameters in common use under MANUAL mode, such as manual low speed, manual high speed, etc. Manual low speed refers to the moving speed when user doesn't press down the manual high-speed key, with range: 1 mm/min ~ manual high speed. Manual high speed refers to the moving speed when user presses the manual high-speed key, with range: manual low speed ~ dry running speed.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Normal (Q)	Setting (W)	UserCom (E)			

[Parameters Setting in Common Use]

In this window, you can set the parameters of manu mode in common use.
Other correlative parameters can be set in the Param Window.

1 Manual Low Speed (J)   1200 mm/min

2 Manual High Speed (K)   3000 mm/min
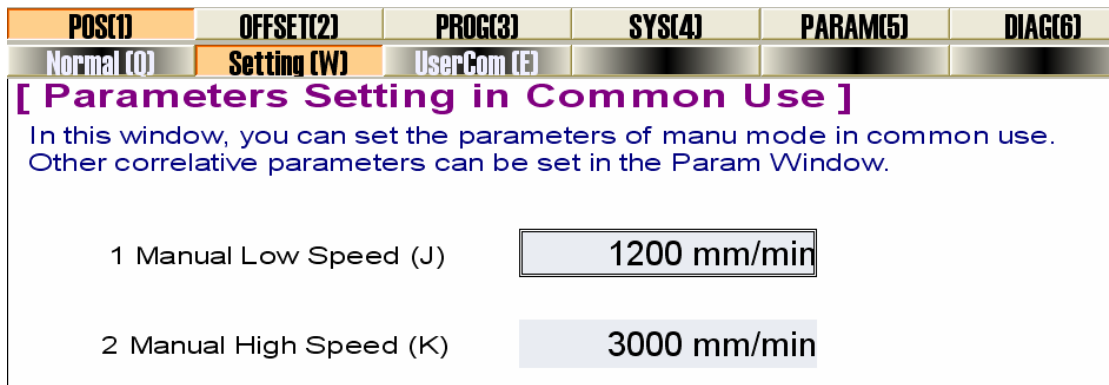
Fig. 6-5 Setting function screen of POS function section under MANUAL mode

◆ **[UserCom] function screen**

UserCom function screen is as shown in Fig. 6-6. In this screen, it supplies ten instruction input items, and every item supports instruction input repeatedly. If no instruction is input into the item, the [Execute] button is disabled. Click [Edit] button, system will eject a dialog "Input Box" as below. Input instruction into the dialog, click [Ok] button, [Execute] button will be available. Click [Execute] button, system will execute this instruction. Moreover, the instructions input will be remembered in the screen, which provides user with a very convenient way to look over the instructions.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Normal (Q)	Setting (W)	UserCom (E)			

[User Command Record]

In this window, you can look over 10 commands recently entered, and also can execute further operation.

	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
	Edit	Execute
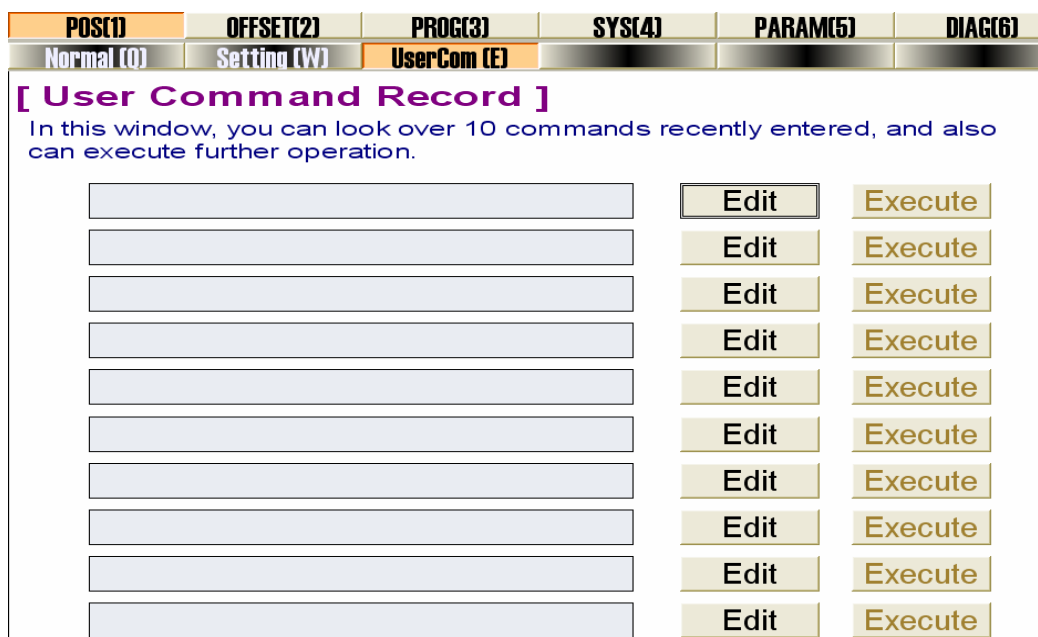
Fig. 6-6 User command function screen

◆ **Manipulation button bar**

Under MANUAL mode, the manipulation buttons are different in different screens of [POS].

Picture of manipulation button column in the Normal function screen of POS function section is as shown in Fig. 6-7.

F1	F2	F3	F4	F5	F6	F7	F8
XClear	YClear		XYClear				BackTo0

Fig. 6-7 Manipulation button column of setting screen

◆ **[Clear] option**

System will eject a dialogue as following in Fig. 6-8, when [clear] button is selected:
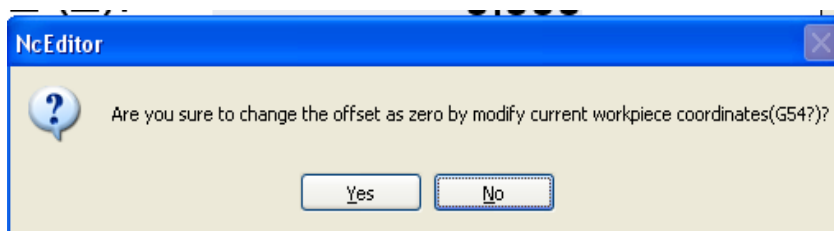
Fig. 6-8 Clear dialogue

Press "Yes": set current point as workpiece coordinate origin.

Press "No": no action will be executed.

◆ **[Back To 0]**

Press the button **BackTo0**, system will be back to the predetermined workpiece origin.

There is only a [Back to 0] button under [setting] screen; the principle is almost the same as the above, which will not be discussed in detail.

7 Reference Point Mode of Ncstudio™

[Back to the Reference Point] mode is the process to unify the internal coordinates with the machine actual coordinates. Therefore, during activation process, it is essential to return to the reference point. After the system is started each time, its default operation mode is the reference point mode, which is convenient for user to perform the operation of "back to origin".

See Fig.7-1:
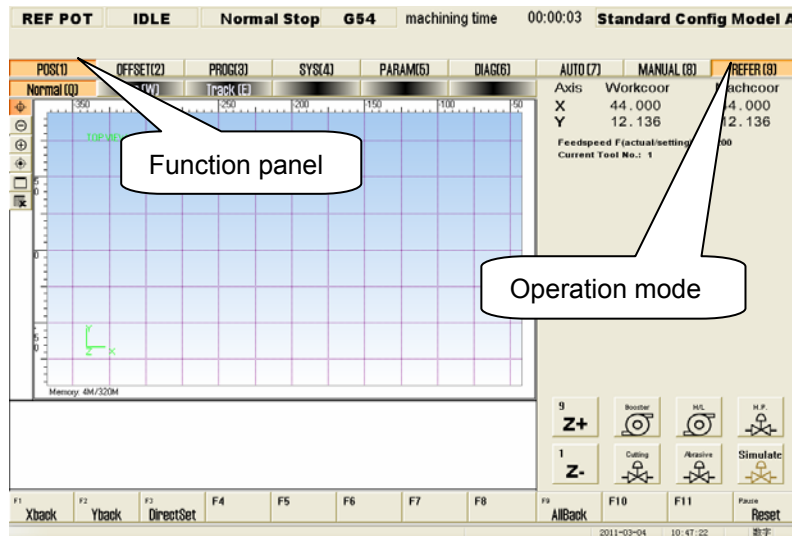
Fig. 7-1   Reference operation panel dialog

The operation panel of reference point displays the operation buttons related with mechanical point, supporting an alternant interface for manipulating machine tool to the origin of mechanical coordinate system. Its function selection buttons and subfunction switch buttons are the same as those in manual and auto operation panels.

7.1   Operation panel section

The operation panel section is shown as following, see Fig.7-2.

Fig. 7-2 Operation panel dialogue

The operation panel of reference point displays the operation buttons related with mechanical point, supporting an alternant interface for manipulating machine tool to the origin of mechanical coordinates system.

◆   **[All Back]**

Shown as: **F9 AllBack**

Machine origin (the reference point), origin of MCS, is a fixed position on the machine, and is positioned by both mechanical switch and electrical system. With regard to the machine coordinate system, please refer to chapter 3.2. Operation of backing to the reference point requires the support of machine hardware, and some type of machine doesn't have this function. Reference point is the benchmark of whole machine, so it is used to calibrate the coordinate of the current point.

To avoid inaccurate coordinate caused by power-off, E-stop and etc, please execute operation of backing to the reference point again after E-stop or program restart.

[AllBack] button is used to call both X-axis and Y-axis (used for two axes) to back to the reference point. V9 Ncstudio also supports single-axis backing to the reference point. For detailed information, please refer to chapter 7.2.

◆ **[Reset]**

This button has already been introduced in AUTO mode, which will be omitted here.

7.2　Function panel section

Note: as shown in the picture, the function button **POS(1)** of REFER mode is different from that in the AUTO mode and MANUAL mode, while the others are the same.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Normal (Q)	Pos (W)	Track (E)			

Under REFER mode, **POS(1)** subfunction section includes: Normal, Pos, and Track.

◆ **Normal function screen**

The normal function screen is similar to the one in auto and manual mode, only manipulation button bar in the lower part is different. Therefore, this screen will not be introduced in detail. The picture is as following:
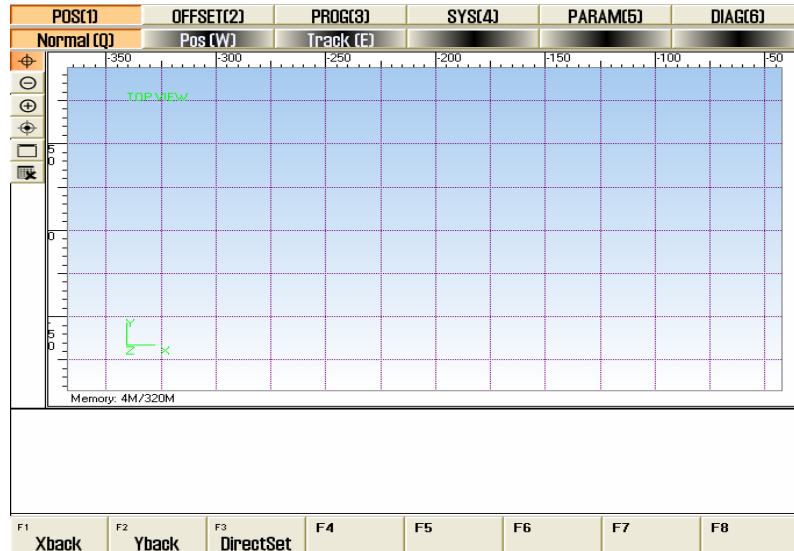
Fig. 7-3 Normal function screen under REFER mode

◆ **Position function section**

The POS function under REFER mode is the same as the one under auto mode, only manipulation button bar in the lower part is different. This is convenient for user to look for the position of water jet under these two modes, including mechanical coordinate, workpiece coordinate, current workpiece coordinate, and remaining distance. The picture will be omitted.

◆ **[Track] function screen**

The upper part of track function screen is the same as that under auto mode, only manipulation button bar in the lower part is different. The picture will be omitted.

◆ **Manipulation button bar**

The manipulation buttons in different function screen under this mode are all the same, as following:

Fig. 7-4 Manipulation button bar under REFER mode

◆ **[X Back]**

Hotkey: F1, it is only available under REFER mode. Press the button to control the water-jet to go back to the machine origin along X-axis. When the operation is finished, in the mini-coordinate display zone of operation panel and in [Pos] function screen, there will be a mark (⊕) of completing backing to the machine origin along X-axis.

◆ **[Y Back]**

Hotkey: F2, it is only available under REFER mode. Press the button, and system will control the water jet to go back to the machine origin along Y-axis. When the operation is finished, in the mini-coordinate display zone of operation panel and in [Pos] function screen, there will be a mark (⊕)

of completing backing to the machine origin along Y-axis.

◆ **[DirectSet]**

Hotkey: F3, it is only available under REFER mode. If user can make sure the current position is consistent with its machine coordinate and there is no abnormal state, such as "E-stop", he can directly click the button to set the current machine coordinate as the accurate machine coordinate. When the operation is finished, in the mini-coordinate display zone of operation panel and in the [Pos] function screen, there will be a mark (⊕) of completing the operation of DirectSet before X and Y axis.

8   Menu System of Ncstudio™

    Menu operation, keyboard operation, and shortcut key operation constitute the entire operation framework of Ncstudio$^{TM.}$ Menu operation is one of the most important methods to manipulate Ncstudio^{TM} and provides user with very rich menu commands, which will be introduced in detail in this chapter.

8.1   File menu

[File] menu mainly includes operations about machining file, as shown in Fig. 8-1.
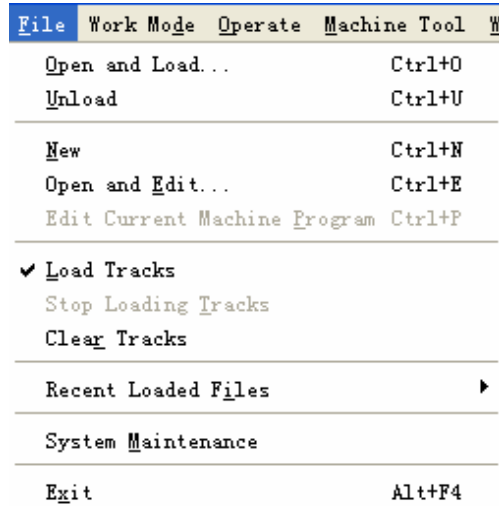
Fig. 8-1 File menu

◆   **[Open and Load]**

Shortcut key: Ctrl + O. The menu is used for opening the processing file from hard disk and loading it into the decoder of the system. Then the parsing engine will parse the file into G instructions, and transfer these instructions to CNC card.

Choose the menu item; system will eject the "Open and Load" dialog box, shown in Fig. 8-2. After selecting the machining program, press [open] button to load it automatically. And the operation panel will display the loaded processing file name.
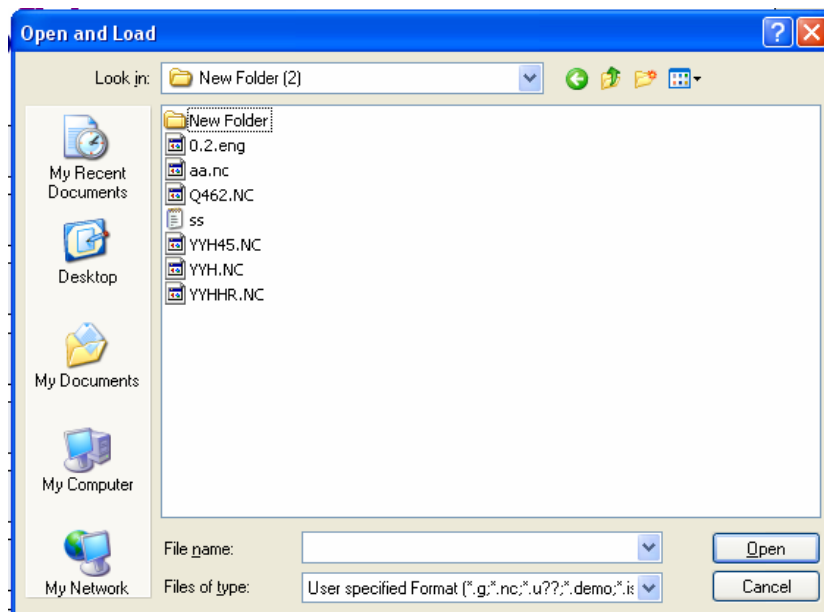
Fig. 8-2 Open and load dialog

Once a processing file is loaded into system, it can not be modified (but it can be "read"). If user wants to modify the processing file loaded, he has to unload the file first.

Note: when the menu item with symbol "…" behind is chosen, the system will eject a dialog.

◆ **[Unload]**

This menu item has the same function as button [Unload] in [POS] function section under AUTO mode. After loading a machining file into the system, choose [Unload] item to close the current file loaded in parsing.

Note: if the loaded file is processing, [Unload] function is invalid.

◆ **[New]**

Choose this menu item, and the system will create a new file ready for editing. User can edit it in NcEditor. What's more, user can save it at any time.

◆ **[Open and Edit]**

This menu item is used for opening processing files saved in disk and loading them into NcEditor, in which user can edit a processing file. Clicking the menu option will eject an [open and edit] dialogue box. User can operate it according to the prompt in the NcEditor, please refer to users' manual of NcEditor for details.

◆ **[Edit Current Machining Program]**

This function can load the procedure file that has already been in digital control procedure explanation buffer into the editor window. This menu item is valid only when there are procedure files in digital control procedure explanation buffer.

It is advised not to edit the files with saved origin of coordinate.

◆ **[Load Tracks]**

This menu item is used to parse the machining file and load the entire track once and for all. In actual machining process, [Pos] function screen will also display the simulated processing trace, but it is stepwise in accordance with machining process. Track and trace can be distinguished by color; select [Load Tracks] will load the track in advance, and its opposite option is [Clear Tracks].

◆ **[Stop Loading Tracks]**

The menu item is used to terminate loading track. If the track of the loaded file is too large, it will slow down the system, so user can choose [Stop Loading Tracks] to terminate the track loading or choose [Clear Tracks] to stop track loading and clear the loaded track.

◆ **[Clear Tracks]**

The menu item is used for clearing the loaded track and forbidden continuing loading track. Clicking this option will clear out the selection state of [load track] menu option.

◆ **[Recent loaded processing file]**

The menu item will open a new submenu; the submenu will display the recent loaded processing files. If user need load these files, he can click the filename directly, which is very convenient to load these processing files.

◆ **[System Maintenance]**

The menu item is mainly used for backups, key data recovery, system installation and installation package making, etc. Selecting this menu item will close Ncstudio temporarily. As Fig. 8-3 below, system will eject a "System maintain" dialog box. The dialog includes: Backup key data, key data resume, system installation, and generate installation packet.
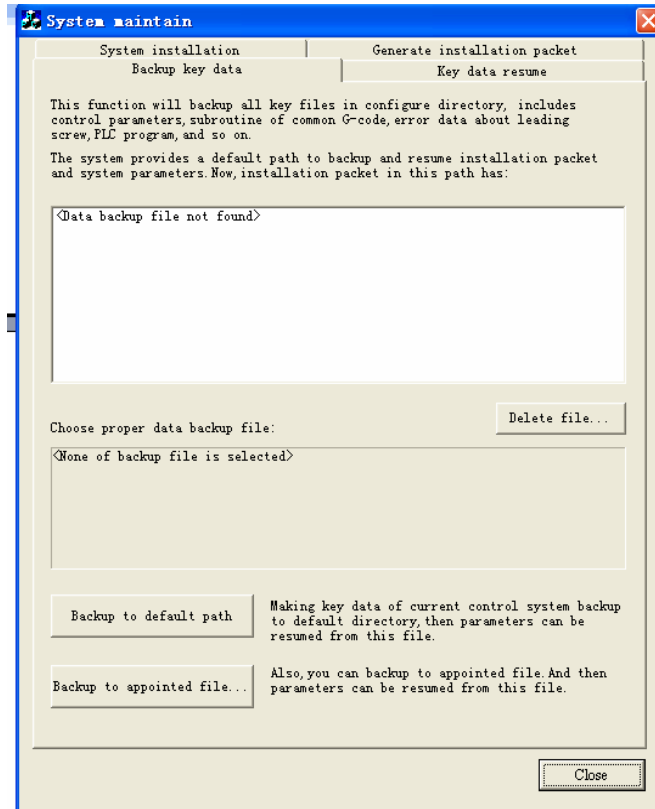
Fig. 8-3 System maintain dialog

➤ Under the "Backup key data" option card, user can back up all the key files in configuration directory, including controlling parameters, sub program of common-G code, error data of lead screw, PLC program, and so on. If there are backups under the default directory, they will be displayed in the middle window. User can delete or preserve them, and check their properties on lower window by clicking them. User can select whether to back up the files under the default directory. If not, user can appoint a directory for files backup.

➤ Under the "Key data resume" option card, user can recover the key files according to backups. If there are backups under the default directory, they will be displayed in the middle window. Clicking a backup in the window, its attribute will be displayed below the window. User can execute "delete file" and "start resume" function. Press direction-key to select other backup files or press [Browse] button to select files under other appointed directory. Then click [Start Resume] button, system will resume it automatically.

> "System installation" option card: user can install NcStudio system once again. However, the installation will delete all former files under the installation directory and data of the previous system will also be deleted. So we advise user to backup the data (parameters) before installation. If there are already any installation packages under the default backup directory (the same path as that of package generation directory), they will be displayed in the middle window. Click one package, and its attribute will be displayed below the window. User can delete or install it. Press direction-key to select other installation package. Click button [Start installation], and system will start installation automatically.

> "Generate installation packet" option card: user can create a new installation package based on the current system. It is very helpful to backup and save a stable system version. If there are already installation packages under the default directory (the same path as that of system installation package directory), they will be displayed in middle window. Click one package, and its attribute will be displayed below the middle window. User can delete it according to actual situation. Press direction-key to select other installation packages. Click button [Generate to default path] or [Generate to appointed path], a "Generate install packet" dialog will pop up, then click button [Start Creating] to make a package.

◆ **[Close]**

The menu item is used to close Ncstudio^{TM} system. If there is any file being edited, system will prompt user to save it. User can complete the operation according to the prompt.

8.2 Work mode menu

[Work Mode] menu mainly includes items about machining mode, as shown in Fig. 8-4
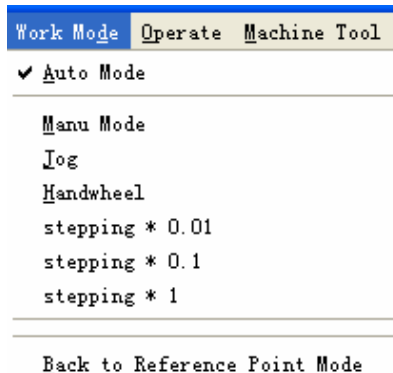
Fig. 8-4 Work mode menu

◆ **[Auto Mode]**

It corresponds to button [AUTO] on the operation panel. Selecting the menu item will switch to AUTO mode.

◆ **[Manual Mode]**

It corresponds to button [Manual] on the operation panel. Selecting the menu item will switch to Manual mode.

◆ **[Jog]**

It corresponds to button [Series] on the operation panel. Selecting the menu item will appoint jog as current manual manner.

◆ **[Handwheel]**

It corresponds to button [Handwheel] on the operation panel. Selecting the menu item will appoint handwheel as current manual manner.

◆ **[Stepping * 0.01], [Stepping * 0.1], [Stepping * 1]**

It corresponds to step-size buttons on the operation panel. Selecting a menu item will appoint jog as current manual manner and use the selected step-size as current step-size.

◆ **[Back to Reference Point Mode]**

It corresponds to button [REFER] on the operation panel. Selecting the menu item will switch to reference point mode.

8.3 Operate Menu

[Operate] menu mainly includes items related to manipulation, as shown in Fig. 8-5.

Fig. 8-5 Operate menu

◆ **[Single Block]**

The menu item is only available under AUTO mode.

User can set the program selected to be executed as [Single Block], which is a good way to support error diagnoses and failure recovery. Under the [Single Block] mode, processing stops when the resultant velocity of each axis is zero.

◆ **[Handwheel Guide]**

If user needs handwheel to control the movement of the water jet, install the handwheel first, then select this menu item, and system will change into handwheel mode.

◆ **[Auto Back]**

The menu item is a new function of V9 NcStudio and only available under AUTO mode. When user needs to return to a foregone position, select this menu item, and system will go back along the track.

◆ **[Set Workpiece Origin]**

This menu item is used to set current position as workpiece origin. It will not cause any movement to actual position. User can also set workpiece origin in [WorkCoor] screen of [OFFSET] function section.

◆ **[Save Workpiece Origin]**

This menu item is used to save current workpiece origin into system machining file. Regarding frequently used workpiece origins, user can save them for future use. There are totally 10 groups of workpiece origin to be saved. After setting a workpiece origin for the first time, user quickly locates to this workpiece origin again by using this function to save the current coordinate as workpiece origin.

◆ **[Load Workpiece Origin]**

This menu item is used to load workpiece origin that has already been saved in advance. After loading a workpiece origin, please use [Back to Workpiece Origin] to go back the workpiece origin.

◆ **[Start], [Pause], [Stop]**

They correspond to button [Start], [Pause], [stop] on operation panel in turn. These menu items are only valid under AUTO mode. They can also be applied to simulation mode.

◆ **[Simulation Mode]**

It corresponds to button [Simmu] of [POS] function screen under [AUTO] mode. If the machining procedure has been loaded in, and the current operation state is "IDLE", choose the menu item, system will carry out simulation automatically with high speed from the first section.

Simulation function provides user with a fast and vivid simulating machining environment.

To run a machining file under simulation mode, system will not drive the machine tool to do the relative actions, but only show the water jet trace with high speed in the process trace window. By simulation, user can know about water jet's movement in advance, avoiding damage (to the machine tool) which is caused by mistake in writing procedure, and get other additional information.

Once simulation starts, this menu item will change into "Stop Simulation and Withdraw the Simulation Mode". Selecting this item will exit from simulation.

◆ **[Select Processing Block]**

The menu item corresponds to button [SelWork] of [POS] function screen under [AUTO] mode, whose shortcut key is "Ctrl+F9". It is used for block skip execution and only available under AUTO

mode. It is also applied to simulation mode. After user choosing the menu item, system will eject an "Execute (Advance Options)" dialog, as Fig. 8-6 below. Use can operate according to prompt.
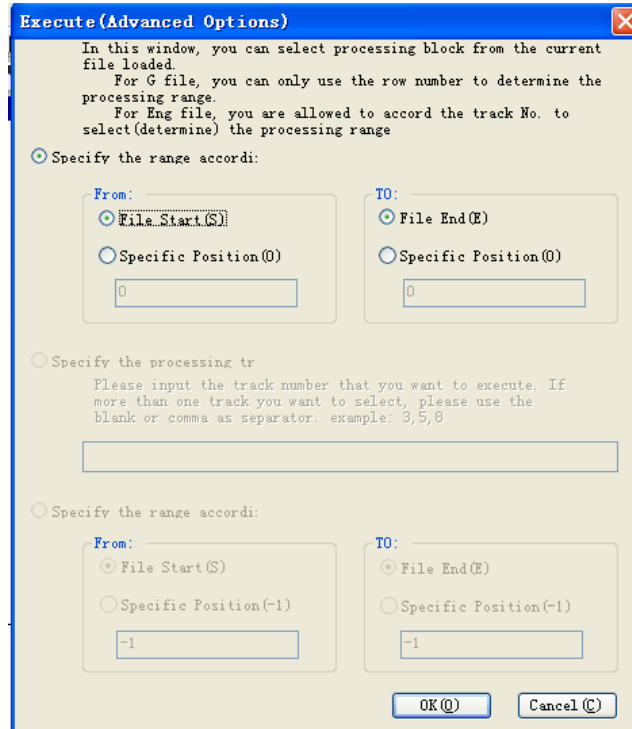
Fig. 8-6 Execute (Advanced Options) dialogue

◆ **[Breakpoint Resume]**

The menu item corresponds to button [Continu] of [POS] function screen under [AUTO] mode, and only available under AUTO mode. Its shortcut key is Shift + F9. After user selecting this menu item, system will resume machining from the point that is broken off last time. It can also be applied to simulation mode.

◆ **[Back to Workpiece Origin]**

Workpiece origin is the zero point of workpiece coordinate system. Generally speaking, it is a start-point of workpiece machining. Programmer can set workpiece origin freely. The position relation between workpiece origin and reference point can be set by [Set Workpiece Origin].

After user selecting the menu item, nozzle will go back to the workpiece origin automatically from current position.

[Back to Workpiece Origin] has the same function with button [Back to 0] of [POS] function screen under AUTO and MANUAL mode.

◆ **[Back to Fixed Point]**

User can set a fixed point in advance. When the water jet is damaged or needed to change, selecting this menu item will uplift the nozzle and back the fixed point for user to change water jet.

◆ **[All Back to Reference Point]**

This menu item is used to call all axes to go back to reference point. We suggest that user should

use this function before machining starts or after E-stop occurs.

◆ **[Reset]**

The menu item corresponds to button [Reset] on operation panel, with shortcut key F12. After user selecting this menu item, the machine will stop current machining task (If it is machining), and restore from warning state (If warning occurs) to IDLE state.

Using [Reset] button to stop is different from using [Stop] button. [Reset] button terminates machining with E-stop without deceleration and water jet uplift, and the state of water valve and high pressure will not be changed.

◆ **[Array Process]**

This function is machining a program with array process. After user selecting this item, an "Array process" dialog will pop up, as Fig. 8-7.
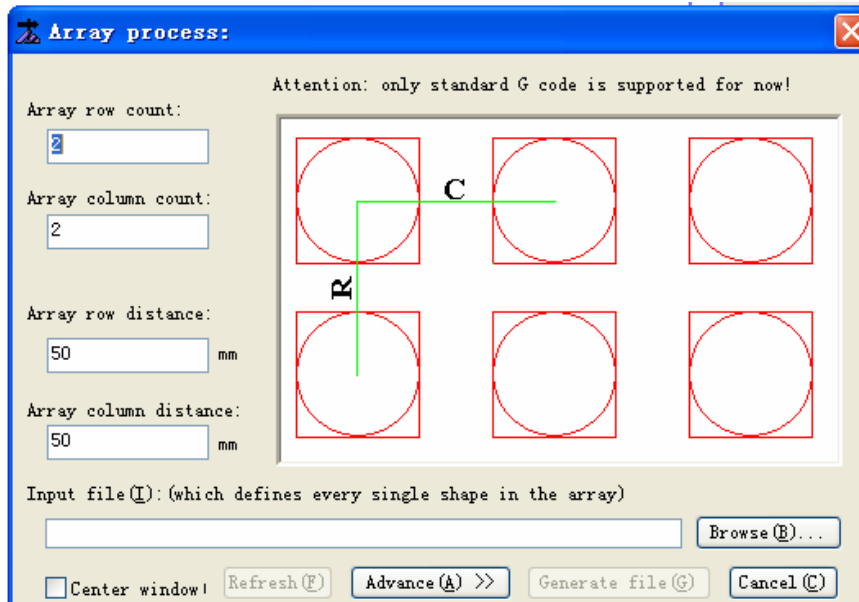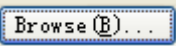
Fig. 8-7 Array process

Click button [Browse(B)...], and select the program which needs array process.

After correctly setting "Array row count", "Array column count", "Array row distance" (the space between two workpiece origins, like "R" above), and "Array column distance" (the space between two workpiece origins, like "C" above), user needs to click "Generate file" to finish the array process. After the array process is finished, the new generated processing program will be loaded into Ncstudio automatically.

If user needs to set different array row distances or column distances, click button "Advance", and a dialog as Fig. 8-8 will pop up.
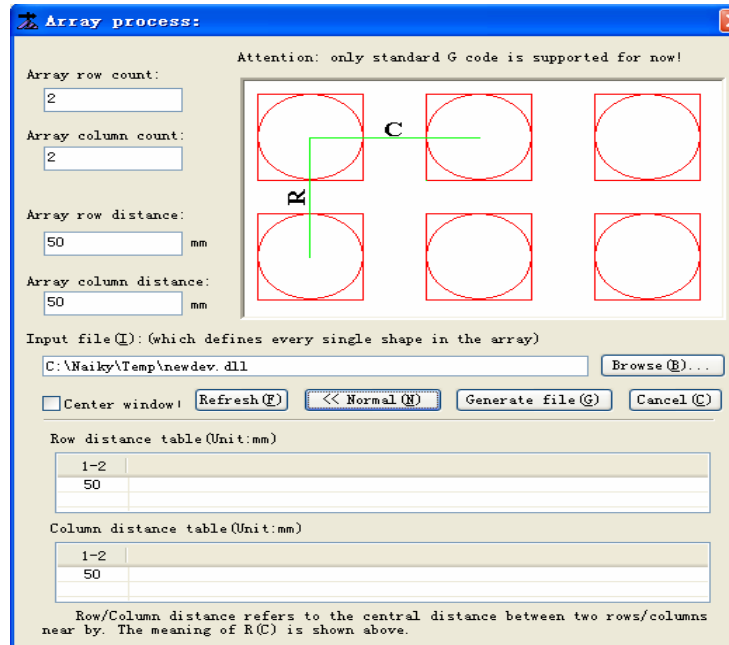
Fig. 8-8 Array process (advanced mode)

User can set every two array row (column) distances separately in "Row distance table" and "Column distance table".

◆ **[Circumrotate Mirror Process]**

This function is to execute circumrotate mirror process centered workpiece origin. After user selecting this item, a dialog will pop up as Fig. 8-9.
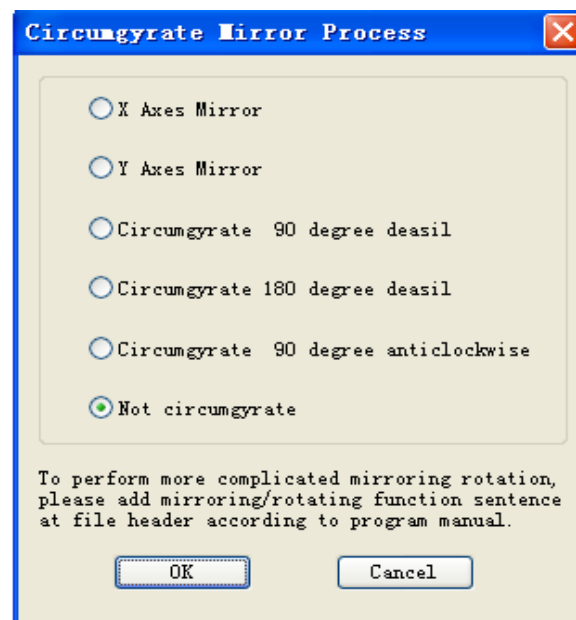
Fig. 8-9 Circumrotate mirror process dialog

◆ **[List Processing Calculated Data]**

This menu item is used for displaying machining file statistical information.

8.4　Machine tool menu

[Machine Tool menu] mainly includes items related to valve, pump and machine origin; it can realize functions of machine origin checkout, "valve or pump start/ stop", etc. See Fig. 8-10.
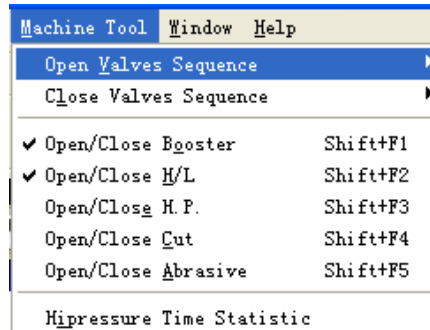
Fig. 8-10 Machine tool menu

◆　**[Open Valves Sequence], [Close Valves Sequence]**

The two menu items can be used to set the sequence to open/close valves and pumps. Their submenus supply all the combination sequences. After user selecting one sequence of them, system will open/close valves according to the selected sequence.

◆　**[Open/Close Booster], [Open/Close H / L], [Open/Close H.P.], [Open/Close Cut], [Open/Close Abrasive]**

They are corresponding to the buttons on operation panel. After user selecting one menu item, state of the corresponding button on operation panel will also be changed. Their shortcut keys in turn are Shift +F1, Shift +F2, Shift +F3, Shift +F4, Shift +F5.

◆　**[Highpressure Time Statistic]**

The menu item is used to count the time of opening/closing high-pressure valve. High-pressure equipment on waterjet machine is expensive and easily worn. So machine manufacturer needs to maintain it regularly. Select this item, system will eject a "High Pressure Time Statistic" dialog, as shown in Fig. 8-11. Button [Edit…] on the right is used to clear the time, which needs passwords to enter. Normal user only has right to look over it.
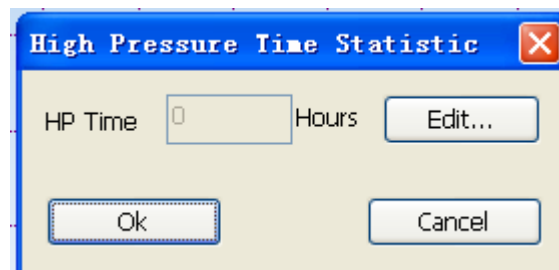
Fig. 8-11 High pressure time statistic dialog

8.5　Window menu

[Window] menu is used for interface navigation. It marks all visible windows under current mode. User can select the wanted window, which may be different under different modes. See Fig. 8-12.
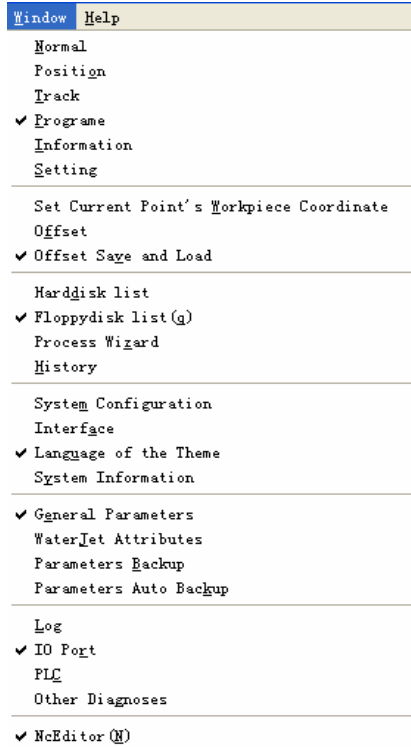
Fig. 8-12 Window menu

8.6   Help menu

The menu is used for displaying system and registration information. See Fig. 8-13.
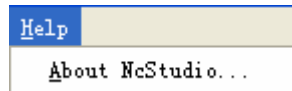
Fig. 8-13 Help menu

◆   **[About NcStudio]**

After user selecting the menu item, system will eject a dialog "About NcStudio". See Fig. 8-14.
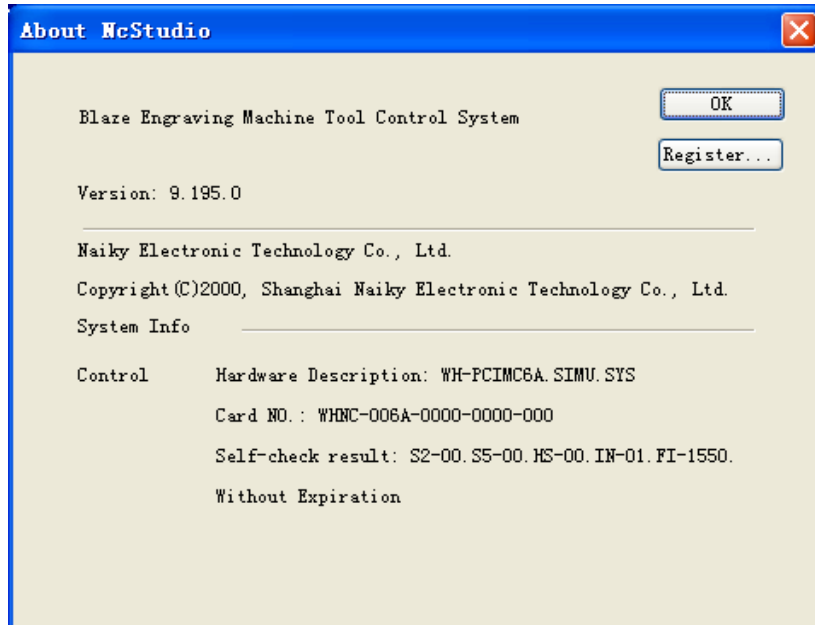
Fig. 8-14 "About Ncstudio" dialogue

From above figure, user can look up such information as version, motion card, and manufacturer. After user clicking button [Register…], system will eject a "REGISTER" dialog, as shown in Fig. 8-15, user needs to input correct register code to finish registration.
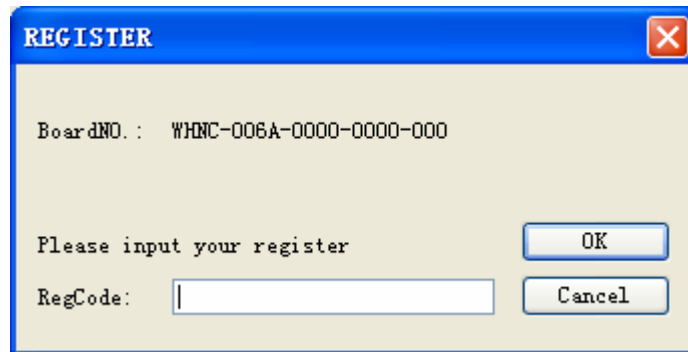
Fig. 8-15 REGISTER dialogue

9   Parameters of Ncstudio™

With abundant machining parameters, Ncstudio™ is competent for various machining tasks. This chapter will specialize in introducing the parameters involved in NcStudio™ system. The parameters are divided into two parts: general parameters and waterjet parameters. General parameters can be subdivided into operators' parameters, manufacturers' parameters and developers' parameters. Waterjet parameters are specially configured for different cutting machine tools. To look over or modify some types of parameters, user has to have enough jurisdictions.

9.1 Methods for parameter modification

Firstly, use the mouse, or press 'Tab' or 'Shift ＋ Tab' key to move the focus onto parameter list box, then press direction key (up and down direction key) to choose the wanted parameter. Finally, press 'Enter' key and input value into the parameters frame, or double click the parameter line and input value into the parameters frame. Regarding parameters whose value is "true" or "false", you can directly input "true" or "false" to modify the parameter, or directly press the buttons before the selection of "true" or "false".

> Note:
>
> 'Tab' key can be used to move the focus on the interface, while 'Tab＋Shift' will move the focus in reverse direction;
>
> In the parameter list box, only part content of the parameters can be displayed. Following two methods will teach you how to look over the complete content of the parameters: look at the lower part of the parameter interface, this window will show you the complete information about the parameter selected (recommended); press direction key (left or right direction key) in the parameter list box to look over the whole information about the parameter selected.
>
> All modification to the parameters must be done under IDLE state.

9.2 List of manufacturers' parameters

Please see the operators' parameters in Fig. 9-1.

Fig. 9-1 Operators' parameters

Manufacturers' parameters are constituted by operators' parameters and leave factory parameters, as shown in Fig. 9-2.

Fig. 9-2 The manufacturers' parameters

Parameters in V9 Ncstudio are numbered with 5 decimal digits. The first digit is main-class number (1~9). The second digit is sub-class number (0~9). Each sub-class has 1000 numbers (000~999). The groups of the parameters are offered according to the first and second digits.

In the following, we list all operators' parameters and manufacturers' parameters according to the coding number of the parameters. You can look up the parameters according to their coding number.

1.0. Axes General

◆ **N10000~N10001 AXIS_DIR（X\Y）**

Type:	INT
Unit:	None
Range:	1:Positive Direction -1:Negative Direction
Default:	-1
Valid time:	Be valid immediately without restarting the system.
Instruction:	It is used to specify incremental direction of machine coordinates of the specified axis: 1 indicates the positive increment, and -1 indicates the negative increment.

◆ **N10010~N10011 AXIS_PULSE_FACTOR（X\Y）**

Type:	Float

Unit:	mm/p（millimeter/pulse）
Range:	0．0000009~999.0
Default:	0．001
Valid time:	Be valid after restarting the system.
Instruction:	Pulse equivalent is the displacement or angle produced on the corresponding axis by each control pulse. Specifically, it refers to the least displacement which the motion control card can deal with. In stepping system, it is corresponding to a stepping pulse, then according to transmission connection, the angle displacement of the stepping pulse will be converted into linear value

◆ **N10020～N10021   N10020～N10021   WORKBENCH_LOWER_LIMIT（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	X:0.0 Y:0.0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the lower limit of the travel range of the worktable on the direction of each axis.

◆ **N10030～N10031   WORKBENCH_UPPER_LIMIT（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	Lower limit of the worktable travel range (Machine Coordinate)~99999.0
Default:	X:800.0 Y:600.0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the upper limit of the travel range of the worktable on the direction of each axis.

◆ **N10040～N10041   CHECK_WORKBENCH_RANGE（X\Y）**

Type:	BOOL
Unit:	None
Range:	0 (False): Disabled 1(True): Enabled
Default:	1（True）
Valid time:	Be valid after restarting the system.
Instruction：	It sets whether to check the worktable stroke.

1.2. Compensations

◆ **N12000　ENABLE_LEADSCREW_CMPN**

Type:	BOOL
Unit:	None
Range:	0(False): Disabled 1(True): Enabled
Default:	1（True）
Valid time:	Be valid after restarting the system.
Instruction:	It specifies whether the lead screw error compensation is valid, including backlash and screw pitch compensation.

◆ **N12001　ONLY_BACKLASH_CMPN**

Type:	BOOL
Unit:	None
Range:	0(False):Disabled 1(True): Enabled
Default:	1（True）
Valid time:	Be valid after restarting the system.
Instruction：	If it is set as "true", system will only read backlash data to compensate in terms of parameter <lead screw>. Or, system will carry out comprehensive compensation in terms of data of backlash and screw pitch.

◆ **N12010～N12011 BACKLASH（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~1.0
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the lead screw backlash of each axis. It is only effective under the condition that parameter "N12001" is set as true.

◆ **N12020 ENABLE_QE_CMPN**

Type:	BOOL
Unit:	None
Range:	0(False): Disabled 1(True): Enabled
Default:	0（False）
Valid time:	Be valid after restarting the system.
Instruction:	The parameters can be used to specify whether the cross quadrant errors compensation is valid.

QE_DATA（Group 0\ Group 1\ Group 2\ Group 3\ Group 4\ Group 5）

QE_DATA is used as the compensation parameters of the sharp-angle when the processing arc goes across the quadrant. Settings of the parameters are the same on both directions of X-axis, Y-axis and Z-axis.

◆ **N12030\ N12040 \N12050\ N12060\ N12070\ N12080 TIME**

Type:	Float
Unit:	Sec（second）
Range:	0.0~10.0
Default:	0.0
Valid time:	Be valid after restarting the system.

Instruction:	The bigger the value is, the larger the range influenced by the compensation will be. Generally, around 0.02 second is recommended.

◆ **N12031\ N12041 \N12051\ N12061\ N12071\ N12081 DISTANCE**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~10.0
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	The larger the value is, the more obvious the compensation effect is. But be noted, if the value is too large, the curve will be sinking inward, and if too small, it won't reduce the height of curve effectively. It's recommended to measure the actual height of sharp-angle by the laser interferometer in debugging the machine. The parameter is usually set as 0.3~3 times of the sharp-angle height (The effect and time of compensation are related to compensation intension).

◆ **N12032\ N12042 \N12052\ N12062\ N12072\ N12082 DELAY**

Type:	Float
Unit:	Sec（second）
Range:	0.0~10.0
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	Owing to the different mechanical properties of different machines, the position where the sharp-angle appears may not just be at the quadrant conversion point, but some distance far away from the quadrant conversion point. Please estimate the related time of this distance and set it as this parameter value.

◆ **N12033\ N12043 \N12053\ N12063\ N12073\ N12083 INTENSITY**

Type:	Float
Unit:	None
Range:	0.01~0.99
Default:	0.75
Valid time:	Be valid after restarting the system.
Instruction:	Value of the parameter affects the compensation effect. The larger the value is, the more obvious the compensation effect will be.

1.3. Velo/Acc Limits

◆ **N13000～N13001　AX_MAX_VOL（X\Y）**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.001~100000.0
Default:	48000．0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the maximum allowed acceleration of each axis.

4.2.

◆ **N42000　RED_LIGHT_WHEN_FINISH**

Type：	BOOL
Unit：	None
Range：	0 (False): Not prompt 1 (True): Prompt
Default：	0(False)
Valid time:	Be valid after restarting the system.

Instruction：	It sets whether to use red light for tips in ending processing until external input generated by mouse or keypad is received.

5.2. Handwheel

◆ **N52001　HW_COUNT_STRICTLY**

Type:	BOOL
Unit:	None
Range:	0(False): Disabled 1(True): Enabled
Default:	0（False）
Valid time:	Be valid after restarting the system.
Instruction:	If the parameter is set as "valid", the moving distance of machine will be strictly corresponding to the count of the handwheel pulse. But if the handwheel rotates too fast, the machine would likely keep moving quite a long time even if the handwheel has stopped. If the parameter is set as "invalid", the system may have rapid response time to the rotation of handwheel. But when the handwheel moves too fast, the moving distance of machine tool may not be consist with the rotation distance of handwheel.

◆ **N52002　HW_DIRECTION**

Type:	INT
Unit:	None
Range:	1, Maintain the original moving direction of the machine tool during the motion of handwheel. -1, Maintain the original moving direction of the machine tool during the reverse motion of handwheel.
Default:	-1
Valid time:	Be valid after restarting the system.

Instruction:	The parameter can be used to set the positive and negative moving direction of the lathe when the handwheel rotates. Specifically, if the parameter is set as '1' and the handwheel runs clockwise, the machine tool will move in the positive direction. If the parameter is set as '-1' and the handwheel runs clockwise, the machine tool will move in the negative direction.

◆ **N52003   HW_MULTIPLE_1**

Type:	Float
Unit:	None
Range:	0.001~10.0
Default:	0.001
Valid time	Be valid after restarting the system.
Instruction：	Under the circumstance of X1 ratio, the system will take the pulse quantity produced by the handwheel as the actual pulse quantity that has been received.

◆ **N52004   HW_MULTIPLE_10**

Type:	Float
Unit:	None
Range:	0.001~10.0
Default:	0.01
Valid time	Be valid after restarting the system.
Instruction：	Under the circumstance of X10 ratio, the system will take 10 times of pulse quantity produced by the handwheel as the actual pulse quantity that has been received.

◆ **N52005   HW_MULTIPLE_100**

Type:	Float
Unit:	None
Range:	0.001~10.0

Default:	0.1
Valid time	Be valid after restarting the system.
Instruction：	Under the circumstance of X100 ratio, the system will take 100 times of pulse quantity produced by the handwheel as the actual pulse quantity that has been received.

◆ **N52006～N52007 HW_GEAR_FACTOR (numerator/denominator)**

Type:	INT
Unit:	None
Range:	1~1000
Default:	1
Valid time:	Be valid after restarting the system.
Introduction:	When the machine tool is controlled by handwheel, the parameter is used to specify the feed speed's magnification of the machine tool. User needs to set the value of the numerator and the denominator. If the numerator is small as the denominator is large, the proportion will be small, and the relative feed speed of the machine tool will be small too. On the contrary, if the numerator is big as the denominator is small, the proportion will be large, and the relative feed speed of the machine tool will be fast too.

◆ **N52010 HW_ACC**

Type:	Float
Unit:	mm/s^2（millimeter/second²）
Range:	1.0~1000.0
Default:	200.0
Valid time:	Be valid after restarting the system.
Instruction：	This parameter can set the acceleration speed when handwheel is selected. The smaller the value is, the more stable the motion will be.

5.3. Operation panel

◆ **N53009　Turn Off Ports When Tracking Back**

Type:	BOOL
Unit:	None
Range:	0 (False): Not change 1 (True): Close
Default:	1 (True)
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter is used to set whether to close ports in auto backing. If it is "true", when the port is on, the port will be closed. If it is "false", the state of the port will stay the same.

◆ **N53010　Turn On Ports When Tracking Forward**

Type:	BOOL
Unit:	None
Range:	0 (False): Not change 1 (True): Open
Default:	1 (True)
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter is used to set whether to turn on ports in auto tracking forward. If it is "true", when the port is off, the port will be on. If it is "false", the state of the port will stay the same.

6.1. Coordinate

◆ **N61300　Enable_LocatedCoor**

Type:	BOOL
Unit:	None
Range:	0 (False): Not use 1 (True): Use
Default:	0 (False)
Valid time:	Be valid immediately without restarting the system.

Instruction:	It specifies whether to use located coordinate.

6.2. G code options

◆ **N62000   RT_DEC_LEN**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~999.0
Default:	5.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	The machine tool will decelerate when it is close to a target position on the process of positioning (moving at approach speed). The parameter indicates the distance away from the target position where the machine tool begins to decelerate.

◆ **N62001   SPEED_WHILE_APPROACHING**

Type:	Float
Unit:	mm/min（millimeter /minute）
Range:	0.001~99999
Default:	300.0
Valid time:	Be valid immediately without restarting the system.
Instruction：	In the process of positioning, it indicates the feed speed when the water jet approaches the workpiece. (Distance between the water jet and the workpiece is smaller than the distance of deceleration)

◆ **N62020   IJK_INC**

Type:	BOOL
Unit:	None
Range:	0(False）:Disabled 1(True): Enabled
Default:	1（True）

Valid time:	Be valid immediately without restarting the system.
Instruction:	It indicates that increment manner of IJK is adopted in arc programming. The value of the arc instruction address word under the incremental manner is the increment value corresponding to the centre of the circle, about which please refer to the programming manual.

◆ **N62021   RADIUS_TOL**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~9999.0
Default:	1
Valid time:	Be valid immediately without restarting the system.
Instruction:	In the IJK incremental manner of G02 and G03, it will count the radius twice. Generally speaking, the two values of the radius are not equal, and their difference is called tolerance of arc. Usually，the tolerance of arc instruction is not very large. Recommended tolerance is 10 micron.

◆ **N62090～N62091   G09_TOL（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~99.0
Default:	0.001
Valid time	Be valid immediately without restarting the system.
Instruction:	The parameter can be used together with the accurate stop command of G09 and with the parameters of pulse equivalent. Regarding the detailed instruction, please refer to the additional introduction of parameters.

N62090～N62091   G09_TOL（X\Y）additional introduction of parameters.

G09 Accurate Stop, not in simulation mode.

The instruction is only effective on the specified procedure block. To execute this command, the water jet will decelerate at the end of the procedure block, and implement the arriving check, and then execute the next block.

If user is critical of the accuracy and the machine is equipped with the device of grating measure or encoder, please connect the device of grating measure (encoder) with the corresponding port of the Ncstudio motion control card. When the system implements the accurate stop, the water jet will decelerate to stop at the end point of the procedure block. In the meantime, system will compare the water jet actual position transmitted back by the device of grating measure (encoder) with the target position. If the warp is larger than the tolerance, system will make fine adjustment and check again the warp until it is within the tolerance. If the serial motions have not been completed within the limited time, an error will be reported. To this point, user needs to regulate the following parameters or properly increase the tolerance: N11302【ACPOS_SAMPLE_INTERVAL】,N11320～N11321【ACPOS_ENCODER_DIR】、N11330～N11331【ACPOS_ENCODER_PULSE_FACTOR】、N11340～N11341【ACPOS_ENCODER_GAIN】.

6.3. Track control

◆ **N63001   TRACK_CORNER_TOL**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~0.1
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	For the fine finish of the whole workpiece, at the connection point between two procedure blocks, the cutter may not move to the specified position accurately. When the difference between the actual position of cutter and the specified position is equal to the value of this parameter, the system will regard it as the completion of this procedure block.

◆ **N63002   EXACT_STOP_TIME**

Type:	Float
Unit:	s（second）
Range:	0.0~999.0
Default:	0.0

Valid time:	Be valid immediately without restarting the system.
Instruction:	During the processing, because of the different inertia of each axis, the servo system may meet lag phenomenon. This parameter is used to overcome the lag phenomenon produced by the servo system.

◆ **N63006   SMOOTH_TIME**

Type:	Float
Unit:	s（second）
Range:	0~0.064
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction：	The bigger the value of the parameter is, the smoother the surface of the workpiece will be. But if the value of the parameter is too large, it will affect the dimension of the workpiece. Setting the value as "0.05" is recommended.

◆ **N63007   Slow Distance of intro line**

Type:	Float
Unit:	None
Range:	0~2.0
Default:	1.5
Valid time:	Be valid immediately without restarting the system.
Instruction：	It specifies that a section of segment will be marked out on the entrance of lead-in line to set a comparatively low speed.

◆ **N63008   Whether the Slow Distance of intro line is valid**

Type:	BOOL
Unit:	None
Range:	0 (False): Disabled 1 (True): Enabled

Default:	0 (False)
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies whether the lead-in line deceleration distance is valid.

◆ **N63009   The Slow Distance of intro line is near to the start of intro line or to the figure**

Type:	BOOL
Unit:	None
Range:	0 (False): Near to the figure 1 (True): Near to the start point of lead-in line
Default:	0 (False)
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies whether the lead-in line distance is near to its start point or to the figure.

◆ **N63010   Whether link the next graph with one line**

Type:	BOOL
Unit:	None
Range:	0 (False): Not link 1 (True): Link
Default:	0 (False)
Valid time:	Be valid immediately without restarting the system.
Instruction:	It decides whether to add connecting line between independent figures.

6.4. Speed/Acc

◆ **N64000   AX_STARTUP_VOL**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~ 100000.0
Default:	0.0

Valid time:	Be valid immediately without restarting the system.
Instruction:	The parameter is corresponding to the jump frequency of the servo motor, which is the minimum speed of the machine motion.

◆ **N64001   AX_CORNER_VOL**

Type:	Float
Unit:	None
Range:	0.0~ 100.0
Default:	100.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets the percentage of turning speed to normal processing speed.

◆ **N64020   PAPID_TRAVEL_FEEDRATE**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~10000.0
Default:	3000.0
Valid time:	Be valid immediately without restarting the system.
Instruction：	It is the default running speed (not the processing speed) of the machine tool in positioning (under G00 command).

◆ **N64021   Slow Distance of intro line**

Type:	Float
Unit:	None
Range:	0.0~100.0
Default:	50.0
Valid time:	Be valid immediately without restarting the system.

Instruction：	It sets the percentage of speed of lead-in line deceleration distance (namely, the percentage of actual processing speed, which is lower than processing speed).

◆ **N64101　AX_LINEAR_POST_ACC**

Type:	Float
Unit:	mm/s^2（millimeter/second^2）
Range:	0.001~100000.0
Default:	400.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the maximum acceleration of each axis when machine is positioning.

◆ **N64102　Z_LINEAR_ACC**

Type:	Float
Unit:	mm/s^2（millimeter/second^2）
Range:	0.001~100000.0
Default:	400.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the maximum acceleration of Z-axis.

◆ **N64150　AX_ACC_ACC**

Type:	Float
Unit:	mm/s^3（millimeter/second^3）
Range:	0.001~10000000000.0
Default:	10000.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the growth rate of single axis acceleration (acceleration of acceleration).

◆ **N64203　INTERP_ALGO**

Type:	INT
Unit:	None
Range:	0, Echelon Algorithm 1, Type S Algorithm 2, LEP Algorithm
Default:	1
Valid time:	Be valid immediately without restarting the system.
Instruction:	To decrease error, please choose the most suitable interpolation algorithm after debugging.

◆ **N64210 ENABLE_MCM**

Type:	BOOL
Unit:	None
Range:	0: Invalid 1: Valid
Default:	0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies whether the system will process steadily. When this parameter is valid, the machine will be more stable during processing, but the processing speed will slow down.

◆ **N64211 CUR_EFF_SCHEME_INDEX**

Type：	INT
Unit：	None
Range：	0-4
Default：	0
Valid time:	Be valid immediately without restarting the system.
Instruction：	It is used to specify an efficiency strategy group.

◆ **N64212\N64217\N64222\N64227\N64232 ANGLEACCGENE**

Type：	Float

Unit：	None
Range：	None
Default：	Group 0: 1 Group 1: 1.5 Group 2: 2 Group 3: 2.5 Group 4: 3
Valid time:	Be valid immediately without restarting the system.
Instruction：	None

◆ **N64213\N64218\N64223\N64228\N64237 ACCGENE**

Type：	Float
Unit：	None
Range：	None
Default：	Group 0: 1 Group 1: 1.1 Group 2: 1.2 Group 3: 1.3 Group 4: 1.5
Valid time:	Be valid immediately without restarting the system.
Instruction：	None

◆ **N64214\N64219\N64224\N64229\N64234 CIRCLELIMITSPEEDGENE**

Type：	Float
Unit：	None
Range：	None
Default：	Group 0: 1 Group 1: 1.2 Group 2: 1.6 Group 3: 2 Group 4: 999
Valid time:	Be valid immediately without restarting the system.
Instruction：	None

◆ **N64240   SLIDE_TIME_GENE**

Type：	Float
Unit：	None
Range：	0.01-10
Default：	1
Valid time:	Be valid immediately without restarting the system.
Instruction：	It is used to specify the ratio of unit processing time to control periodic time when the speed is smooth.

6.5. File translation

DXF file translation parameters include parameters of PLT, DXF and ENG file.

◆ **N65002   TOOL_DISTANCE_AT_AREA (PLT_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0001~99999.0
Default:	0.025
Valid time:	Be valid after loading processing procedure again.
Instruction：	It specifies distance between tools during machining in PLT area.

◆ **N65103   FIRST_POINT_AS_ORIGIN (DXF_FORMAT_COVERT_PARAM)**

Type:	BOOL
Unit:	None
Range:	0(False): Not take the starting point as the zero point. 1(True): Take the starting point as the zero point.
Default:	1(True)
Valid time:	Be valid after loading processing procedure again.
Instruction：	It specifies whether to set the first coordinate point as zero point when using the DXF processing file.

◆ **N65105   ENABLE_MACHINE_BOTTOM (DXF_FORMAT_CONVERT_PARAM)**

Type：	BOOL
Unit：	None
Range：	0(False): Invalid 1(True): Valid
Default：	0(False)
Valid time:	Be valid after loading processing procedure again.
Instruction：	It indicates execution of valve operation every time on the surface of workpiece during the tridimensional cutting.

◆ **N65106   Force To Recognize Dxf File As Metric Sys (DXF_FORMAT_CONVERT_PARAM)**

Type：	BOOL
Unit：	None
Range：	0(False):Invalid 1(True): Valid
Default：	0(False)
Valid time:	Be valid after loading processing procedure again.
Instruction：	It specifies whether to recognize DXF format files as metric size.

◆ **N65202   Recycle Machining Times (ENG_FORMAT_CONVERT_PARAM)**

Type：	INT
Unit：	None
Range：	1~9999
Default：	1
Valid time:	Be valid after loading processing procedure again.
Instruction：	It decides the cycle times when machining ENG format files.

◆ **N65203   Machining Eng File By Tool Number(ENG_FORMAT_CONVERT_PARAM)**

Type：	BOOL
Unit：	None
Range：	0(False):Not use 1(True): Use
Default：	0(False)
Valid time:	Be valid after loading processing procedure again.
Instruction：	It sets whether to use selected tool number to machine the ENG format files.

◆ **N65300 FORCE_CONVERT_NCE**

Type：	BOOL
Unit：	None
Range：	0(False): Not convert to NCE format files 1(True): Convert to NCE format files
Default：	0(False)
Valid time:	Be valid immediately without restarting the system.
Instruction：	It sets whether to directly convert a loaded file to NCE format file.

7.1. Manu

◆ **N71000 JOG_VOL**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~3000.0
Default:	1200.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	The speed specified by this parameter is the default maximum running speed of system when user presses the direction keys under manual mode.

◆ **N71001 RAPID_JOG_VOL**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~3000.0
Default:	3000.0
Valid time：	Be valid immediately without restarting the system.
Instruction：	The speed specified by this parameter is the default running speed of system when user presses down both direction key and acceleration key under manual mode.

◆ **N71021   JIGGLE_VOL**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~ the maximum supported speed of hardware
Default:	1200.0
Valid time:	Be valid immediately without restarting the system.
Instruction：	The speed specified by this parameter is the jiggle speed under manual mode.

7.2. Auto

◆ **N72001   SPEED_ASSIGN_TYPE**

Type：	INT
Unit：	None
Range：	0: Use the speed of file 1: Use the default speed 2: Specify the speed according to ratio
Default：	0
valid time：	Be valid immediately without restarting the system.
Instruction：	It sets the style of feed speed.

◆ **N72002   IGNORE_PROG_SPINDLE_REV**

Type：	BOOL
Unit：	None
Range：	0 (False）：Not neglect 1 (True）：Neglect
Default：	0 (False)
Valid time：	Be valid immediately without restarting the system.
Instruction：	If this parameter is "True", the specified revolving speed in machining file will be invalid. Otherwise, the system will adopt the revolving speed in procedure.

◆ **N72006　HEIGHT_Z_RAISE**

Type：	Float
Unit：	mm（millimeter）
Range：	None
Default：	5.0
Valid time：	Be valid immediately without restarting the system.
Instruction：	It sets cutter relieving height of dry running

◆ **N72010~N72011　ENABLE_WORKAREA_CHECK（X\Y）**

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default：	0（False）
Valid time:	Be valid immediately without restarting the system.
Instruction:	It is used to specify whether to open the workpiece coordinates range check. The range is constituted by the upper limit and the lower limit of the workpiece coordinates.

◆ **N72020~N72021　WORKAREA_LOWER_LIMIT（X\Y）**

Type:	Float

Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	-10000．0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the lower limit of workpiece coordinates on each direction when parameter "ENABLE_WORKAREA_CHECK" is valid.

◆ **N72030~N72031　WORKAREA_UPPER_LIMIT（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	10000.0
Valid time:	Be valid immediately without restarting the system.
Instruction：	It is used to specify the upper limit of the workpiece coordinates on each direction when parameter "ENABLE_WORKAREA_CHECK" is valid.

7.4. Bkref

◆ **N74000　CANCEL_REFPT_AFTER_ESTOP**

Type:	BOOL
Unit:	None
Range:	0 (False): Not cancel 1 (True): Cancel
Default:	1（True）
Valid time:	Be valid immediately without restarting the system.
Instruction:	If the parameter is "true", once there is ESTOP, the current symbol of reference point will be cleared.

◆ **N74001　NEED_REFPT_BEFORE_MACHINING**

Type:	BOOL

Unit:	None
Range:	0 (False）: Needn't to back to he reference point 1 (True）: Need to back to the reference point
Default:	1（True）
Valid time:	Be valid immediately without restarting the system.
Instruction:	Servo system adopts incremental coding machine, but the system can only recognize the previous corresponding position. In order to let an arbitrary point have a fixed coordinates, user needs to select a reference point which is the mechanical origin point. During the processing, if an ESTOP or power off occurs, an error of displacement may appear when machine restarts. With the function of backing to the mechanical origin, the system can retrieve the coordinates of the workpiece to ensure the accuracy to continue the processing.

◆ **N74010~N74011　REFPT_COOR（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	lower limit of the worktable travel range (machine coordinate)~ upper limit of the worktable travel range (machine coordinate)
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	The coordinates of the position of the mechanical origin is fixed according to the parameters setting of [lower limit of the worktable travel range] and [upper limit of the worktable travel range].

◆ **N74020~N74021　COARSE_LOCATING_DIR（X\Y）**

Type:	INT
Unit:	None
Range:	1: Positive direction -1: Negative direction

Default:	X: -1 Y: -1
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter can be used to set the motion direction of coarse positioning during the process of backing to the reference point.

◆ **N74030~N74031 COARSE_LOCATING_VELO（X\Y）**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.001~10000.0
Default:	X:1800.0 Y:1800.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter can be used to set the feed speed of coarse positioning during the process of backing to the reference point.

◆ **N74050~N74051 FINE_LOCATING_DIR（X\Y）**

Type:	INT
Unit:	None
Range:	1: Positive direction -1: Negative direction
Default:	X: 1 Y: 1
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter can be used to set the motion direction of fine positioning during the process of backing to the reference point.

◆ **N74060~N74061 FINE_LOCATING_VELO（X\Y）**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.001~10000.0

Default:	X:60.0 Y:60.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter can be used to set the feed speed of fine positioning during the process of backing to the reference point.

◆ **N74080~N74081   BACK_DISTANCE（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	-1000.0~1000.0
Default:	X:2.0 Y:2.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It refers to the additional moving distance after the completion of fine positioning. Its value can be positive or negative. If the value is 0, there is no additional moving distance. And if the value is not 0, the mechanical origin will be located on the tracking back distance of the corresponding coordinates beside the zero switch.

◆ **N74090   Location Times**

Type：	INT
Unit：	None
Range：	None
Default：	1
Valid time:	Be valid immediately without restarting the system.
Instruction：	This parameter can be used for counting the times of fine positioning during backing to the reference point.

◆ **N74091   REFPT_COOR_X1**

Type：	Float
Unit：	None

Range：	-100000.0~100000.0
Default：	0.0
Valid time:	Be valid after restarting the system.
Instruction：	It sets the machine coordinates of mechanical origin.

◆ **N74092 REFPT_COOR_X2**

Type：	Float
Unit：	None
Range：	-100000.0~100000.0
Default：	2000.0
Valid time:	Be valid after restarting the system.
Instruction：	It sets the machine coordinates of mechanical origin.

◆ **N74094 TOOL_WIDTHX**

Type：	Float
Unit：	mm (millimeter)
Range：	0.0~100.0
Default：	50.0
Valid time:	Be valid immediately without restarting the system.
Instruction：	It sets the width of tool in X direction.

7.5. Measure

◆ **N75023   Light Control Outport Index**

Type：	INT
Unit：	None
Range：	1: Without disconnection with tool calibration line -1: With disconnection with tool calibration line
Default：	-1

Valid time:	Be valid immediately without restarting the system.
Instruction：	It sets whether the software has the function of disconnection with tool calibration line. If it is "-1", the port will be given a signal to disconnect with tool calibration line at processing and another signal to connect with calibration line during tool calibration.

7.9. Operation others

◆ **N79100   Action After Programming**

Type：	INT
Unit：	None
Range：	0: Not back to the fixed point 1: Back to the fixed point 2: Back to workpiece origin
Default：	0
Valid time:	Be valid immediately without restarting the system.
Instruction：	To improve efficiency, user may need the machine tool move to a fixed position after the processing is finished. For example, raise Z-axis to a fixed position after the completion of the workpiece processing to replace another workpiece. If user wants the machine tool to back to a fixed point automatically after processing, please choose this parameter and set the fixed point.

◆ **N79110~N79111   FIXPT_POS（X\Y）**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	0.0
Valid time:	Be valid immediately without restarting the system.

Instruction:	This parameter is only valid on condition that the [AutoBack_to_the_FixedPoint_Valid_after_Processing] parameter is valid. It is used to indicate the machine coordinate of the fixed position.

◆ **N79120 Clear View After Programming**

Type:	BOOL
Unit:	None
Range:	0 (False): Not clear 1 (True): Clear
Default:	0 (False)
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets whether to clear the track after machining is finished.

◆ **N79200 Display Simulation Out of Mechanic Travel**

Type：	BOOL
Unit：	None
Range：	0 (False): Not clew 1 (True): Clew
Default：	1(True)
Valid time:	Be valid immediately without restarting the system.
Instruction：	It specifies whether to clew under the situation of out of mechanic travel in simulation mode.

◆ **N79400 Auto back to zero**

Type：	BOOL
Unit：	None
Range：	0 (False): Not back to machine origin 1 (True): Back to machine origin
Default：	0 (False)
Valid time:	Be valid after restarting the system.
Instruction：	It specifies whether to back to the machine origin automatically.

8.1. Position view

◆ **N81000   AUTO_LOAD_TRACK**

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default:	1（True）
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets whether system will analyze the processing track automatically after the machining file is loaded, which will help user load the track in advance.

◆ **N81001   AUTO_LOAD_TRACK_LIMIT**

Type:	INT
Unit:	KB
Range:	0~100000
Default:	1000
Valid time:	Be valid immediately without restarting the system.
Instruction:	When the size of the processing file is smaller than the value of this parameter, system will analyze the processing track automatically; but when the size of the processing file is equal to or larger than this value, system will not analyze the track.

8.3. Program view

◆ **N83000   Y_Axis Work Range**

Type：	INT
Unit：	None
Range：	1: Positive direction 2: Negative direction

Default：	1
valid time：	Be valid immediately without restarting the system.
Instruction：	It sets the work space of Y-axis of machine tool.

9.3   WaterJet parameters

In【WaterJet】Parameters screen, there are parameters closely relative to the waterjet machine. Setting of the parameters can be carried out in this screen and the operation method is the same as that of the operators' parameters and the manufacturers' parameters. See the Fig. 9-3, as following.
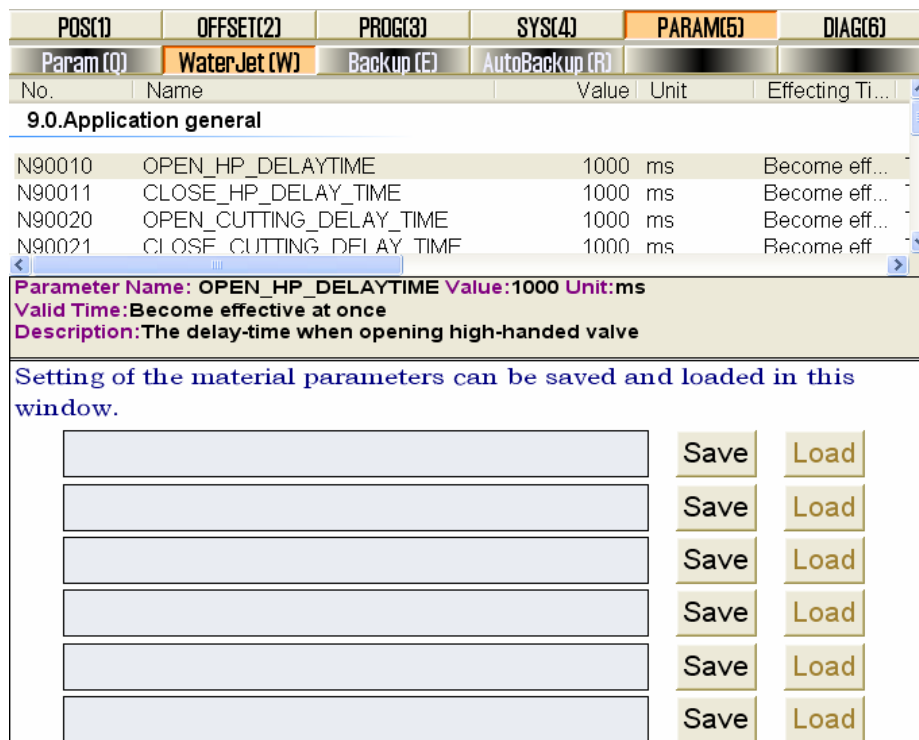
Fig. 9-3 The dialogue of water jet parameters

9.0. Application general

◆ **N90010   OPEAN_HP_DELAYTIME**

Type:	INT
Unit:	ms（millisecond）
Range:	50 ~ 600000
Default:	1000
Valid time	Be valid immediately without restarting the system.

Instruction:	It specifies the delay time of opening the high pressure valve.

◆ N90011 CLOSE_HP_DELAY_TIME

Type:	INT
Unit:	ms（millisecond）
Range:	50～600000
Default:	1000
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time of closing the high pressure valve.

◆ N90020 OPEAN_CUTTING_DELAY_TIME

Type:	INT
Unit:	ms（millisecond）
Range:	50～600000
Default:	1000
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time of opening the water valve.

◆ N90021 CLOSE_CUTTING_DELAY_TIME

Type:	INT
Unit:	ms（millisecond）
Range:	50～600000
Default:	1000
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time of closing the water valve.

◆ N90030 OPEAN_ABRASIVE_DELAY_TIME

Type:	INT

Unit:	ms（millisecond）
Range:	50 ~ 600000
Default:	1000
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time of opening the abrasive valve.

◆ **N90031 CLOSE_ABRASIVE_DELAY_TIME**

Type:	INT
Unit:	ms（millisecond）
Range:	50 ~ 600000
Default:	1000
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time of closing the abrasive valve.

◆ **N90032 PAUSE_BEFORE_CUT**

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	5000
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time before cutting.

◆ **N90034 PAUSE_AFTER_CUT**

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	0

Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies the delay time after cutting is finished.

9.3. Parameter of Programming

◆ **N93000   AX_LINEAR_MACH_ACC**

Type:	Float
Unit:	mm/s^2(millimeter/second^2)
Range:	0.001 ~ 100000.0
Default:	400.0
Valid time	Be valid immediately without restarting the system.
Instruction:	It sets the maximum acceleration of each feed axis in machining.

◆ **N93001   REF_CIRCLE_MAX_VELO**

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.001 ~ 100000.0
Default:	1800.0
Valid time	Be valid immediately without restarting the system.
Instruction:	It sets the maximum allowable speed corresponding to a circle of 10mm diameter.

◆ **N93002   CIRCLE_MIN_VELO**

Type:	Float
Unit:	mm/min （millimeter/minute）
Range:	0.001 ~ 100000.0
Default:	180.0
Valid time	Be valid immediately without restarting the system.

Instruction:	Even low circle feed speed will incur fast centripetal acceleration in machining a small radius circle. But limiting the arc speed will decelerate the machining speed. To ensure the processing efficiency, when the speed calculated by the system is lower than this value, system will regard this value as the processing speed.

◆ **N93003 DEFAULT_FEEDRATE**

Type:	Float
Unit:	mm/min （millimeter/minute）
Range:	0.0~100000.0
Default:	1200.0
Valid time	Be valid immediately without restarting the system.
Instruction:	It is the default machining speed (not the positioning speed).

◆ **N93004 AX_CON_ACC**

Type:	Float
Unit:	mm/s^2 （millimeter/second^2）
Range:	0.0~100000.0
Default:	1000.0
Valid time	Be valid immediately without restarting the system.
Instruction:	It describes the max acceleration of the feed motion on the adjacent axes. The recommended value is 1~2 times of the acceleration of the single axis.

◆ **N93006 IS_VALID_CORNER_APPROACH_SPEED_PERCENT**

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default:	0 (False)

Valid time	Be valid immediately without restarting the system.
Instruction:	It decides whether to use the approaching speed percentage.

◆ **N93007 CORNER_APPROACH_SPEED_PERCENT**

Type:	Float
Unit:	None
Range:	0.0~100.0
Default:	100.0
Valid time	Be valid immediately without restarting the system.
Instruction:	It sets the percentage of the approaching speed near the cornering point of two tracks to normal machining speed.

◆ **N93008 CORNER_APPROACH_DISTANT**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~1000.0
Default:	2.0
Valid time	Be valid immediately without restarting the system.
Instruction:	It specifies certain distance near the cornering point of two tracks. The maximum speed of this distance is limited by parameter "N93007".

10 Functions and Features of NcEditor

The features of NcEditor are as following:

■ Designed for planar cutting, editing planar picture and outputting data of planar machining objects;

■ Supporting directly processing and mating with Ncstudio;

■ Supporting the element creation of basic machining object, including point, line, polyline, rectangle, circle, ellipse, arc, star, character, etc;

■ Supporting setting processing conditions for a single object, distinguished by color, and parameters setting function;

■ Supporting object change function, including translation, rotation, mirror image, zoom, slant, combination and breakup of the object;

■ Supporting view change function, including translation, zooming in, zooming out, fit to window;

■ Supporting spot catch function, such as node and midpoint of straight line catch, center and quadrant point of circle catch;

■ Supporting view grid and ruler displaying function;

■ Supporting grid catch function;

■ Supporting mouse dragging editing of nodes function to adjust the size of the object;

■ Supporting displaying and modification functions of attribute of the object;

■ Supporting multi-object alignment functions, including "align right", "align left", "align top", "align bottom", "align vertical line", "align center point", "align level dispersion" and "align vertical dispersion";

■ Supporting object selection function;

■ Supporting auto and manual bridge addition function;

■ Supporting removal of duplicate objects;

■ Supporting inserting other format of programmable files into the existed processing file; Supporting saving parts of a processing file as another processing file;

■ Supporting basic editing functions: copy, cut and paste;

■ Supporting analog processing function;

■ Supporting read and transformation of ENG files, G code files, DXF files and PLT files;

■ Supporting Chinese and English bilingual interface, internal exchange of the two languages;

- Supporting resetting and adjusting of round spot, especially for the file with round spot mark, setting the position of processing beginning spot on the round spot after reading the file;

- Supporting viewing and editing of text file;

- Easy to operate, fast and quickly to learn and master.

10.1 Integral interface

The integral interface of NcEditor includes: [title bar], [menu bar], [toolbar], [drawing toolbar], [status bar], [machining parameters attribute column], [object editor space], [object list bar], [ruler bar]. Among them, the [object list bar] includes: [object list window], [object property window]; [machining parameters attribute column] includes: [machining parameters color window], [machining parameters modification window]. See Fig.10-1
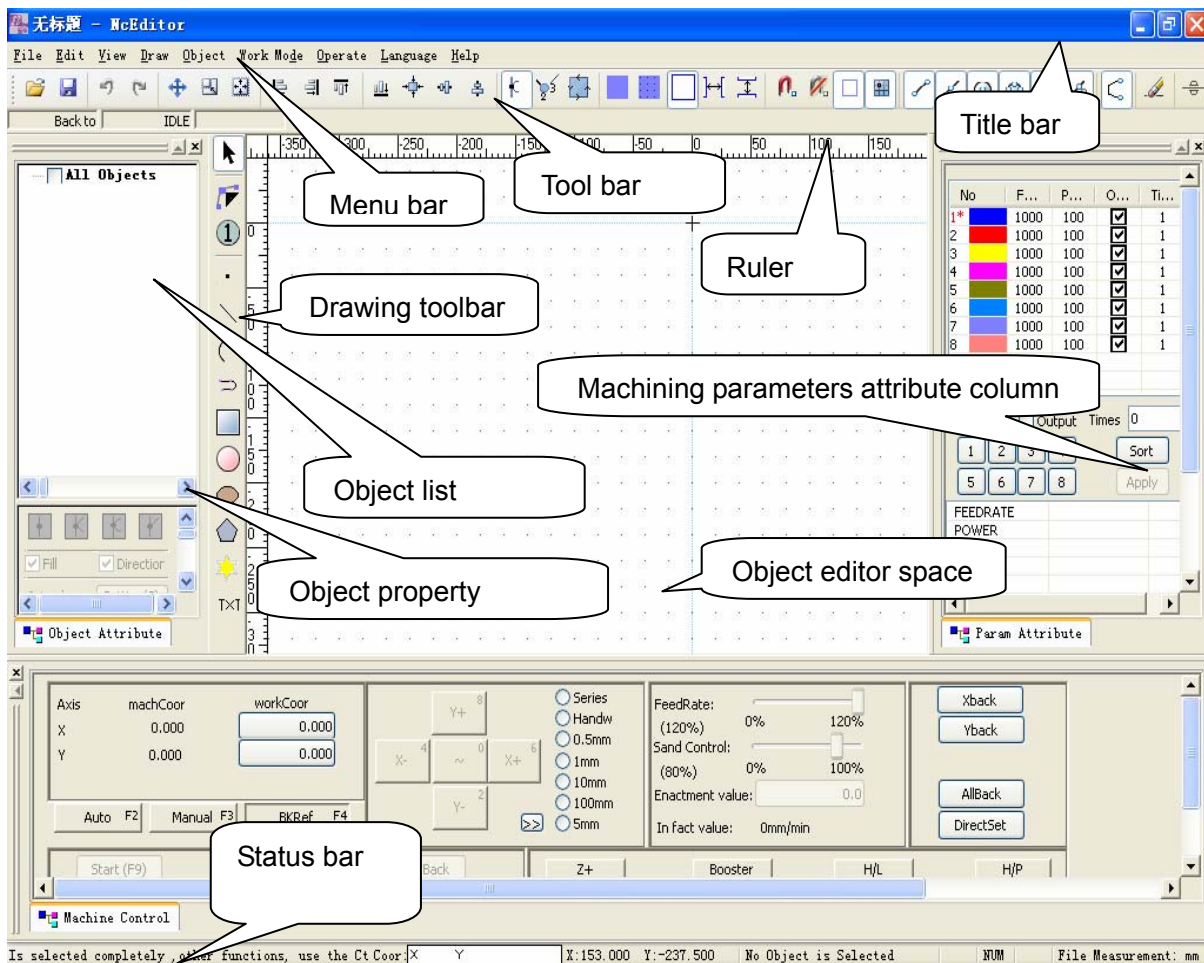
Fig 10-1 Integral interface dialogue

10.2 Object list window

The object in [Object list window] is corresponding to what in [Object editor space]; when you draw an object in the [Object editor space], a corresponding object will be added automatically in [Object list window]; when you select or cancel selecting an object in [Object list window], the object in [Object editor space] will be executed the same operation.

The concrete functions of [Object list window] are as following:

➢ Selecting the box ahead of the object can select a single object, or several objects;

➢ Cancelling the selected box ahead of the object can cancel selection of a single object, or

several objects;

➤ The selected multiple objects can be set as one group;

➤ Breaking the objects in one group will lead them back to the previous group;

➤ Setting a group as the current working group: when a file includes multiple groups, if you want to edit one of these groups, you have to set the group as the current group.

10.3 Object attribute window

[Object attribute window] is mainly used to set the property of the selected object, involving lead-in line setting, filling or not, the precise position of object in coordinate system and machining direction setting, etc.

10.3.1 Classification of lead-in line

[Lead-in line] is exclusive for closed figures, which can be classified into 4 types according to their shapes: none lead-in line, tine lead-in line, arc lead-in line and kerf lead-in line. The type of [lead-in line] is chosen in terms of the cutting.

◆ **[None lead-in line]**

It means no lead-in line, you can decide whether to set lead-in line or not.

Fig. 10-2 None [lead-in line]

◆ **[Tine lead-in line]**

Cutting-in edge and cutting-out edge are all straight lines, tools cut in along one straight line of the taper angle and cut out along the other straight line. For instance, [lead-in line] is on the peak of the rectangle, or on the top of other cross-points.
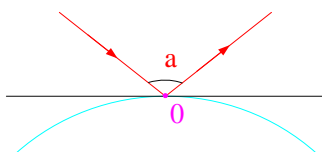
Fig. 10-3 [Lead-in line] of taper angle

◆ **[Arc lead-in line]**

Cutting-in edge and cutting-out edge are all arcs; tools cut in along one edge of the arc and cut out along the other edge, such as, lead-in line of circle, ellipse and arc.
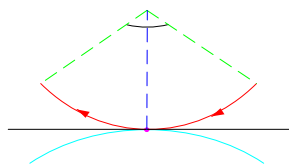
Fig. 10-4 Arc [lead-in line]

◆ **[Kerf lead-in line]**

Operating principle: set only cutting-in [lead-in line], without cutting-out [lead-in line], when the tool cuts to the tail, the tool will return back along the same path and stop after travelling some distance.
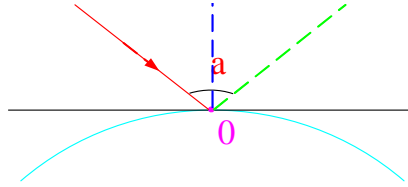
Fig. 10-5 Kerf [lead-in line]
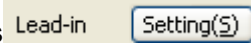
Note: there is no definite limitation on the usage of tine lead-in line, arc lead-in line and kerf lead-in line. They are interchangeable. For instance, setting an arc lead-in line for rectangle vertex, the system will figure out the tine lead-in line automatically; the type of lead-in line is chosen in terms of the cutting; besides, the use of lead-in line is for precisely machining, you can decide whether to set it or not.

[Lead-in line] can be divided into internal lead-in line and external lead-in line according the reserved parts after machining.

If you select filling the object, [lead-in line] will be set as internal one by system; if you select not filling the object, [lead-in line] will be set as external one by system. The detail is as following:

➢ External [lead-in line]: the inner part of object will be reserved, while the outer one will be abandoned.

➢ Internal [lead-in line]: the outer part of object will be reserved, while the inner one will be abandoned.

10.3.2 Setting lead-in line

Method one: select a closed object in the [Object list window]. The top icons of [Object list window] are the varieties of lead-in lines. From left to right in the dialogue, they are [None lead-in line], [Tine lead-in line], [Arc lead-in line], [Kerf lead-in line]. You can press the button to choose one of lead-in lines according to cutting process; on the lower part of the [Object attribute window], shown as Lead-in Setting(S). Pressing the button [Setting], system will eject a dialogue of [Lead-in line] setting as in Fig. 10-6.

Method two: press the button on [object editor space] to select a closed object →right-click the object, a menu bar will pop up → select the option [Set introduce]→ "Set Introduce" dialogue will pop up.

Method three: select a closed object by [Object list window] or by the button on drawing toolbar →choose the [Object] menu → select [Set introduce] option →"Set Introduce" dialogue will pop up.
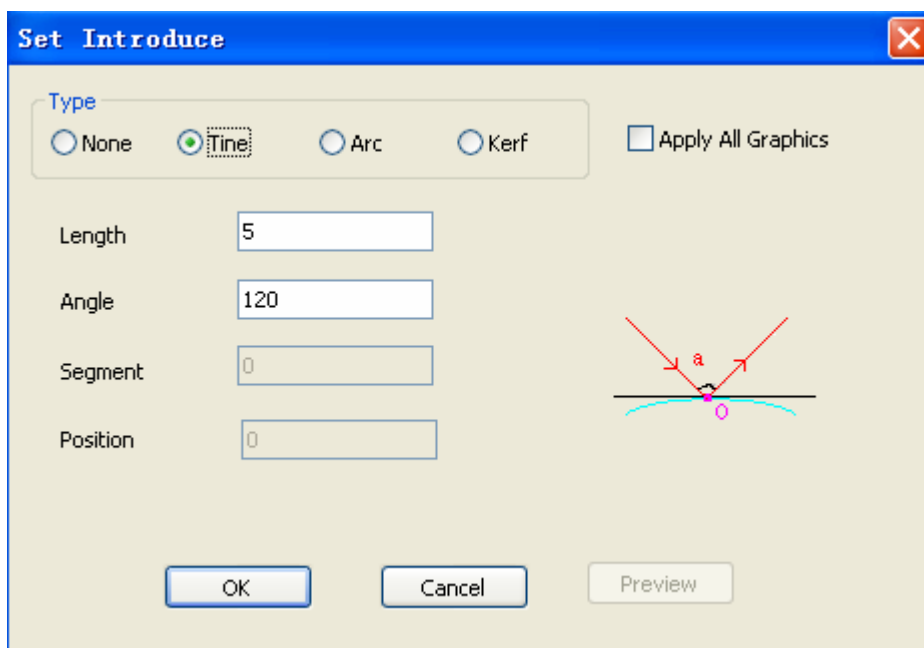
Fig. 10-6 Setting lead-in line dialogue

In the dialogue [Set Introduce], you can set type, length, angle, segment and position ratio. The concrete definitions are as following:

➢ [Length]: the length of cutting-in edge or cutting-out edge of lead-in line;

➢ [Angle]: the angle between the two sides of lead-in line, like the size of angle "a" in Fig. 10-2, Fig. 10-3, Fig. 10-4, Fig. 10-5.

➢ [Segment]: there are probable N straight lines (N is natural number) on the selected object. ''Segment No.'' refers to the sequence number of the segment where the lead-in line is on among all the segments of the object (It is used to fix on which segment the location of lead-in line is).

Sort segment: choosing a closed object or a combination of multiple objects, there will be a remark or marks on the object. Choose [view direction] under [view] menu. You can modify the direction in [object attribute window]. Sorting begins from the remark or remarks, in line with the machining direction. The first order number is 0, and you can set the segment number according to your own desire.

➢ [Position]: it refers to the ratio of distance from the position of lead-in line on segment to the beginning of this closed segment to the whole length of the segment, whose range is (0, 1).

➢ [Apply All Graphics]: if this function is not selected, there will not be lead-in line on graphic after pressing "ok", or there will be lead-in lines on all graphics. If the function of [Apply All Graphics] is selected, pressing "ok" and "preview", there will be a dialogue as following if the segment setting is not fit for all the graphics (Different range will be displayed for different graphics, depending on actual situation).
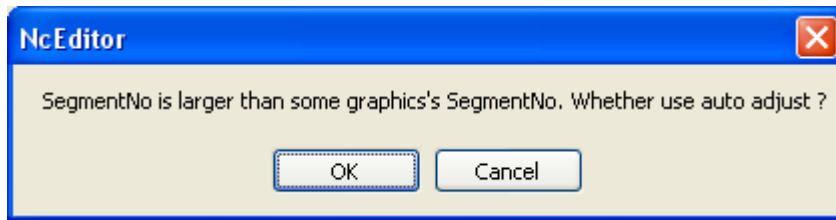
Fig. 10-7 Inquiry dialogue

If the type of lead-in line is kerf, the dialogue is as following:
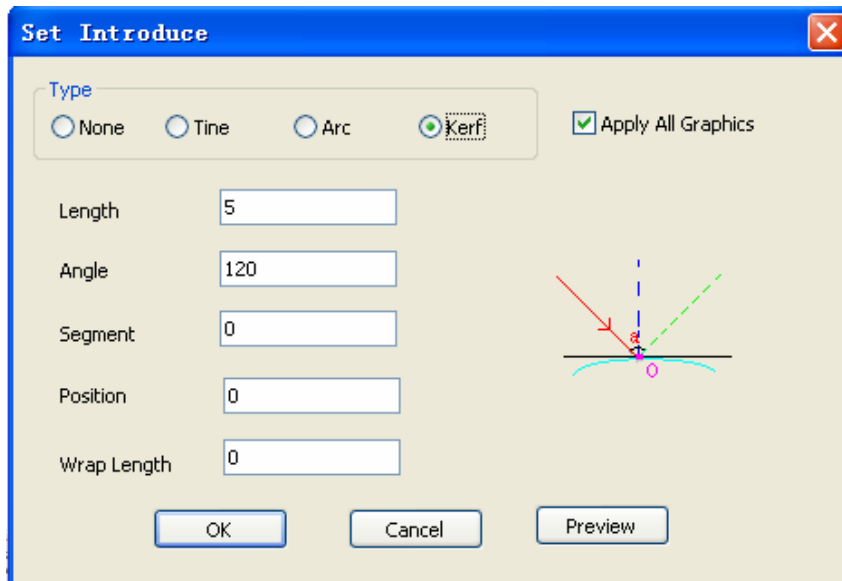
Fig. 10-8 Setting [kerf lead-in line] dialogue

The dialogue of kerf lead-in line holds one more function "wrap length" than tine lead-in line dialogue.

Definition of [Wrap length]: when tool cuts until cut-in point, it will go forward along the same path before stopping, the repeated path distance from cut-in point to the stop point is called [Wrap length].

An example for setting lead-in line:

Note: [Lead-in line] is only available in closed graphics. For polylines, only polylines finished with shortcut key "Shift+C" can be set lead-in line.

Drawing a triangle with polylines, you should finish drawing by shortcut key "Shift+C" (note: the input method should be English), as following:
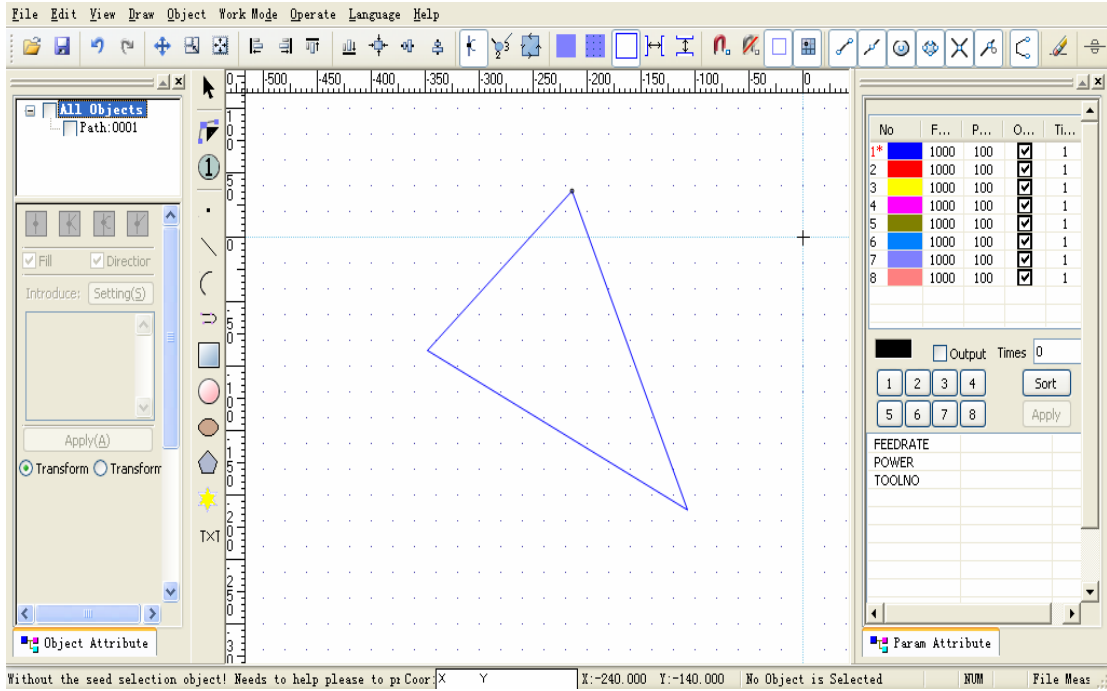
Fig. 10-9 Draw closed triangle with polylines dialogue

Select "View" menu→ select "Line Rim Model" and "View Direction". See Fig. 10-10.

Fig. 10-10 [Line Rim Dodel] and [View Direction]

Select this object in [Object list window] or via button on drawing toolbar;
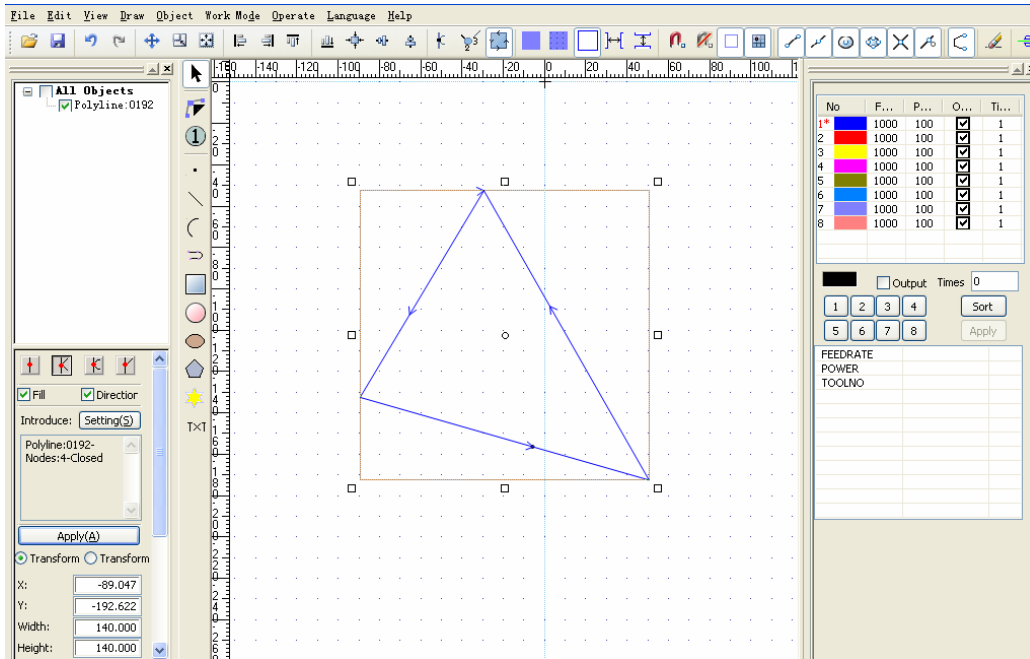
Fig. 10-11 Selection of object dialogue

Select this object → right-click mouse button→ select "set introduce" or [Lead-in Setting(S)] on [Object attribute window] or [Set introduce] under [Object] menu, input the data as following:
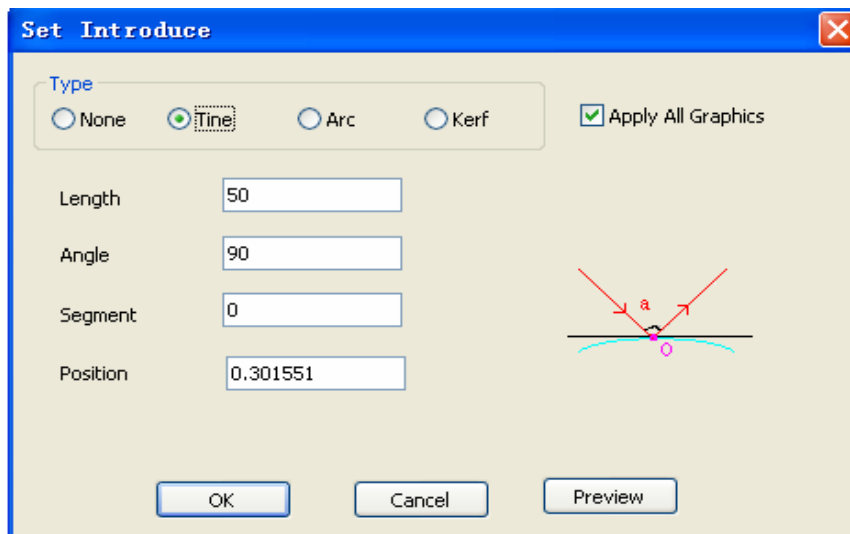
Fig. 10-12 Setting lead-in line data

Pressing "OK", the dialogue is as following:
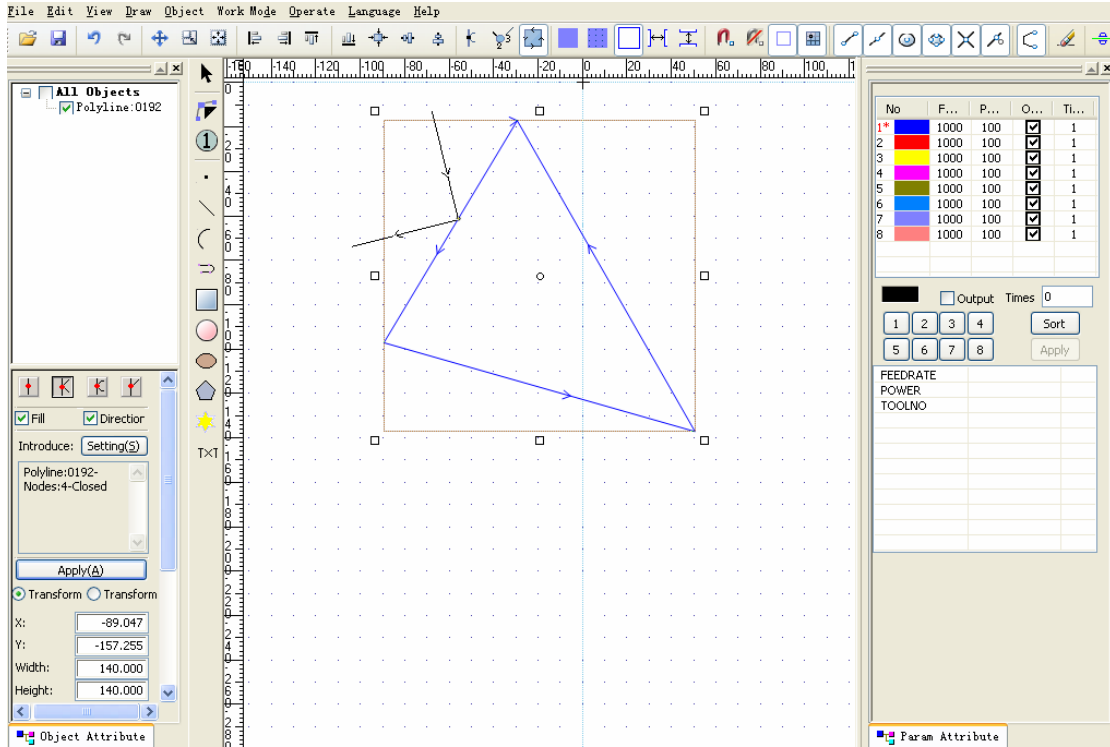
Fig. 10-13 After setting lead-in line

Besides, NcEditor can support manually setting lead-in line position. In [Set introduce], don't put in any data in [Position], then click the frame of the processing object to set the position of lead-in line.

10.3.3 Selection of color of machining objects

In the middle of [Machining parameters attribute window], there are two lines of color buttons, which are used to set the machining color of selected object. System will confirm the machining parameters by colors; same parameters share the same color. In [Machining parameters attribute window] part, we will introduce delegation of color corresponding to related parameters in detail.

◆ **Operation procedures:**

➢ You can select the box ahead of [Object list window] to choose an object, or, use the button

 on [Drawing toolbar] (only the object totally in the frame is selected);
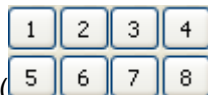
➢ Press one button in ( ), the selected object will change into the color you have chosen.

The objects with the same color are regarded as default same parameters, including velocity, power and tool NO., etc. Thus, you have to set all the objects with same parameters in the same color.

10.3.4 Fill or not

[Fill or not] of the objects are only for the closed objects.

> If you want to keep the inner parts of objects, please choose [fill], shown as ☑Fill ;

> If you only want the shape of the objects, please don't select [fill], shown as ☐Fill, the inner parts of objects will be milled.

10.3.5 Selection of machining direction

[Selection of machining direction] is used to change the machining direction of object. You can adjust it according to cutting method, workpiece texture, tool machining direction, etc.

The concrete procedure is as following:

> Open "view" menu, select "view direction", all the objects in [Object editor space] will show the machining direction;

> Select the box ahead of [Object list window] to choose an object, or, use the button ↖ on [Drawing toolbar] (only the object totally in the frame is selected);

> In the [Object attribute window], choose ☑Direction, the selected machining object will reverse direction.

> If you want to cancel the alteration of direction, you just need to cancel the selection of the choice box, see ☐Direction.

10.3.6 Precise positioning of machining objects

In order to make it more convenient for you to position machining objects precisely, NcEditor offers precise positioning.

A concrete example: drawing a circle in [Object editor space], then select this circle, as following:
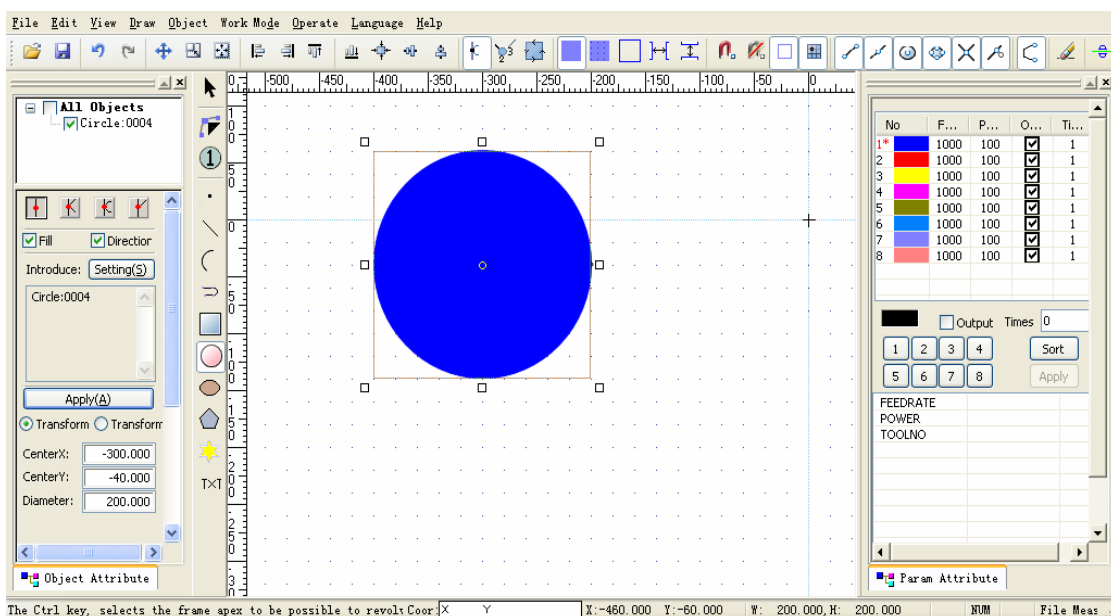
Fig. 10-14 Positioning a circle

Input the data as following in [object attribute window]:
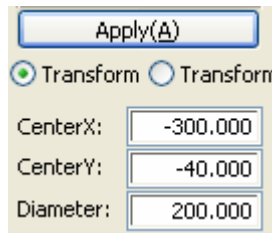
Fig. 10-15 Precise positioning window

After inputting the data, press "Apply", then this circle can be precisely positioning the position centered XY(-300,-40), radius 100.000.

If the selected object is straight-line, rectangle, arc, ellipse or point, the precise positioning dialogue will alter accordingly. The precise positioning is unavailable for the object drawn by polylines.

10.3.7 Object information diagram

When you select one or multiple objects in [Object editor space], the [Object information dialogue] will display the detailed information of the objects, for instance, name, node and fill or not (among them, "fill" selection for open figures is invalid in grey). Therefore, you can check the detail information of the object to avoid misunderstanding of object attribute under different views.

For instance, draw a picture with polyline, and select it, as Fig. 10-16.
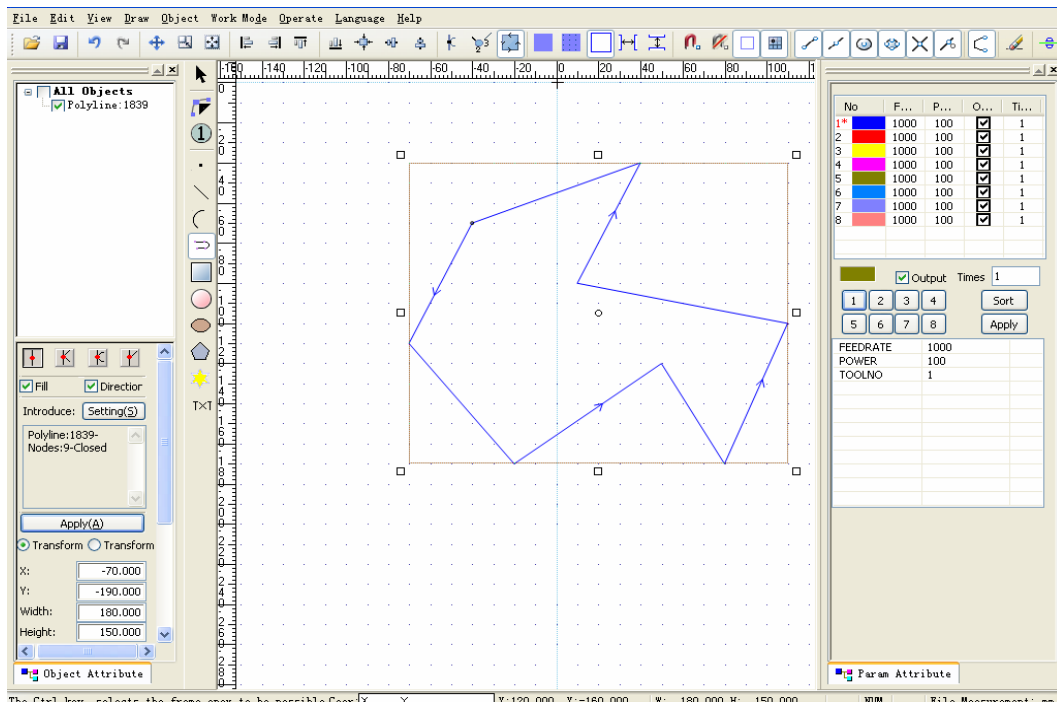
Fig. 10-16 The object drawn by polylines

From the picture above, the [object information dialogue] has displayed the name and node of objects, etc.
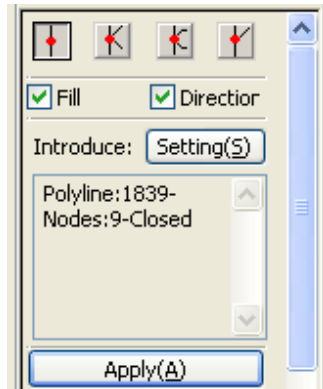
Fig. 10-17 Object information dialogue

10.4 Origin setting

The criterion of [Object editor space] is workpiece coordinate, the origin of [Object editor space] is the origin of workpiece coordinate.

In the crossing of vertical ruler and horizontal ruler, there is a mark; click the cross center and drag it into [Object editor space] and release it in the position to be set as origin. This origin is also the origin of workpiece coordinate.

The advantages of setting an origin:

➢ The position of object in workpiece coordinate can be positioned more precisely;

➢ When a file is loaded, the position of workpiece in workpiece coordinate can be more precisely fixed;

➢ It will be more convenient for you to set origin as the beginning of machining.

10.5 Machining parameters attribute bar

The machining parameters attribute bar can be divided into color window and modification value of machining parameters window.

10.5.1 Color window of machining parameters

Currently there are 8 colors in color window with corresponding number. Each color is related to different machining parameters, including feedrate, power, tool NO., etc.

10.5.2 Modification window of machining parameters

The machining objects in the same color are regarded as having the same machining parameters in NcEditor two dimensional model, so if you change the parameters related to a certain color, the machining parameters of all the objects in this color will be altered.

Choose color "4 "in [Color window of machining parameters], as following:
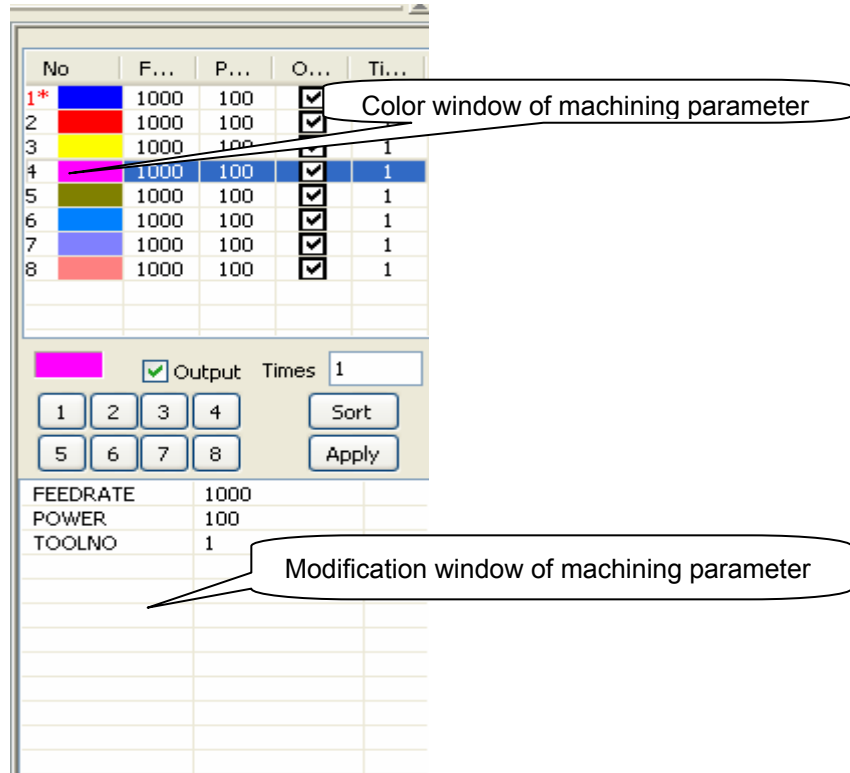
Fig. 10-18 Machining parameters attribute bar

In the [modification window of machining parameters] , the related parameters of color "4" can be modified directly, such as "feedrate", "power", "tool NO.", press" apply" to confirm modification.

10.6 Figure selection

To select a figure, you first need to ensure that there isn't other running command. Then press down the button, shown as [cursor icon]. Only the objects total in the box can be selected.

You can select a single object in [Object editor space], by following method:

➢ Click the frame of an object to select it;

➢ Select the required object in [Object list window];

➢ Click the left mouse button and move the mouse, and there will be a rectangle box with dotted line in the space to ensure the selection of object frame.

If you want to select multiple objects or object groups, the following methods can be used:

➢ Click an object frame and press the button shift / ctrl at the same time; this new object will be added into the selection.

➢ Select a group or the required objects in [Object list window];

➢ Holding down the left mouse button and moving the mouse, there will be a rectangle box with dotted line in the space. The objects total in the box will be selected, and there will be red dotted-line rectangle around the selected objects. Besides, you can hold down the left

mouse button to drag the selected objects to the required place, which is also fit for a selected object.

10.7 Menu function section

The menu items the same with those of Ncstudio will not be introduced here.

10.7.1 File menu

[File menu] includes every operation command related to machining file, as following:

Fig. 10-19 File menu of NcEditor

◆ **[New]**

Shortcut key: Ctrl + N, this menu is used to create a new NcEditor editing file (.nce).

◆ **[Open]**

Shortcut key: Ctrl + O. The menu is used for opening the processing file from the hard disk and loading it into the decoder of the system. Then the parsing engine will parse the file into G instructions, and transfer these instructions to CNC card.

After you select this menu, [Open dialogue] will pop up, as Fig. 10-20. After selecting the machining file, press "open" to load it automatically; you can view it clearly, since the file name will appear on the operation panel.

Once the machining file is loaded, it is in use, which means other programme can't modify the file (but it could be read), so, if you want to modify the file, the file must be closed in Ncstudio™ system.
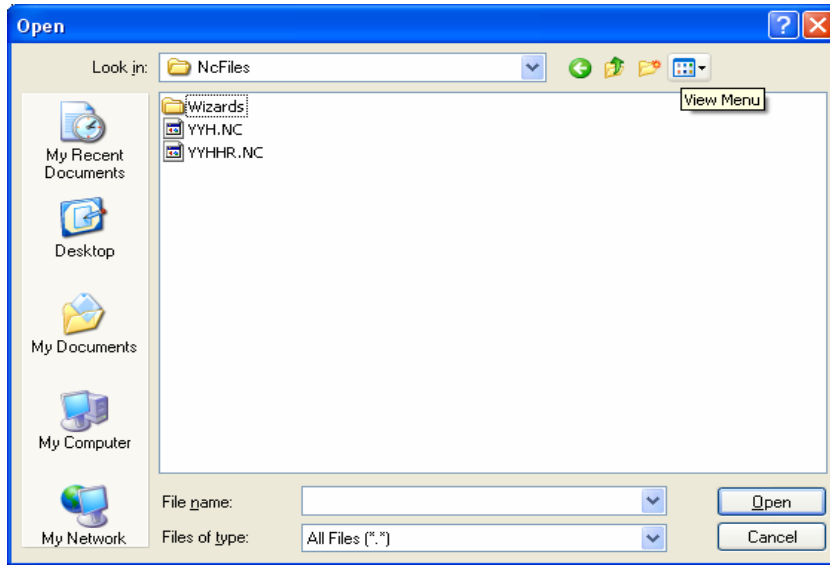
Fig. 10-20 [Open] dialogue

◆ **[Open Text File]**

Shortcut key: Ctri+T, the main function of this menu is to open all the format files supported by system, for instance, G code files (.txt, .g, .nc format). If unsupported files are opened, the warning box "fail to open the file" will pop up.

◆ **[Import G Code File]**

Shortcut key: Ctrl+G, the function of this menu is to add machining object of G code file on the basis of an opened file. Method is as following:

➢ Open or draw a machining file;

➢ Select [Import G code file] under [File] menu, [Opening file dialogue] will pop up as Fig. 10-21.

The method of importing DXF file, ENG file and PLT file in NcEditor is identical as above.
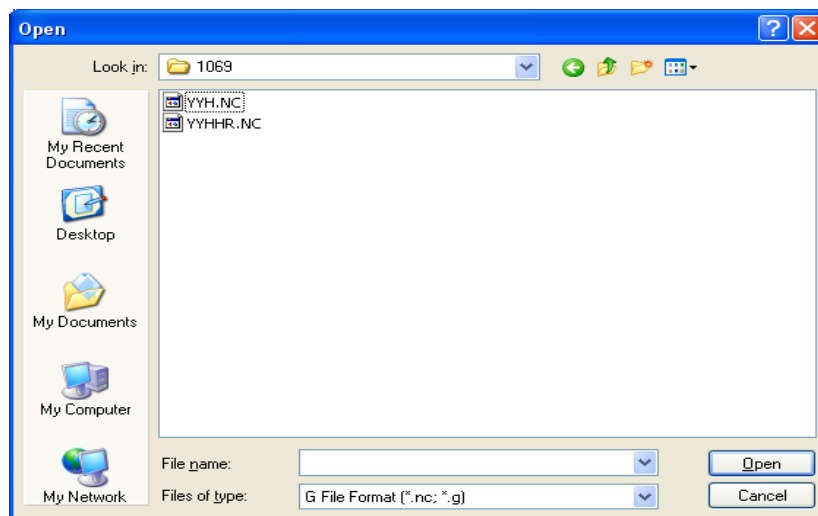
Fig. 10-21 [Open G code format files] dialogue

◆ **[Save]**

Shortcut key: Ctrl+S, the main function is to save the current editing file.

◆ **[Save as]**

Shortcut key: Ctrl+F+A, the function is using another file name to save the current editing file.

◆ **[Save Selected Objects as]**

This function is used in the file with multiple machining objects. You can select parts of objects to save them as a single file and process it.

For instance: open a file, select some figures, as following Fig.10-22.
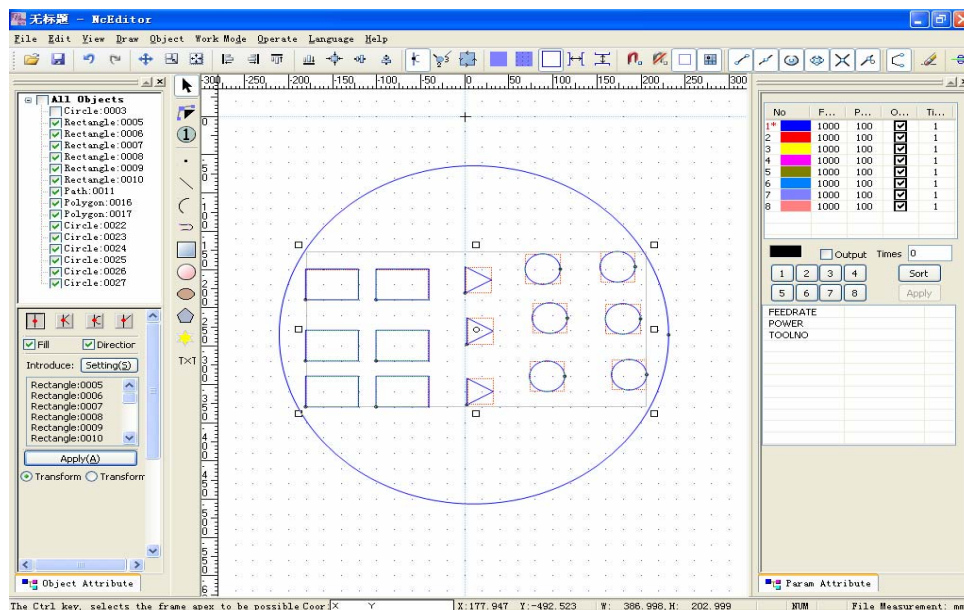
Fig. 10-22 Selection of figure dialogue

Choose the option of [save selected objects as] under [File] menu to eject a dialogue, input the file name and press "yes"; the selected objects will be saved as a single file.

◆ **[Recent Loaded NCE Files]**

This function allows you to view recent loaded NCE files and open these files.

◆ **[Recent Loaded Text Files]**

This function allows you to view recent loaded text files and open these files.

◆ **[Exit]**

Shortcut key: Alt+F4, it is used to exit from the software.

10.7.2 Edit menu

The submenu of edit menu is as following, Fig.10-23
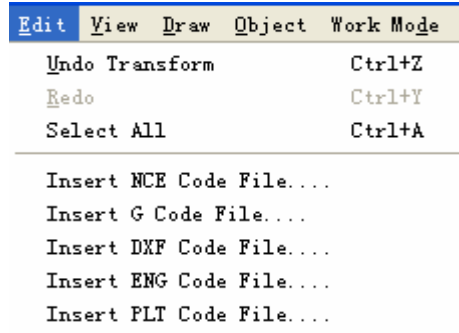
Fig. 10-23 Edit menu picture

◆ **[Undo Transform/ Redo]**

Saving all the operation histories, the software is in possession of strong functions of undoing and redoing. You can undo and redo former manipulations continuously until satisfied, avoiding the bothering of mishandling.

You can also fulfill the same function by button [undo] and [redo] on toolbar.

◆ **[Select All]**

This function is used to select all the objects in [object editor space] so as to avoid continuously selecting objects.

◆ **[Insert File]**

Insert machining object of other file on the basis of an opened file. For instance, insert a DXF file.

Methods are as following:

➢ Open or draw a machining file;

➢ Select [Insert DXF code file] under [Edit] menu, [Opening] dialogue will pop up as Fig.10-24.
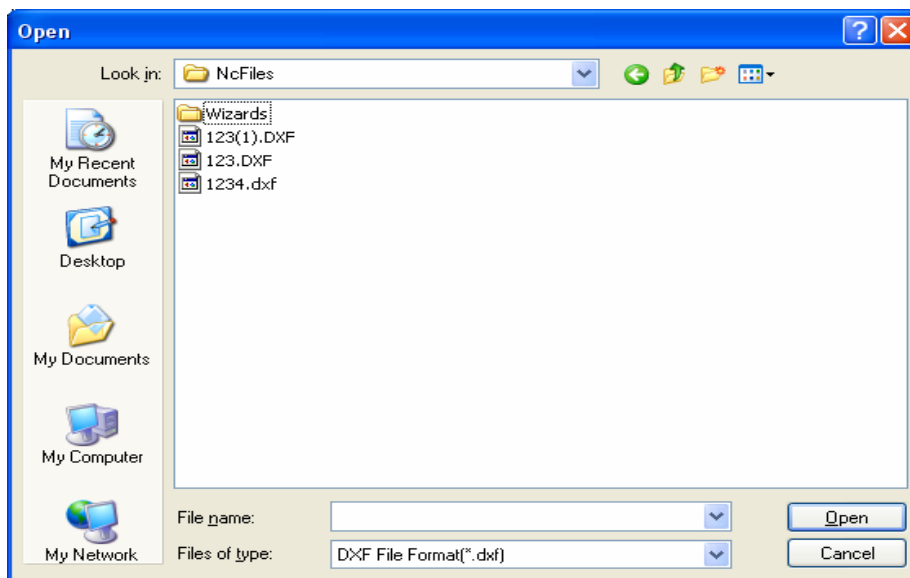
Fig. 10-24 Open file dialogue

The method of inserting G code file, NCE file, ENG file is the same as above.

10.7.3 View menu

Main options of view menu are: toolbar, status bar, object list, draw toolbar, parameters bar, view translate, view zoom, catching option, etc, as Fig.10-25.
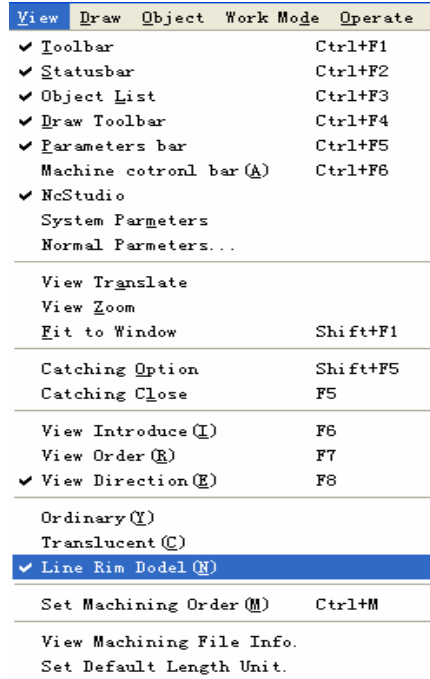
Fig. 10-25 [View] menu picture

◆ **[Toolbar]**

This function is used to display/hide toolbar.

◆ **[Status Bar]**

This function is used to display/hide status bar.

◆ **[Object List]**

This function is used to display/hide object list and object attribute window.

◆ **[Draw Toolbar]**

This function is used to display/hide draw toolbar.

◆ **[Parameters Bar]**

This function is used to display/hide parameters bar.

◆ **[Machine Control Bar]**

This function is used to display/hide the bottom machine control bar.

◆ **[System Parameters]**

This function is used to open system parameters.

◆ **[Normal Parameters]**

This function is used to open normal parameters.

◆ **[View Translate]**

Note: you can reposition a figure to clearly view other parts of the figure. This command will not change the position or ratio of the figure in the coordinate, only the view. You can translate the figure up and down, left and right.

➢ Method one: open [view] menu, select the option [view translate];

➢ Method two: select the translate button ✛ on toolbar.

Then press down the left mouse button in the [Object editor space] and drag the object to the aimed position and release the mouse.

◆ **[View Zoom]**

Note: you can zoom in and zoom out the track by "menu + mouse" or "toolbar + mouse", the actual size of the figure won't change, and only the view will be enlarged or minified.

➢ Method one: open [view] menu, select the option [view zoom];

➢ Method two: select the [view zoom] button 🔲 on toolbar.

Then you should slide the mouse wheel in the [object editor space]. Sliding forward means "zoom in", while sliding backward means "zoom out".

◆ **[Fit to Window]**

Note: after zooming and translating, if you want to fit the view to window, this menu item can be used. This function can make you get a general view without sliding the wheel, since this function can make track view fit to window.

➢ Method one: open [view] menu, select the option [Fit to window];

➢ Method two: select the button 🔲 on toolbar.

◆ **[Catching Option]**

[Catching option] is used to better position to the feature points of some figures during drawing objects, so you can easily catch the objects when a feature point is approaching.

Select [View] menu → select [Catching option] or press the icon 🔲 (🔲 catching off) on toolbar, the [catch option] dialogue will pop up.
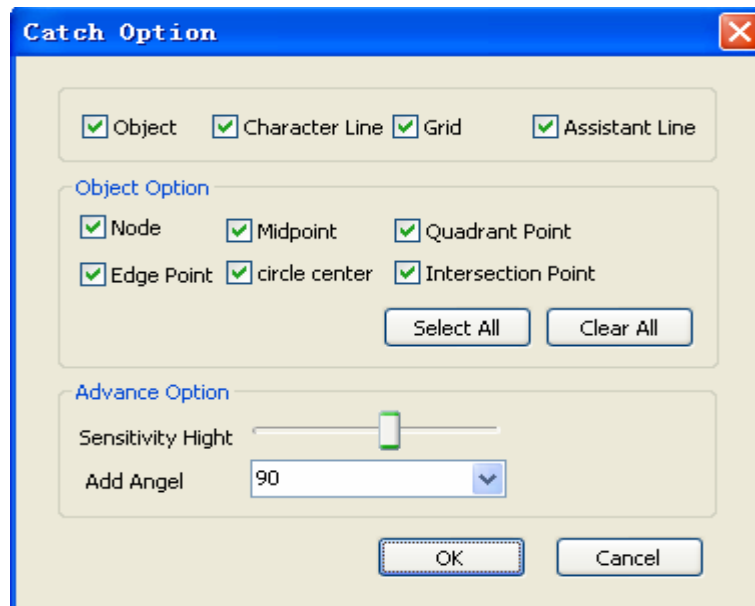
Fig. 10-26 [Catch option] dialogue

If a check box is selected, the corresponding object will appear in [Object editor space], or it will be concealed.

◆ **The character point of each figure:**

➢ The character point of spot: point;

➢ The character point of straight line: two endpoints and one midpoint;

➢ The character point of polyline: every node and circle center;

➢ The character point of rectangle: four vertexes;

➢ The character point of circle: four quadrant points and circle center;

➢ The character point of ellipse: four quadrant points and circle center;

➢ The character point of arc: start point, end point, midpoint and circle center;

◆ **The advantage of [Catching option]**

You can directly use cursor to position to the aimed point precisely (such as the above character point in figure) without knowing the concrete coordinates, which provides convenience for accurate figure connection. This function will be further used in following [Combine object].

For instance: how to introduce another straight line from the midpoint of a straight line.

➢ Select [Catching Option] under [View] menu, or choose the icon on toolbar.

➢ Drawing a straight line in [Object editor space], a little pane, the caught character point (midpoint of straight line), will appear when the cursor is on the straight line near the midpoint, so you can find the midpoint accurately, click the pane to introduce another straight line from the midpoint.

◆ **[Catch grid]**

The advantage of [Catch grid]:

[Catch grid] can execute integer positioning on the objects, which can make the drawn points in grids of [Object editor space] automatically. The default unit of grid in NcEditor is "1", and there will be catching grids if the integer crossing is encountered on the direction of X, Y, which is more precise than other grid positioning software. The coordinate of grid point will display in the status bar precisely.

Manipulation method:

Select the option [Grid] of [Catching Option] under [View] menu, or choose the icon 🧲 (🧲 catching off) on toolbar.

> Note: [Catching grid] is only available under the condition that drawing function is activated (one option is selected under draw menu).

◆ **[Sensitivity Height], [Add Angel], [Character Line] and [Assistant Line]**

[Sensitivity Height] is used to specify the reflection intensity of catching when the cursor is approaching the character point.

[Character Line] is used to prompt the related position for you.

[Add Angle] is to catch an object with a certain angle. For instance, choose 45, in drawing a straight line, this line will be caught when the included angle between the line and horizontal direction is 45° after the first point is drawn.

[Assistant line] is used to assist you in limiting the position of the drawn object, referring to the straight line from ruler to [object editor space].

◆ **[View Introduce]**

This function is used to display / hide the lead-in line.

You can decide whether to add lead-in line or not as well as the type of lead-in line according to the cutting method. Please refer to [setting lead-in line] function for detail.

◆ **[View Order]**

This function is used to display / hide the machining order of current objects.

◆ **[View Direction]**

This function is used to display / hide machining direction.

◆ **[Drawing mode]**

In order to provide you with more convenience in drawing, this software sets three view modes, including ordinary, translucent and line rim mode.

1) **[Ordinary view mode]**

It is used frequently. In this mode, when there are many layers, the overlapping parts will be concealed, as Fig. 10-27.
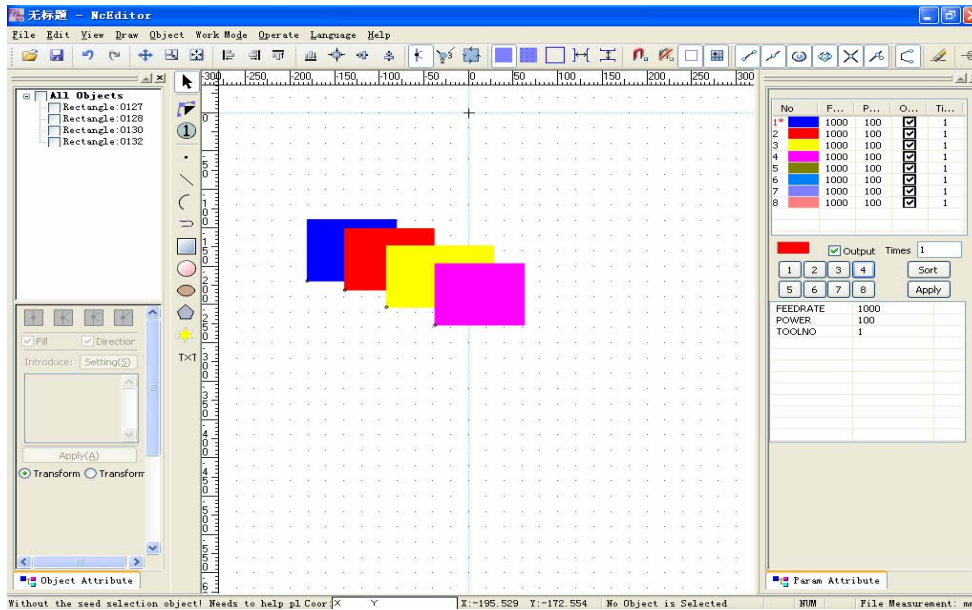
Fig. 10-27 Ordinary view mode

2)   [Translucent view mode]

In order to make drawing more convenient, NcEditor can make layers translucent. That means the backward grid background or overlapping parts will be revealed translucently, as Fig.10-28.
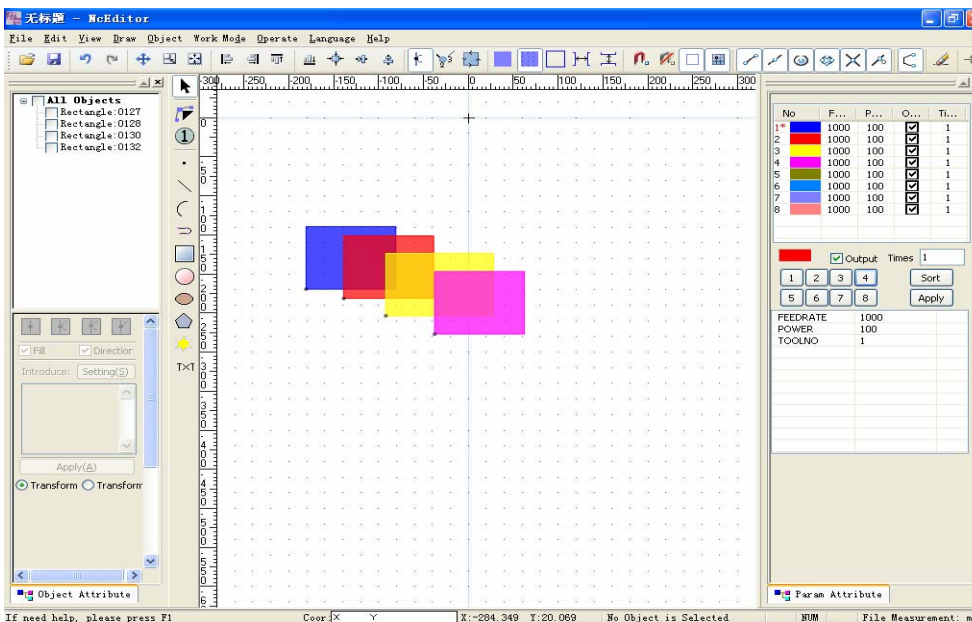
Fig. 10-28 Translucent view mode

3)   [Line rim mode]

This is the default view mode, all the figures are shown in wireframe mode, the overlapping parts will be completely revealed, as Fig.10-29.
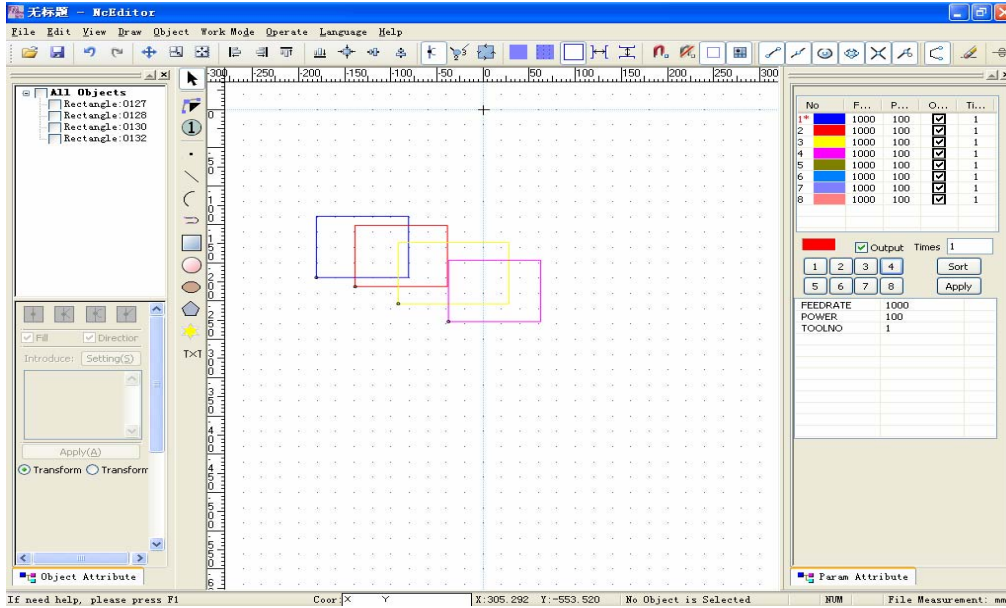
Fig. 10-29 Wire-frame view mode

◆ **[View Machining File Info.]**

The file information can be checked after machining. After clicking the [View Machining File Info.] option, you can know the information of file name, cutting time, dry running time, delay time and cutting length.

After machining is finished, click the option [View Machining File Info.] to eject the following dialogue, as Fig.10-30.
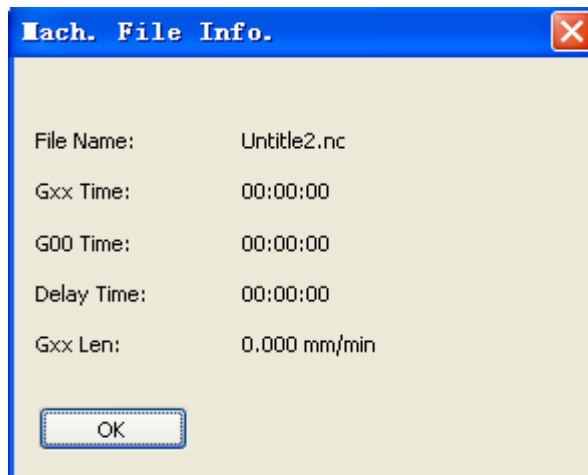
Fig. 10-30 Machining file information

Information will be shown after machining.

◆ **[Set Default Length Unit]**

There are two kinds of length units in NcEditor, metric and imperial, which is for different measurement units of different nations. And you can choose the suitable one according to your needs.

Select [Setting default length unit] option under [View], the following dialogue will pop up.
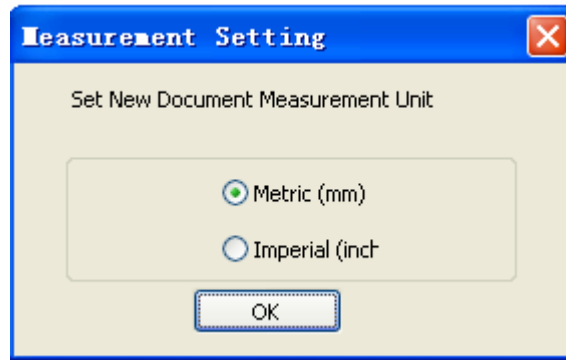
Fig. 10-31 Setting length unit dialogue

10.7.4 Draw menu

Draw menu is used to draw the ordinary figures, including point, straight line, polyline, rectangle, circle, ellipse, arc, and so on. All the menu items are corresponding to the buttons on the drawing toolbar, so you can also execute manipulation by buttons on toolbar. See Fig. 10-32.
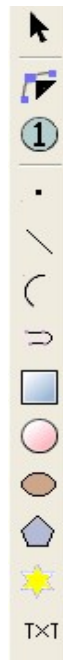
Fig. 10-32 Drawing toolbar

Note: the following drawn figures can be accurately positioned through object properties window. The drawing will be finished by right click.

◆ **[Draw Point]**

The simplest drawing in [Object editor space] is drawing a point.

Select the option [Draw Point] under [Draw] menu or press the button ⋅ on toolbar→ click the left mouse button in the selected place of [Object editor space] to draw a point.

If you need to draw a point more precisely, you can input the coordinates in [Object attribute window], and then press [Enter] or [Apply].

◆ **[Draw Line]**

If you want to draw a straight line, you can select the option [Draw Line] under [Draw] or single-click icon ＼ .

Two points can determine one straight line. Draw two points in [object editor space] as the beginning and end points of straight line.

After selecting the object, you can input the beginning and end points in [Object attribute window] to draw the straight line accurately.

◆ **[Draw Polyline]**

Polyline refers to the curve connected end to end by different straight lines and arcs, which enables you to reduce operation times and improve the velocity of modeling via finishing manipulation of multiple arcs and segments once.

If you want to draw a polyline, you can select [Draw polyline] under [Draw] or single-click icon ⇒ .

Shortcut key: arc→ A; line→ L; close the figure→ C; finishing drawing→ F; Esc→ cancel drawing

▌ Note: in drawing the polyline, the input state should be English.

◆ **[Draw Rectangle]**

If you want to draw a rectangle, you can select [Draw rectangle] under [Draw] or single-click icon ▢ .

NcEditor determines the position and shape of a rectangle by the left bottom vertex coordinates, width and height.

After selecting the figure, you can accurately draw a rectangle by inputting the left bottom vertex coordinates, height and width in [Object attribute window].

◆ **[Draw Circle]**

If you want to draw a circle, you can select [Draw circle] under [Draw] or single-click icon ◯ .

NcEditor ensures the position and size of a circle by radius and circle center.

After selecting the figure, you can accurately draw a circle by inputting circle center coordinates and radius in [Object attribute window].

◆ **[Draw Ellipse]**

If you want to draw an ellipse, you can select [Draw ellipse] under [Draw] or single-click icon ⬭ .

NcEditor ensures the position and size of an ellipse by circle center and radius of X direction and

Y direction.

After selecting the figure, you can accurately draw an ellipse by inputting circle center coordinates and radius in [Object attribute window].

◆ **[Draw Polygon]**

If you want to draw a polygon, you can select [Draw polygon] under [Draw] or single-click icon .

NcEditor ensures the position and size of a polygon by radius, center X and center Y, edge. After selecting the figure, you can accurately draw a polygon by inputting the related value in [Object attribute window].

◆ **[Draw Star]**

If you want to draw a star, you can select [Draw star] under [Draw] or single-click icon .

NcEditor ensures the position and size of a star by radius, center X and center Y, angles.

After selecting the figure, you can accurately draw a star by inputting the related value in [Object attribute window].

◆ **[Draw Arc]**

If you want to draw an arc, you can select [Draw arc] under [Draw] or single-click icon .

NcEditor ensures the position and size of an arc by radius, arc center, beginning angle and end angle.

After selecting the figure, you can accurately draw an arc by inputting circle center coordinates and required radius, beginning angle, and end angle in [Object attribute window].

Note: 1 rad= 180/π degree

◆ **[Draw Text]**

If you want to write a section of text, you can select [Draw text] under [Draw] or single-click icon .

Left click the mouse in "Object editor space". "TEXT" characters will be shown as Fig. 10-33
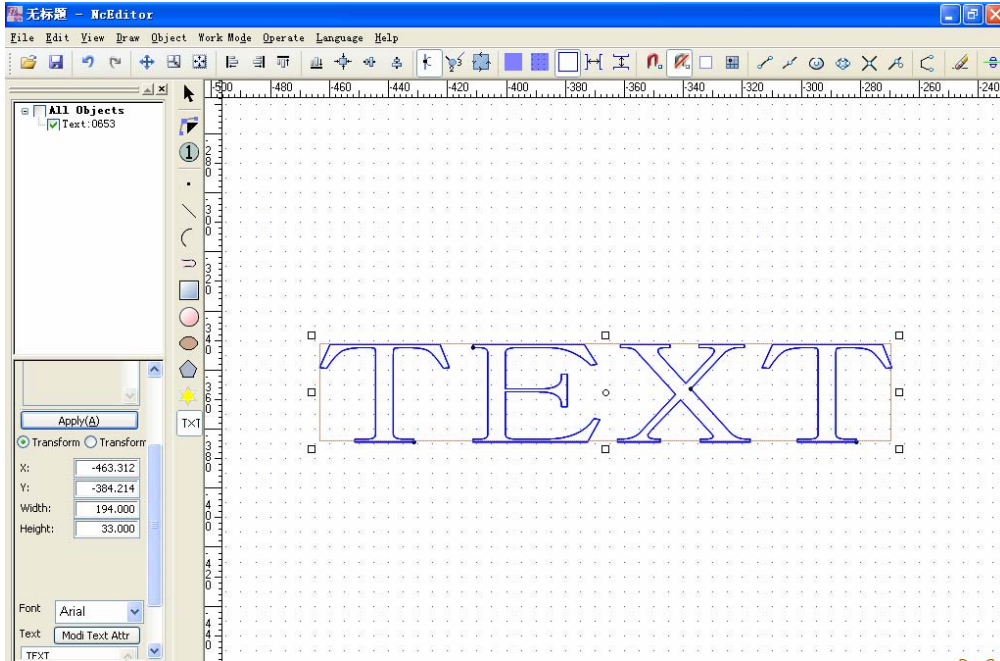
Fig. 10-33 Text function

When "TEXT" appears in "Object editor space", the object attribute window on the left will show the text modification column simultaneously, as Fig. 10-34.
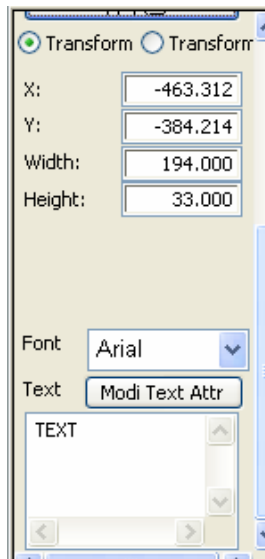
Fig. 10-34 Text modification column

You can substitute the desired characters for "TEXT". After setting the font and size, click "Apply" to finish the operation.

In addition, relative to drawing toolbar, draw menu adds [Measure distance] and [Set Origin]. Draw menu is as following:
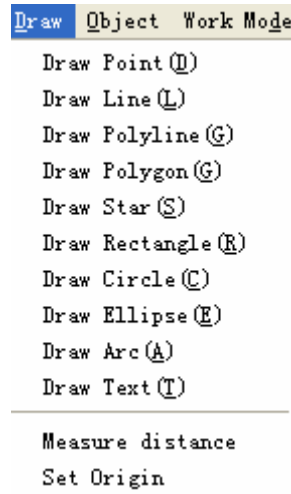
Fig. 10-35 Draw menu

◆ **[Measure distance]**

This item is used to measure certain specific distance, which is mainly used in dispenser software instead of waterjet cutting.

◆ **[Set Origin]**

If you need to set workpiece origin, you can select this item. After you select it, a dialog will pop up, as following.
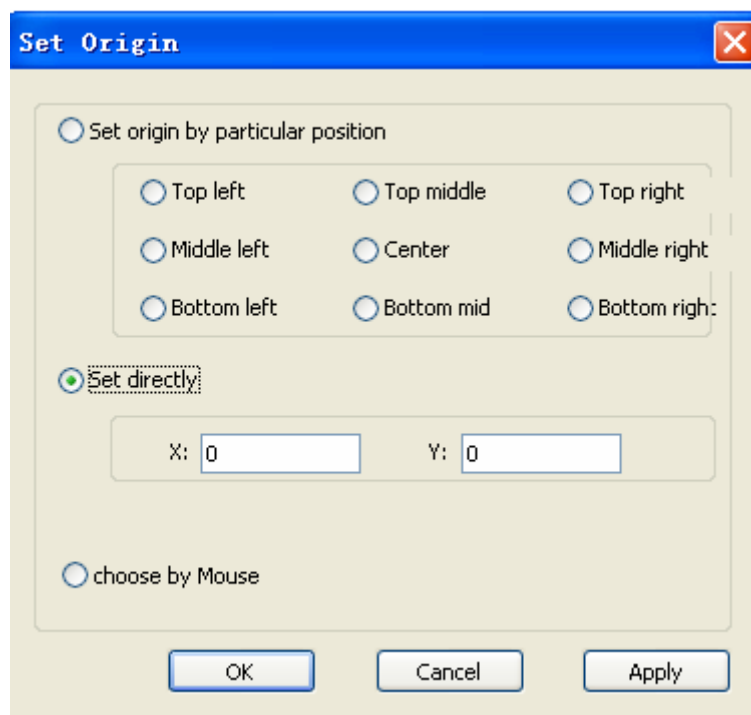
Fig. 10-36 Set workpiece origin dialog

You can set the workpiece origin by feature points or mouse, or directly input the X and Y coordinates of the origin.

10.7.5 Object menu

◆ **[Delete Object]**

The object should be selected before deleted.

Following three methods are used to delete objects:

➢ After selection of an object, you can right-click to eject a menu and choose [Delete Object];

➢ Select [Delete Object] under [Object];

➢ Use the button [Delete] on keyboard.

You can select all the objects by shortcut key: "Ctrl+A" and press [Delete] button to delete all of them.

◆ **[Combine object]**

> Note: the points of unclosed objects don't connect together because of misoperation or other reasons. When the objects need integrated operation, you can use [Object combination] to combine several unclosed objects (straight line, arc and unclosed polyline) with nearby endpoint (<0.1) to a single path object (polyline or path). It is suggested to use [catching option] before using [object combination].

For instance, combine two straight lines.

First, draw the first line "0001". And then, you can open [Catching option] to precisely position the endpoint of first line when drawing the second line "0002". Then you can select option [Combine object] under [Object] or right-click the menu to choose [Combine object].See Fig.10-37.
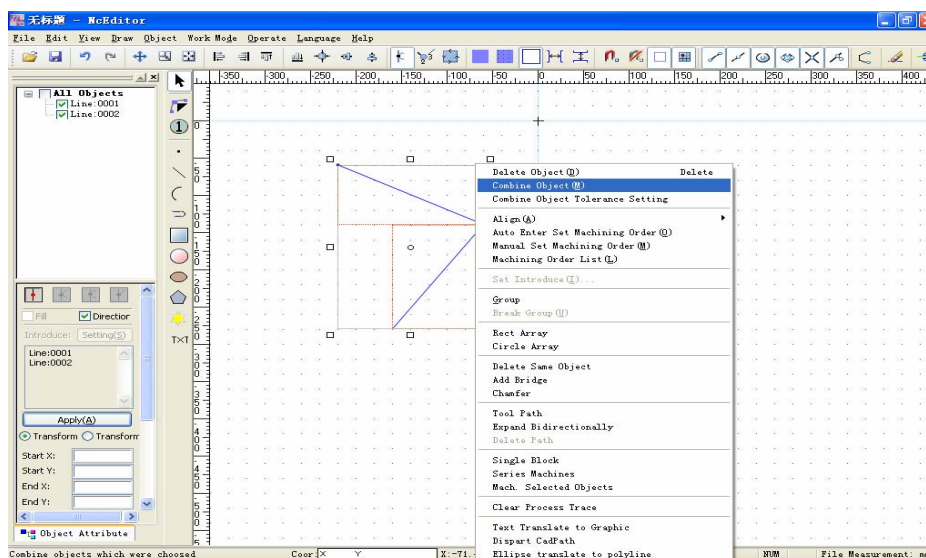
Fig. 10-37 Manipulation of combination of two straight lines

The combination output of two straight lines is as Fig. 10-38, and the two objects are combined to one single object (polyline).
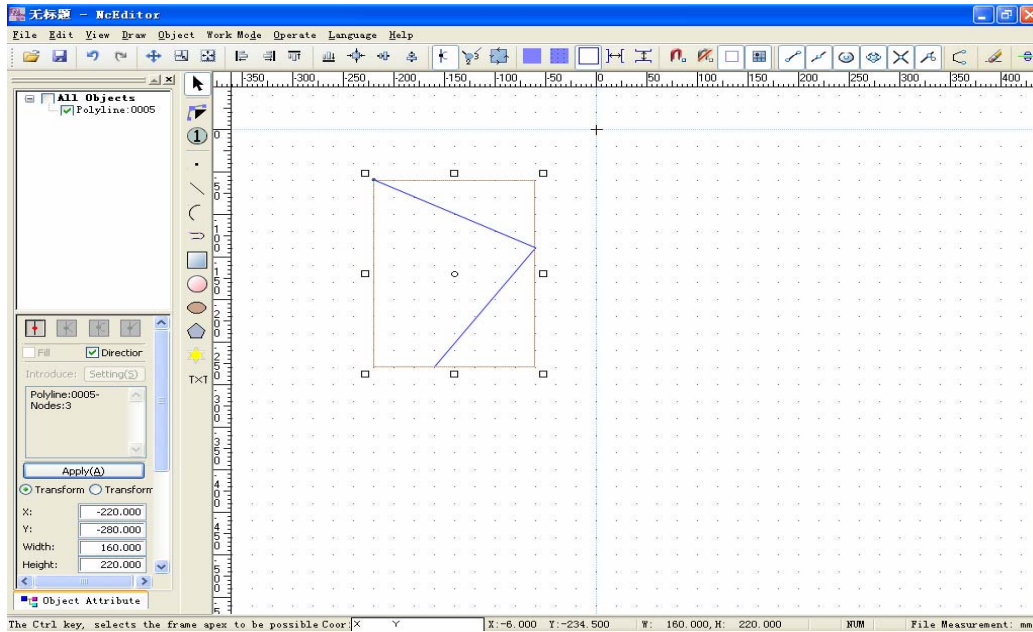
Fig. 10-38 Output of the combination of two straight lines

For combination of straight lines, arcs and unclosed polylines, you can select the [Catching option] function to catch the endpoints of these figures in drawing to improve the effectiveness of [Combine object].

Note: in combination of two or multiple arcs, the process will begin from the origin object counter-clockwise.

◆ **[Combine Object Tolerance Setting]**

In combination, you need to combine objects following certain criteria, which require setting combine tolerance (range of combination distance between objects), so the function of setting combine tolerance is offered.

The following dialogue will pop up in selecting [Combine object tolerance setting]. You only need to input a number between 0.1-5 into the input dialogue.
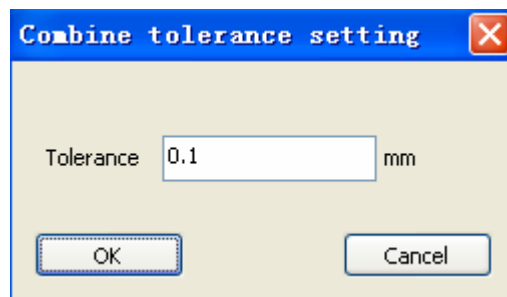
Fig. 10-39 Setting combine tolerance

◆ **[Align]**

When you choose above two objects in [Object editor space], the [Align] is enabled. This function can align all the selected objects in two-dimensional plane.

Note: all the following alignment functions are on the basis of top, bottom, left and right frame and center point of the rectangle after the selection of figures.

See Fig. 10-40, all the following alignment functions will be exemplified through the two below objects.
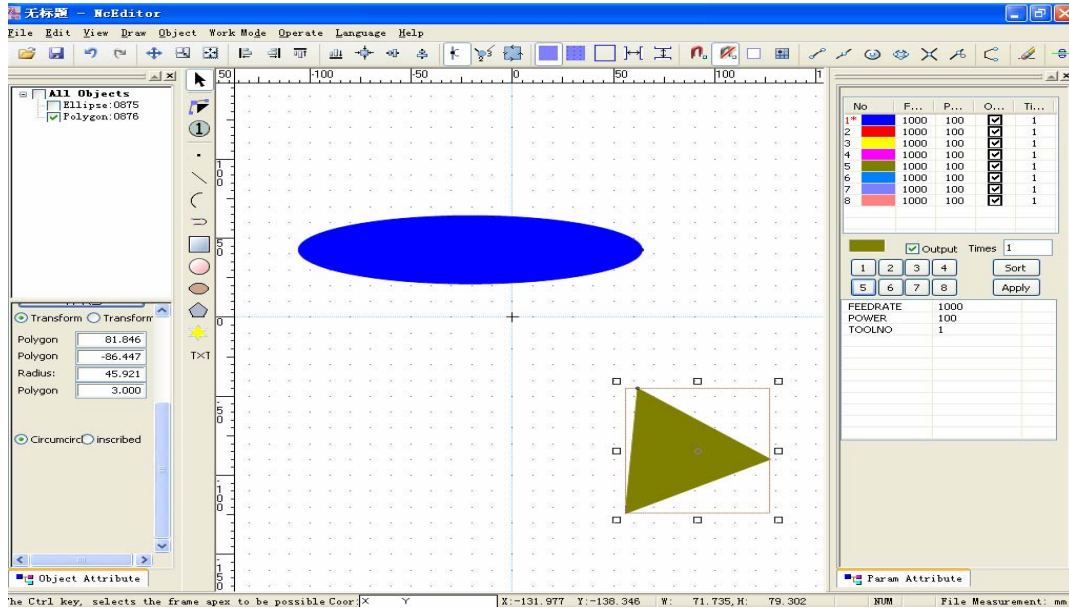
Fig. 10-40 Two objects used for align

◆ **[Align Left]**

The left edge of rectangle box of the selected figure is regarded as the remark for alignment. After you select the objects, there are three methods to fulfill [Align Left] function:

➢ Right-click the object and select the option of [Align Left] under the menu;

➢ Select option [Align left] under [Object] menu;

➢ Press the [Align Left] button on the toolbar.

After [Align Left] is executed, the view is as following:
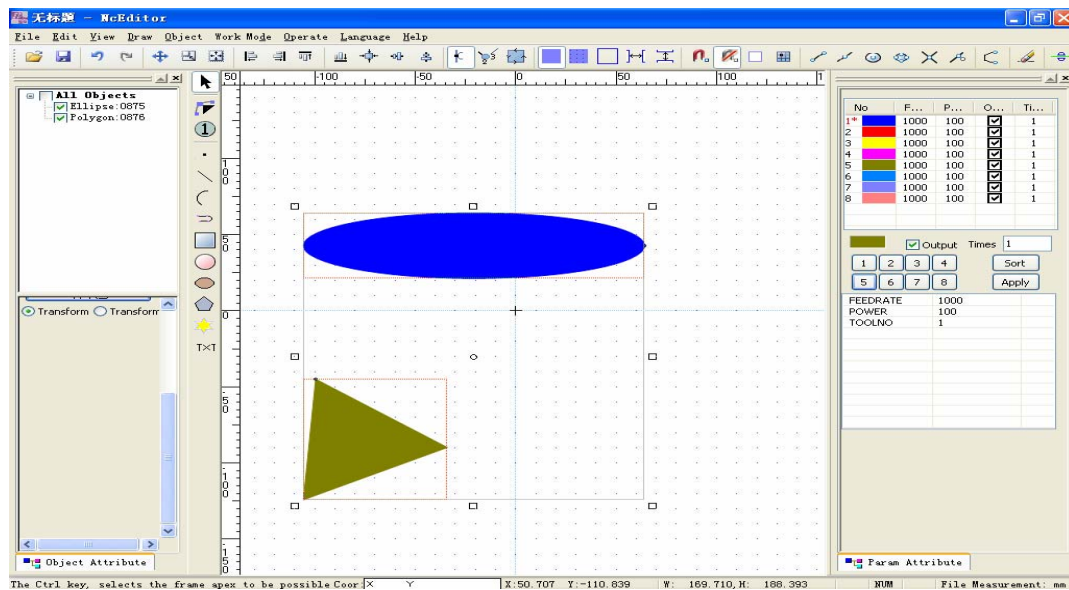
Fig. 10-41 [Align Left] operation

◆ **[Align Right]**

The right edge of rectangle box of the selected figure is regarded as the remark for alignment.

After the objects are selected, there are three methods to fulfill [Align Right] function:

➢ Right-click the object and select the option of [Align Right] under the menu;

➢ Select option [Align Right] under [Object] menu;

➢ Press the [Align Right] button ⌐ on the toolbar.

After [Align Right] is executed, the view is as following:
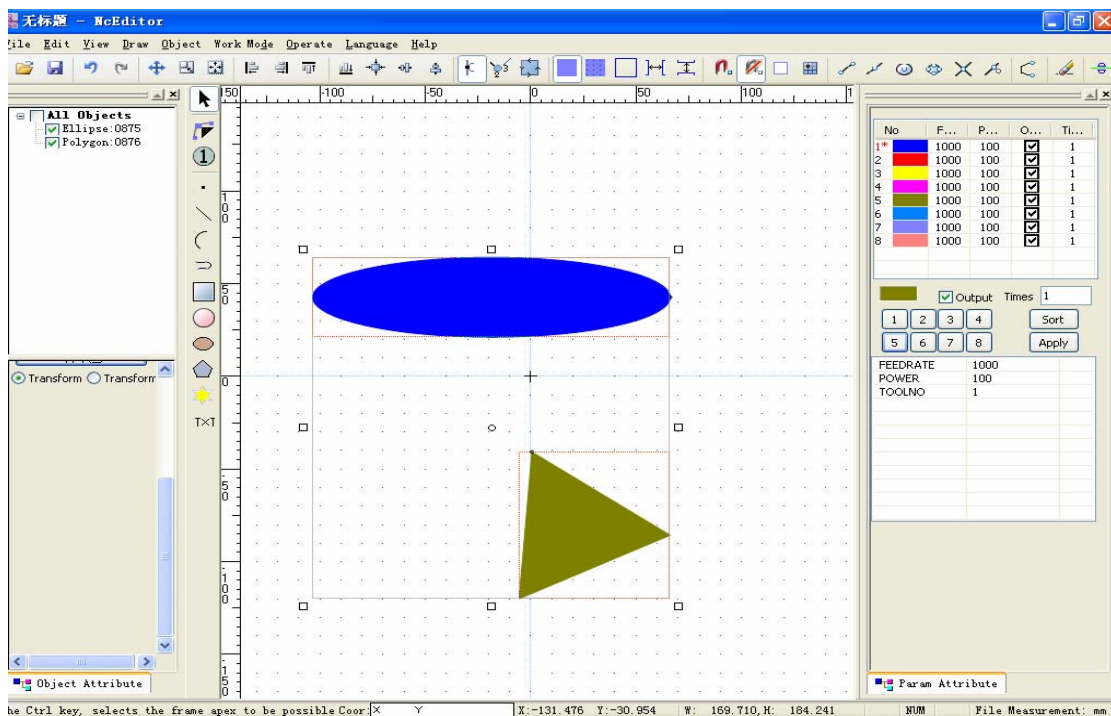
Fig. 10-42 [Align Right] operation

◆ **[Align Vertical Line]**

The X-axis coordinate of center point of rectangle box is regarded as the datum for those of all the selected objects.

After the objects are selected, there are three methods to fulfill [Align Vertical Line] function:

➢ Right-click the objects and select the option of [Align Vertical Line] under the menu;

➢ Select option [Align Vertical Line] under [Object] menu;

➢ Press the [Align Vertical Line] button ⊹ on the toolbar.

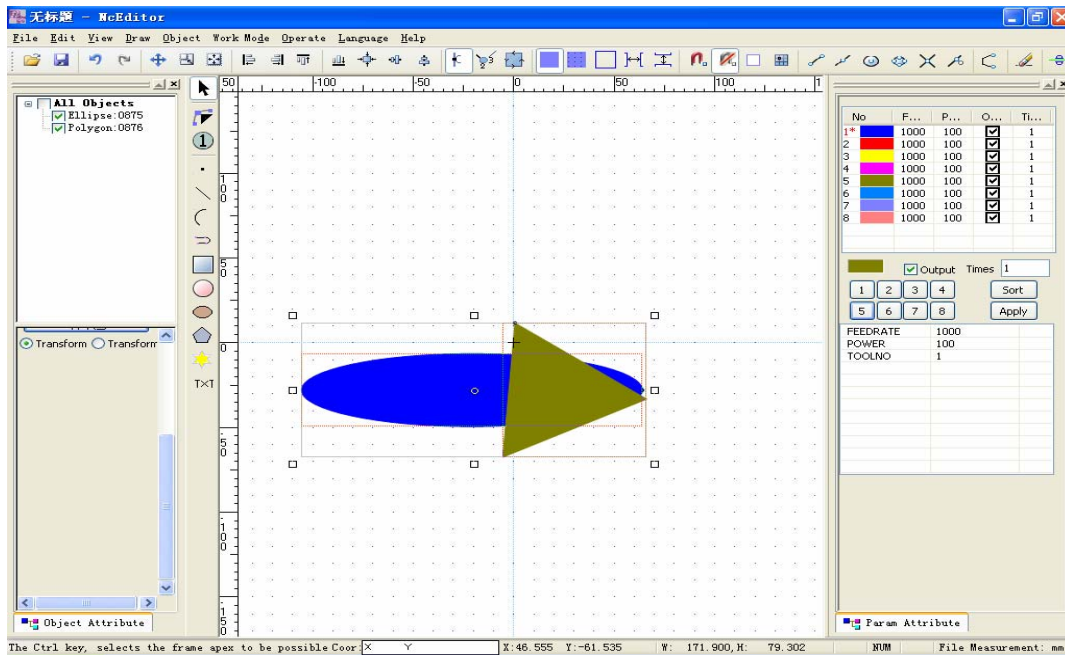After [Align Vertical Line] is executed, the view is as following:
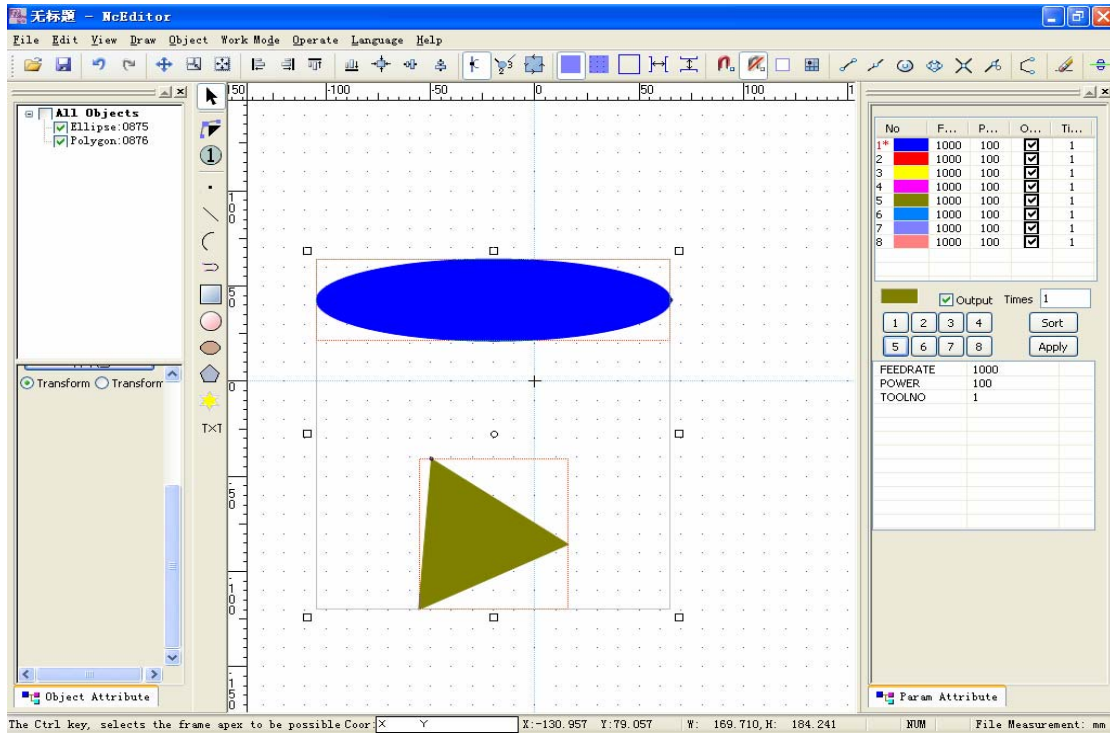
Fig. 10-43 [Align Vertical Line] operation

Note: the objects only move in horizontal direction in [Align Left], [Align Right], [Align Vertical Line].

◆ **[Align Top]**

The top edge of rectangle box of the selected figure is regarded as the datum for alignment.

After the objects are selected, there are three methods to fulfill [Align Top] function:

➢ Right-click the objects and select the option of [Align Top] under the menu;

➢ Select option [Align Top] under [Object] menu;

➢ Press the [Align Top] button ⬚ on the toolbar.

After [Align Top] is executed, the view is as Fig. 10-44.
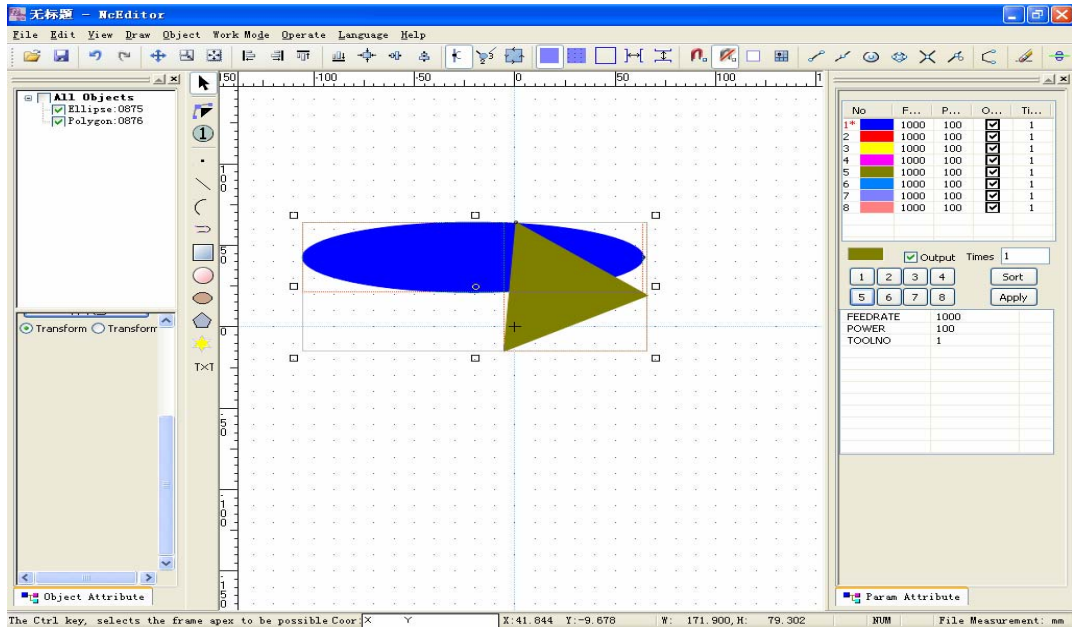
Fig. 10-44 [Align Top] operation

◆ **[Align Bottom]**

The bottom edge of rectangle box of the selected figure is regarded as the datum for alignment.

After the objects are selected, there are three methods to fulfill [Align Bottom] function:

➢ Right-click the objects and select the option of [Align Bottom] under the menu;

➢ Select option [Align Bottom] under [Object] menu;

➢ Press the [Align Bottom] button on the toolbar.

After [Align Bottom] is selected, the view is as following, Fig. 10-45.
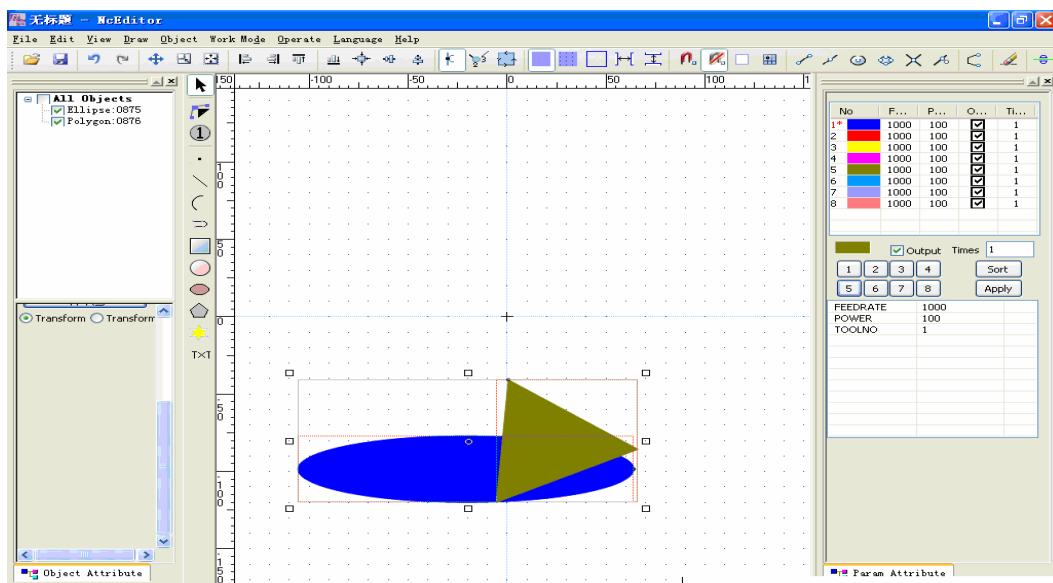
Fig. 10-45 [Align Bottom] operation

◆ **[Align Level Line]**

The Y-axis coordinate of center point of rectangle box is regarded as the datum for the ones of all the selected objects.

After the objects are selected, there are three methods to fulfill [Align Level Line] function:

➢ Right-click the objects and select the option of [Align Level Line] under the menu;

➢ Select option [Align Level Line] under [Object] menu;

➢ Press the [Align Level Line] button on the toolbar.

After [Align Level Line] is executed, the view is as following, Fig. 10-46.

Fig. 10-46 [Align Level Line] operation

Note: the objects only move in vertical direction in [Align Top], [Align Bottom], [Align level line].

◆ **[Align Center Point]**

The center point of rectangle box of the selected figure is regarded as the datum for alignment.

After the objects are selected, there are three methods to fulfill [Align Center Point] function:

➢ Right-click the objects and select the option of [Align Center Point] under the menu;

➢ Select option [Align Center Point] under [Object] menu;

➢ Press the [Align Center Point] button on the toolbar.

After [Align Center Point] is executed, the view is as following, Fig. 10-47.
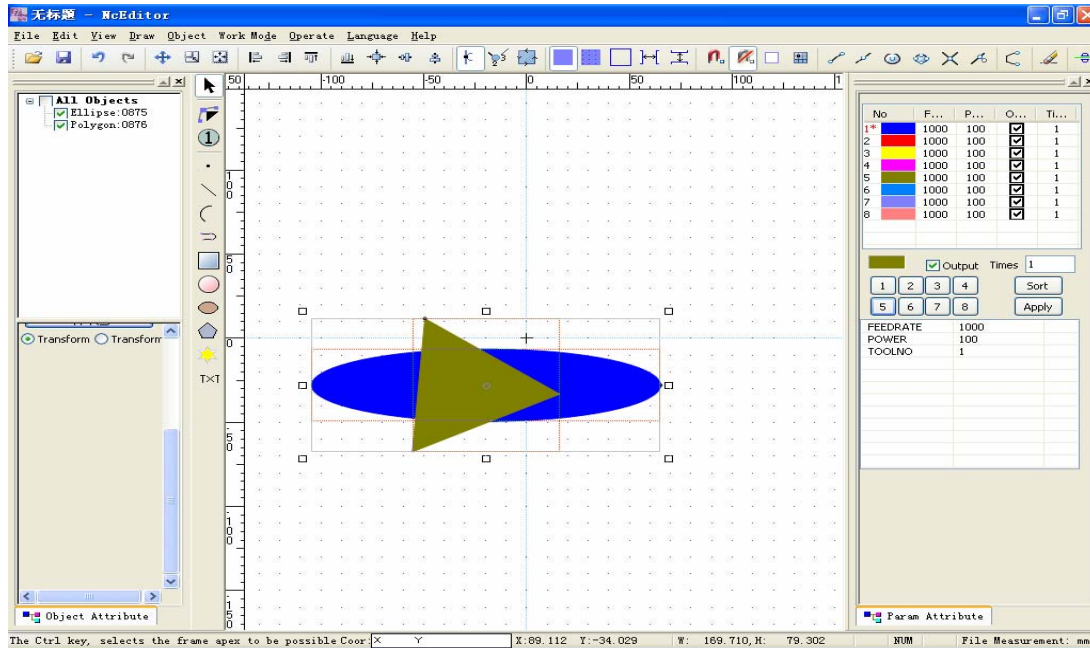
Fig. 10-47 [Align Center Point] operation

Note: the objects move in both vertical direction and horizontal direction by [Align Center Point].

◆ **[Align Level Dispersion]**

There are two alignment criteria for this method:

➢ Center point distance;

➢ Graphic distance standard;

In [Center point distance], the horizontal distances between center points of figures are equal, and the positions of two objects next to the horizontal edge are fixed. In [Graphic distance standard], the horizontal blank spaces between each figure are equal.

The selected objects must be three or above three, and there are three methods to fulfill [Align Level Dispersion].

➢ Select option [Align Level Dispersion] under [Object] menu;

➢ Press the icon ⊬⊣ on toolbar;

➢ Right-click the object, and select the menu option [Align Level Dispersion].

After button [Align Level Dispersion] is pressed, the following dialogue will pop up, as Fig. 10-48.
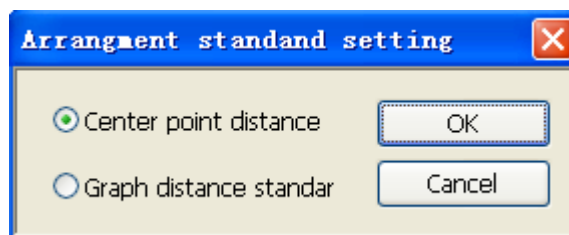
Fig. 10-48 [Arrangement standard setting]

Select [Center point distance] under [Align Level Dispersion], see Fig. 10-49, Fig. 10-50.
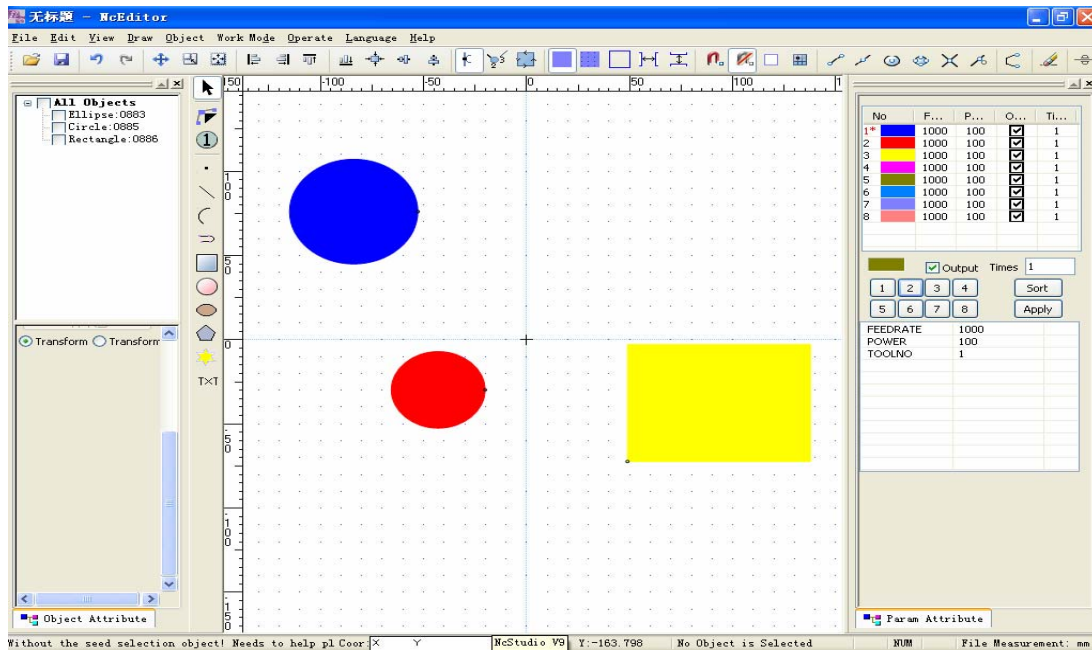

Fig. 10-49 Before [Align Level Dispersion]
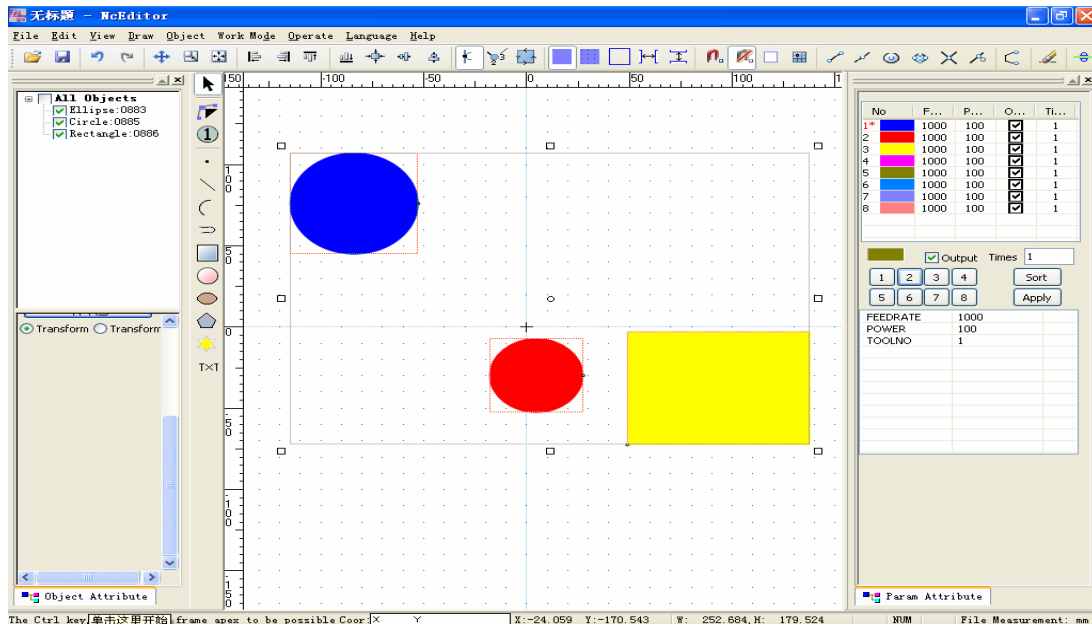
Fig. 10-50 After [Align Level Dispersion]

Note: the selected objects must be three or above three, or, a dialogue will pop up: the selected objects are less than 3, and the operation is cancelled.

◆ **[Align Vertical Dispersion]**

Just like [Align Level Dispersion], there are two criteria for [Align Vertical Dispersion]:

➢ Center point distance;

➢ Graphic distance standard;

In [Center point distance], the vertical distances between center points of figures are equal, and the positions of objects next to the vertical edge are fixed. In [Graphic distance standard], the vertical blank spaces between each figure are equal.

The selected objects must be three or above three, and there are three methods to fulfill [Align Vertical Dispersion].

➢ Select option [Align Vertical Dispersion] under [Object] menu;

➢ Press the icon ⬍ on toolbar;

➢ Right-click the objects, and select the menu option [Align Vertical Dispersion].

After the button [Align Vertical Dispersion] is pressed, a dialogue will pop up, which is the same as that of [Align Level Dispersion].

Select [center point distance] under [Align Vertical Dispersion], the result is as Fig. 10-51, Fig.10-52.
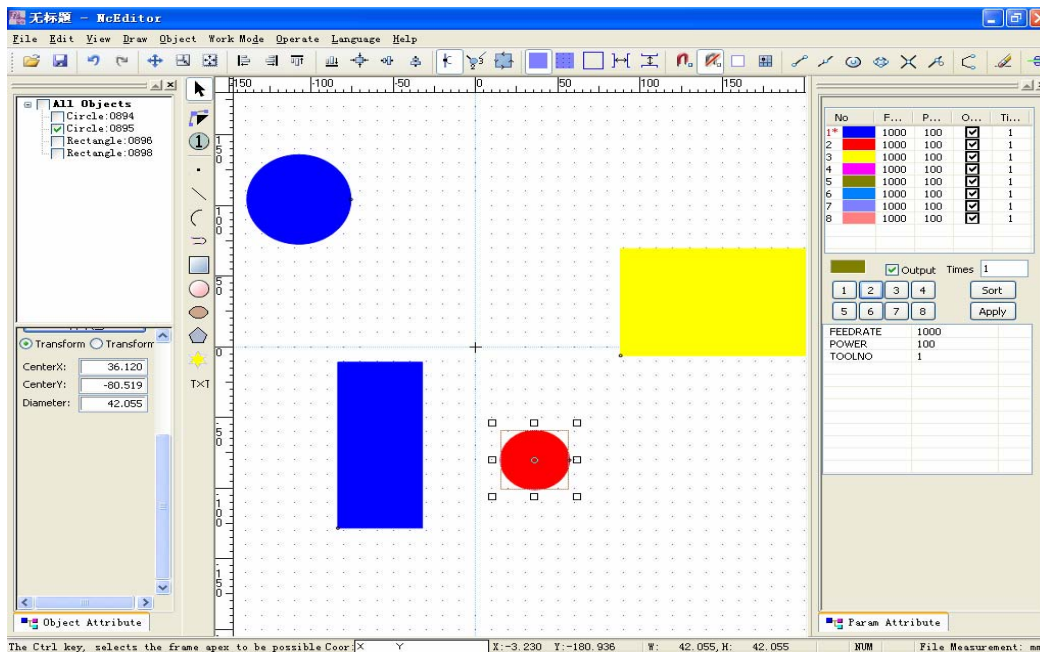
Fig. 10-51 Before [Align Vertical Dispersion]
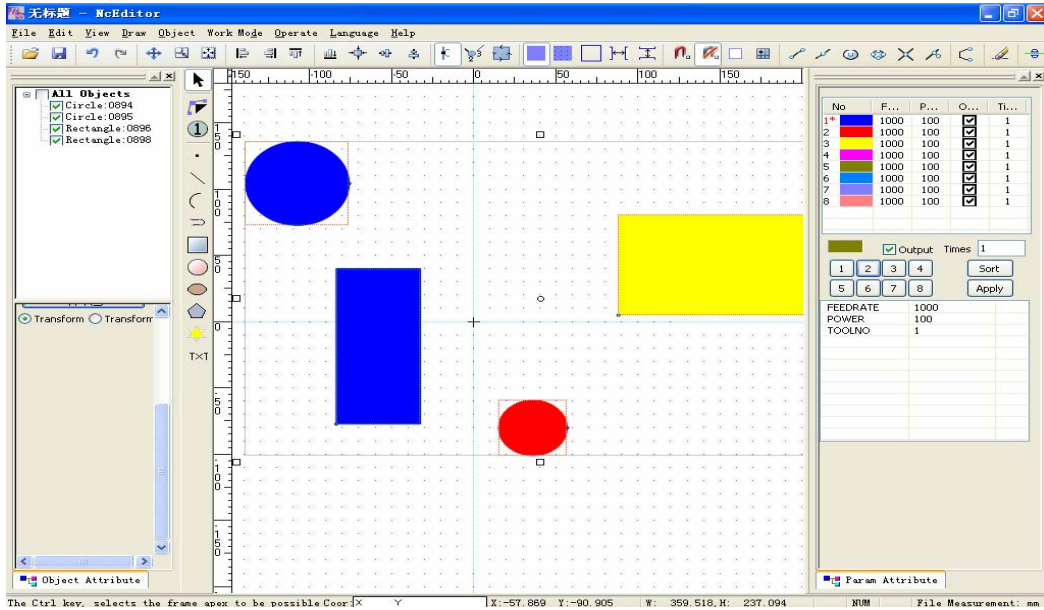
Fig. 10-52 After [Align Vertical Dispersion]

◆ **[Auto Enter Set Machining Order]**

In order to save time and improve efficiency in machining, NcEditor holds the function of [Auto Enter Set Machining Order], and system will automatically set the machining order following the best path algorithm.

Manipulation method:

➢ Draw several different types of figures.

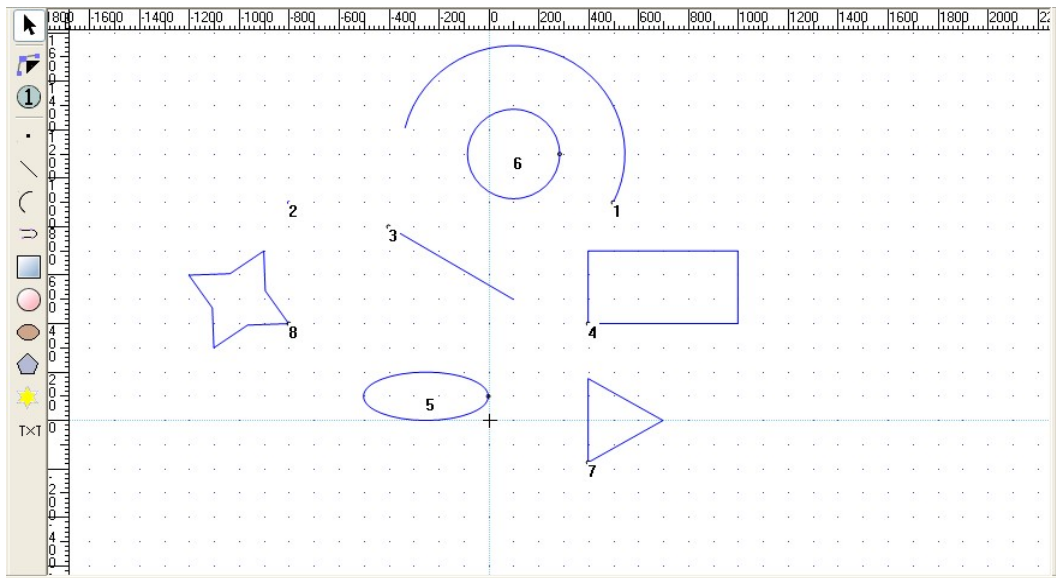➢ Select [View Order] option under [View] or press the icon on toolbar to display the machining order of the figures, see Fig. 10-53.

Fig. 10-53 Displaying the machining order of the objects

➢ Click the option [Auto Enter Set Machining Order] under [Object] or under right-click menu, and system will automatically set the machining order following the best path algorithm, see

Fig. 10-54.
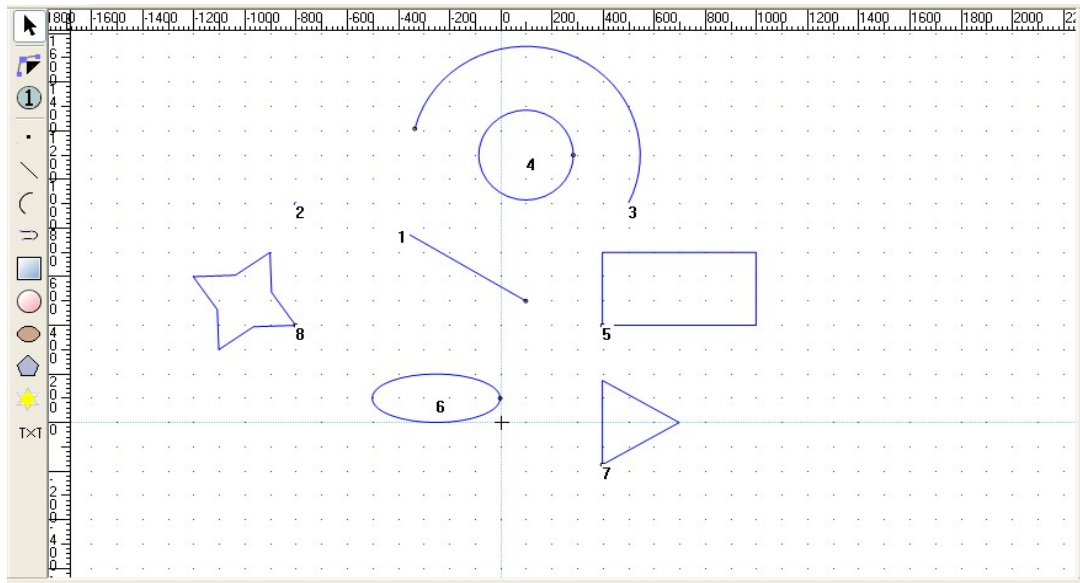
Fig. 10-54 Auto setting machining order

➢ You can also modify machining orders of objects according to your demands:

For instance, click the object "6", the machining order will change to No.1; then click the object "5", the machining order will be No.2; the other objects will be arranged according to the origin ascending sequence.

➢ Save the sequence of several previous objects in [Auto Enter Set Machining Order], and change the sequence of following machining objects. For example, press the button "Ctrl" and then click the frame of object No. 4 to keep the machining order of No.1-4 unchanged; if you want to change the machining order of No.5-8, you can click the required object to mark it as "No.5", then click the rest objects sequentially.

◆ **[[Manual Set Machining Order]**

This function is to make it more convenient for you to set the special machining order.

After [Auto Enter Set Machining Order] is executed, if you want to change the machining order of certain object, the method is as following:

Select the object → choose [Manual Set Machining Order] under [Object] menu or under right-click menu → input the number into the following input frame → press [OK]
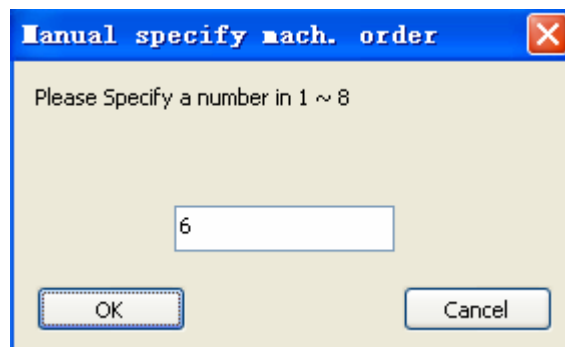

Fig. 10-55 [Manual Set Machining Order] dialogue

In the above figure, please specify a number between 1 and 8, "8" is the total number of objects in [object editor space], which is variable.

[Set Introduce] has been introduced in object attribute window, which will be omitted here.

◆ **[Group]**

If multiple objects，multiple groups of objects or objects and groups are edited simultaneously, they should be set as a group.

See Fig. 10-56, please select the required objects or object groups in the [Object list window], and then choose the option [Group] under [Object] menu or under right-click menu.
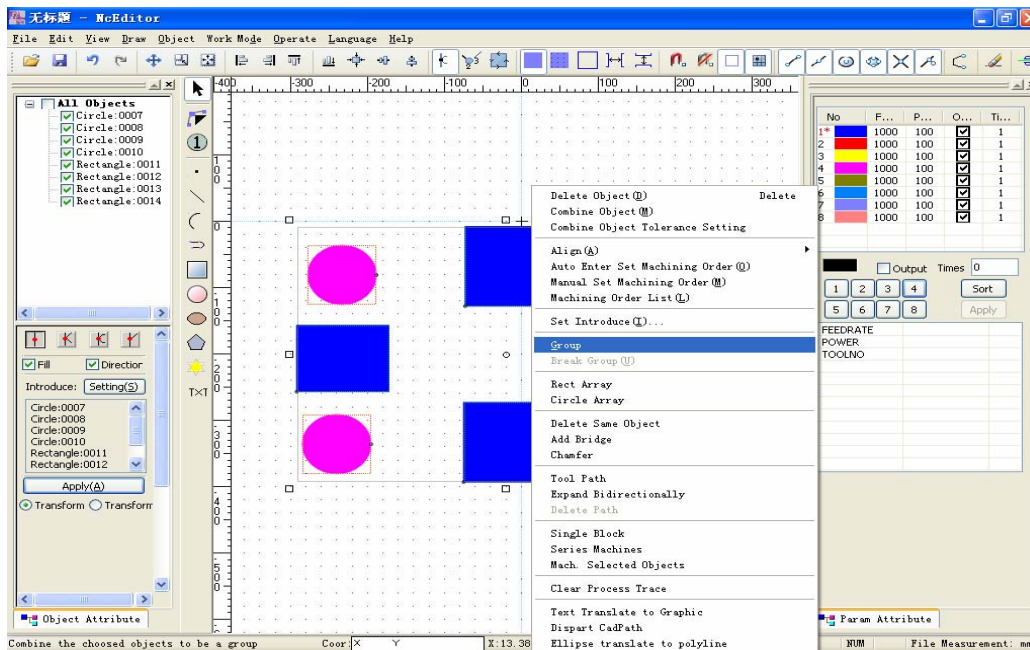
Fig. 10-56 Setting group objects

See Fig. 10-57, the circles and rectangles are set as a new group.
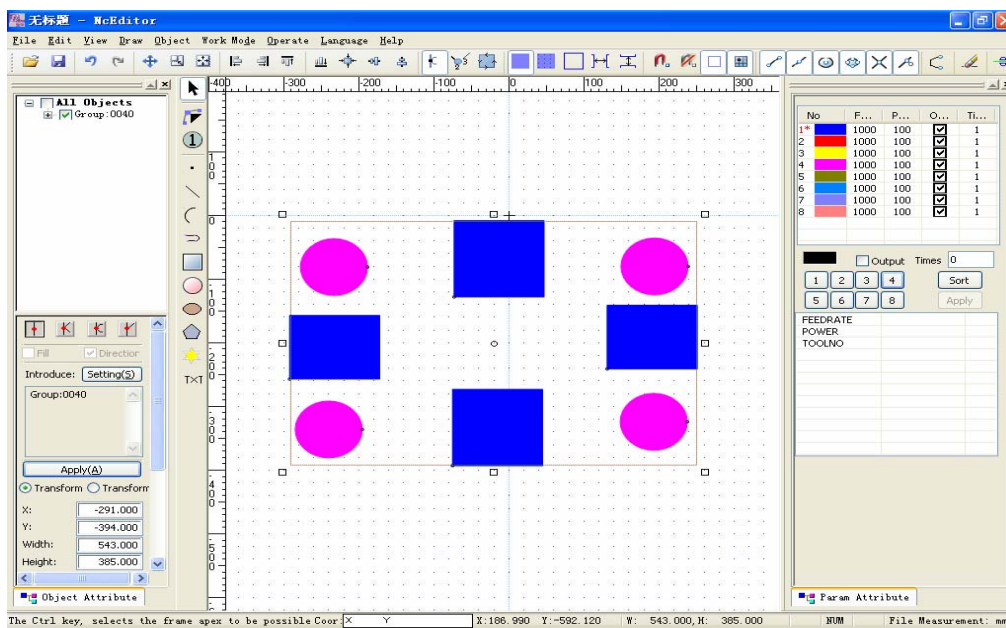
Fig. 10-57 Setting rectangle as a group

At this time, the new group can be edited as a whole.

◆ **[Break Group]**

If you only want to edit parts of the objects in a group, the group should be dismissed first.

Please select the group in the [object list window] or click the group name to highlight it →choose the option [break group] under [Object] or under right-click menu→ the objects in the group will back to the previous group.

◆ **[Rect Array]**

Select one or multiple objects or groups (objects in different groups should be set as a group first) in [object list]→ choose option [Rect Array] under [Object] menu→ input the array rows, columns, line space, column pitch into the dialogue→ click "OK".

For instance, draw a picture as following → set them as a selected group and select it.
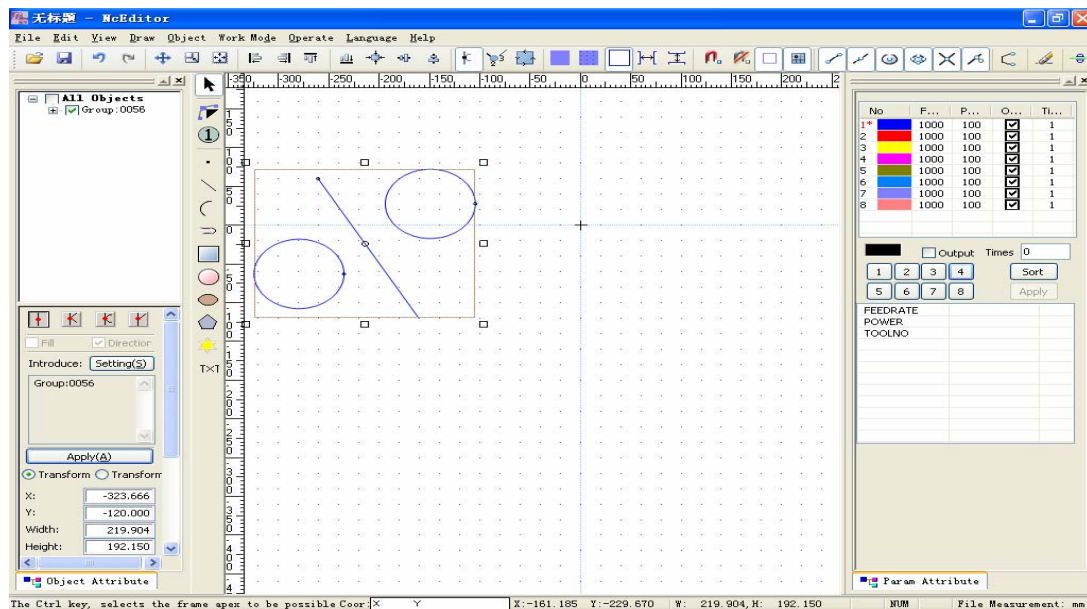
Fig. 10-58 Example figure

Select [Rect Array], input the data and press "Ok", as following, Fig.10-59.
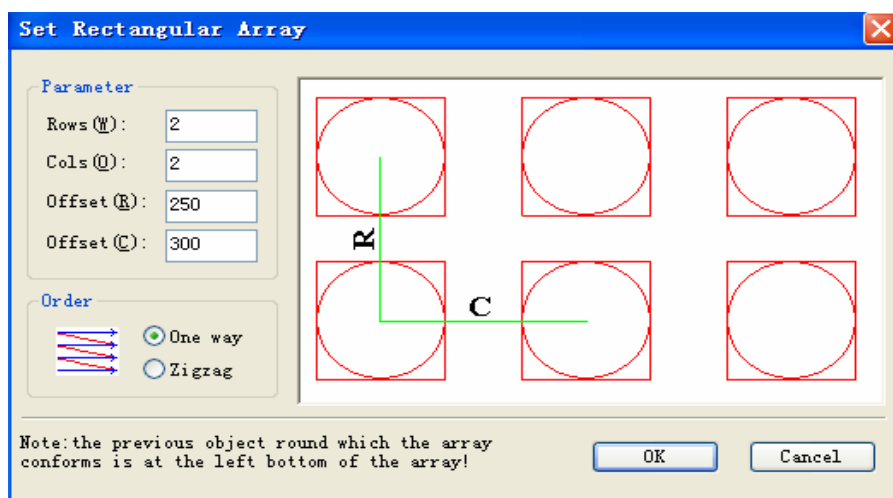

Fig. 10-59 [Rect Array] dialogue

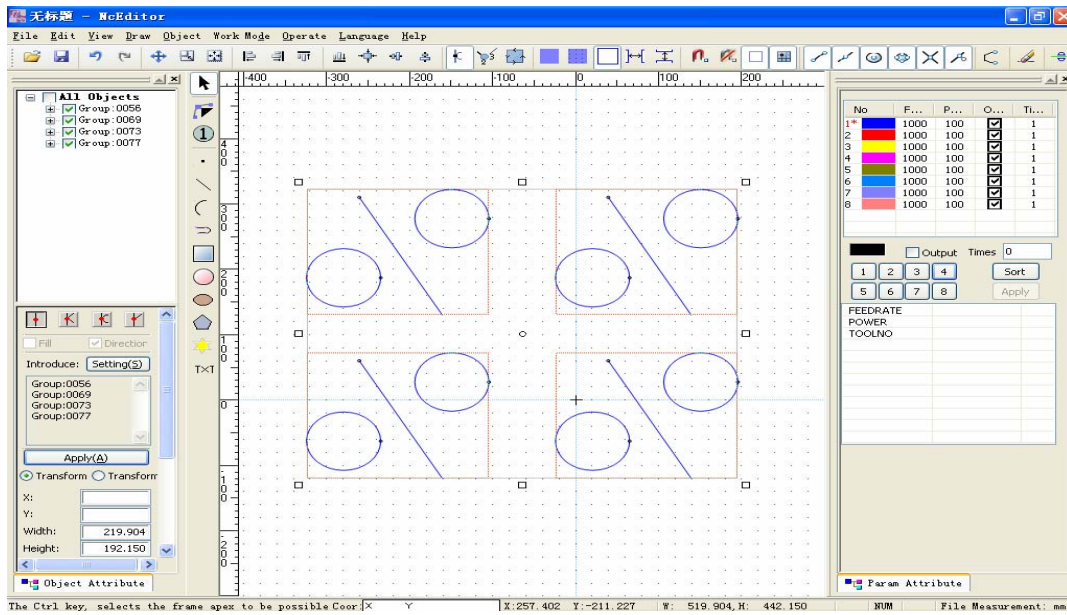The output after setting rectangular array is as following, Fig. 10-60.

Fig. 10-60 Output after setting rectangular array

◆ **[Circle Array]**

Select one or multiple objects or objects in a group in [Object list] (objects in different groups

should be set as a group first) → choose [Circle Array] under [Object] menu or press the button

on toolbar → input the X-coordinate and Y-coordinate of circle center, units on circle into the input

frame→ click "OK".

For instance, select the group in Fig. 10-61→ choose [Circle Array] → input the data as following
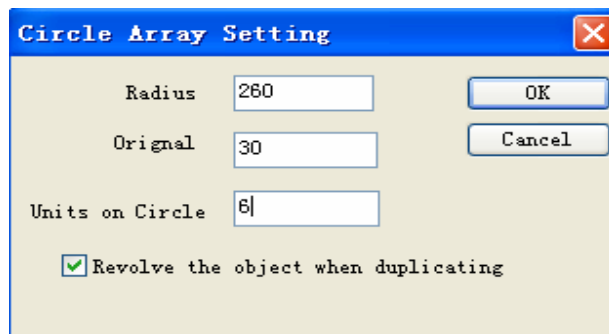
and press "OK".

Fig. 10-61 [Circle Array] dialogue

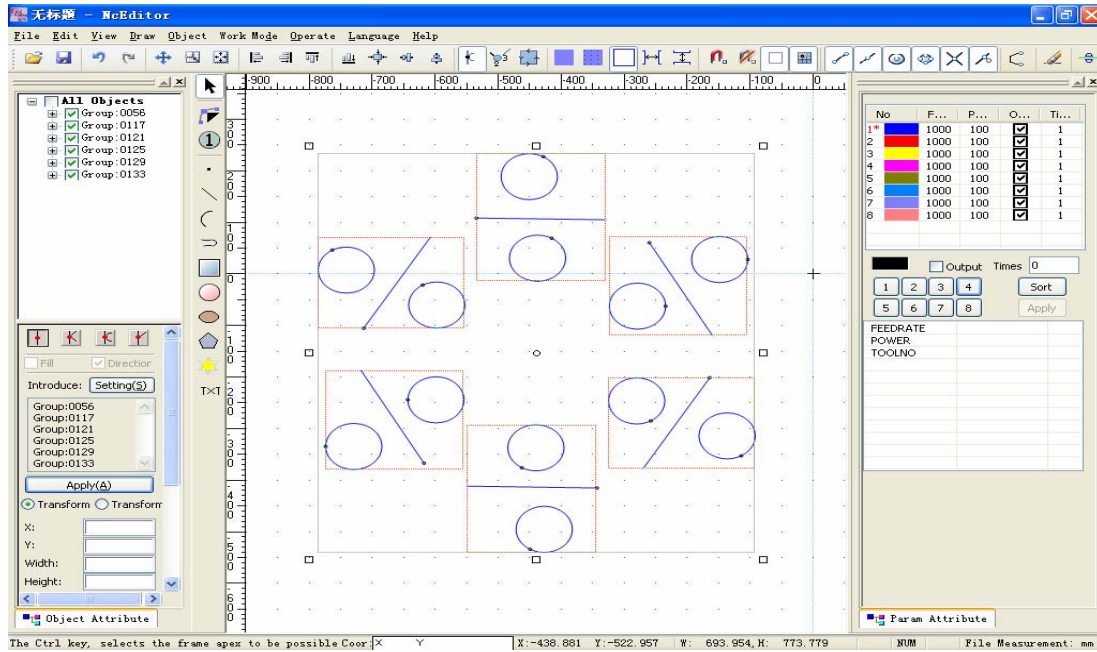The output after setting circle array is as following, Fig.10-62.

Fig. 10-62 Output after setting circle array

Note: multiple objects should be set as a group before using array function.

◆ **[Delete Same Object]**

This function is used to delete the objects of the same size, position and shape, which are totally reduplicated. The manipulation is as following:

➢ Select the objects;

➢ Right-click the mouse → choose [Delete Same Object] under menu or choose [Delete Same Object] under [Object] menu.

NcEditor will save only one machining object and delete the other duplicate objects in a position.

◆ **[Add Bridge]**

The bridge function is to make it more convenient for you to manage the processed parts. After processing, the parts will not fall off and remain in the original position.

[Add Bridge] is divided into [Auto-add-bridge mode] and [Manually-add-bridge mode], and [Add Bridge] is available for a single object, but unavailable for groups.

Currently, [Add Bridge] is available for circle, arc, rectangle and straight line; [Add Bridge] is unavailable for ellipse, ellipse arc and polyline.

◆ **[Auto-add-bridge mode]**
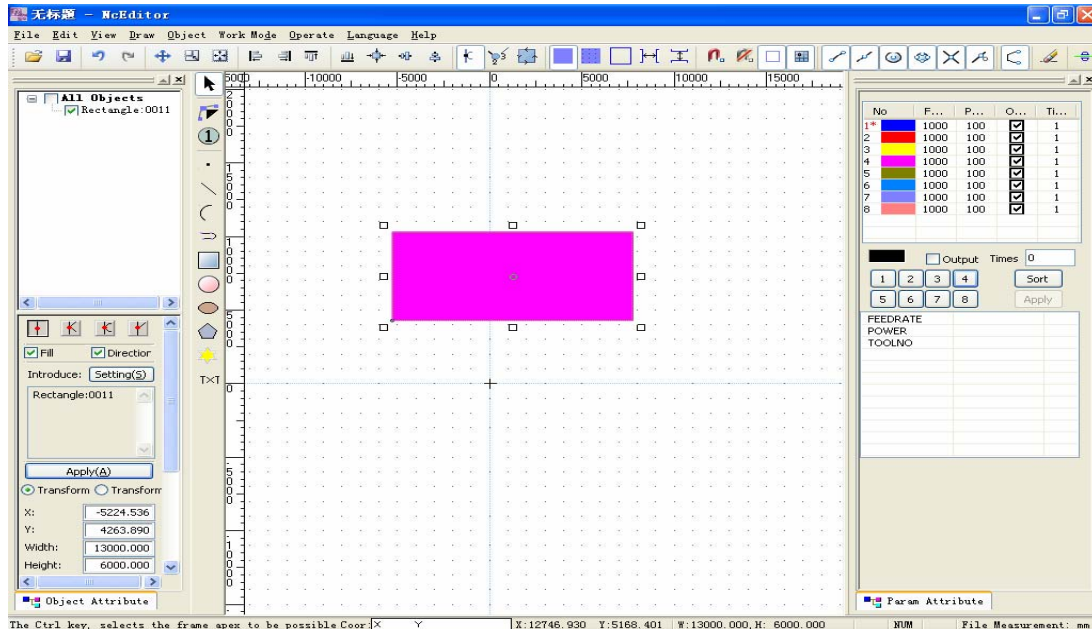
Draw a rectangle and select it as following:

Fig. 10-63 Draw a selected rectangle

Right-click this object →choose [Add Bridge] under right-click menu or under [Object] menu → input data as following:
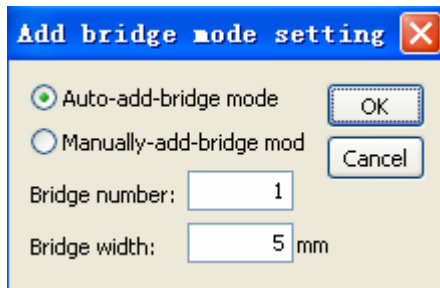

Fig. 10-64 [Auto-add-bridge mode] dialogue

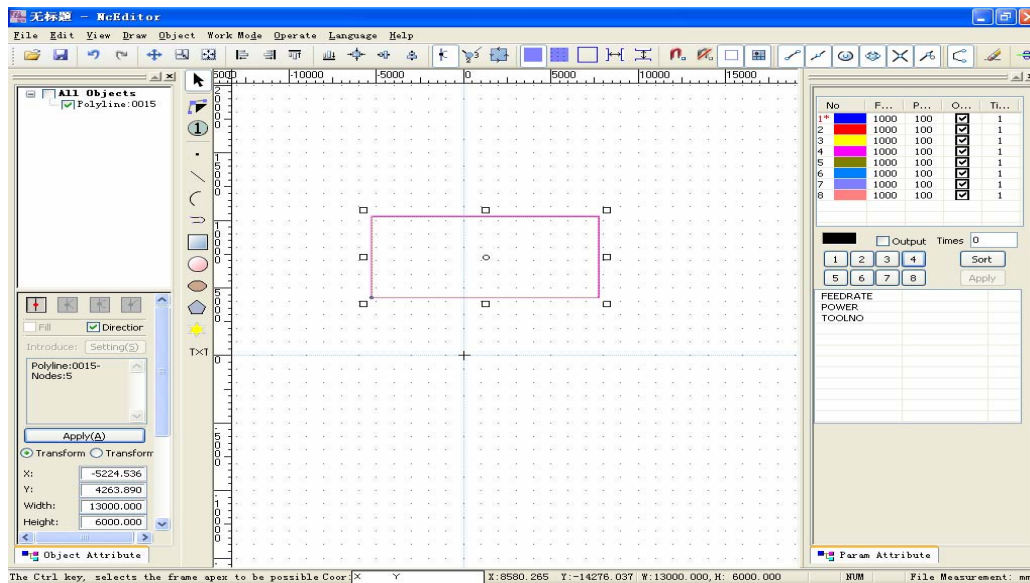Click "OK", the output of rectangle is as following, Fig. 10-65.

Fig. 10-65 After auto adding bridge

The bridge is located in the gap on the lower left corner. And during machining, this gap will be skipped to keep rectangle on the workpiece.

◆ **[Manually-add-bridge mode]**

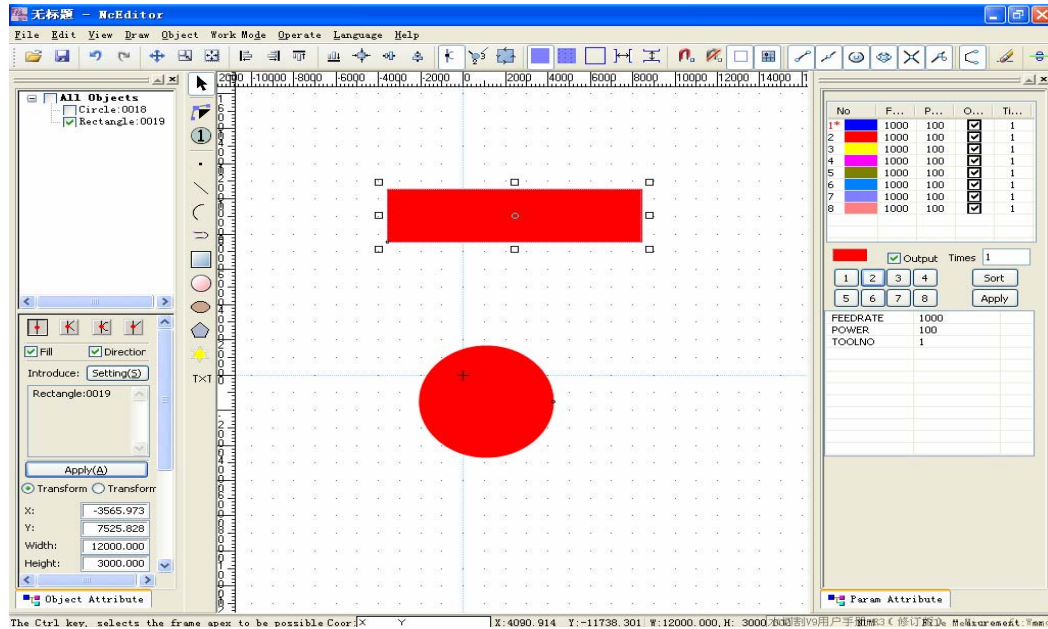Draw a circle and a rectangle without selecting as following:

Fig. 10-66 Example of manual add bridge mode

Select [Add Bridge] under right-click menu or under [Object] menu → choose manual add bridge mode → input the following data and press "OK".
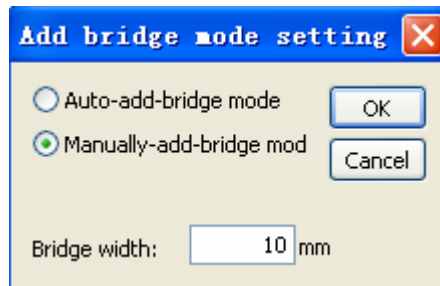

Fig. 10-67 Manual add bridge mode dialogue

Single click the frames of the circle and rectangle respectively to add bridge, as Fig.10-68.
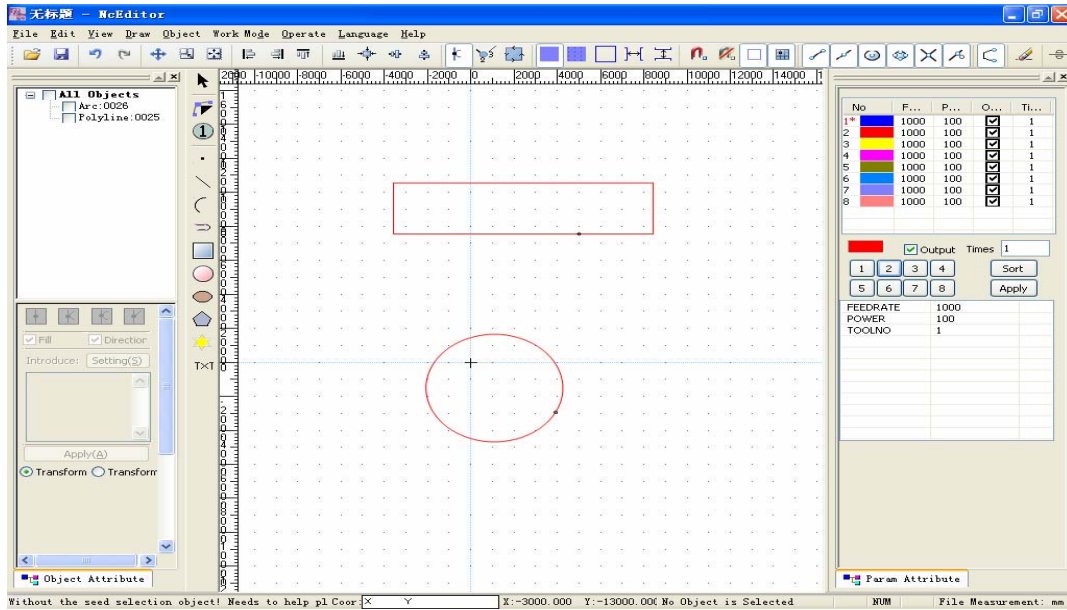
Fig. 10-68 After manual add bridge

◆ **[Chamfer]**

The software supports fillet and chamfer. As Fig.10-69, draw a rectangle and select it.
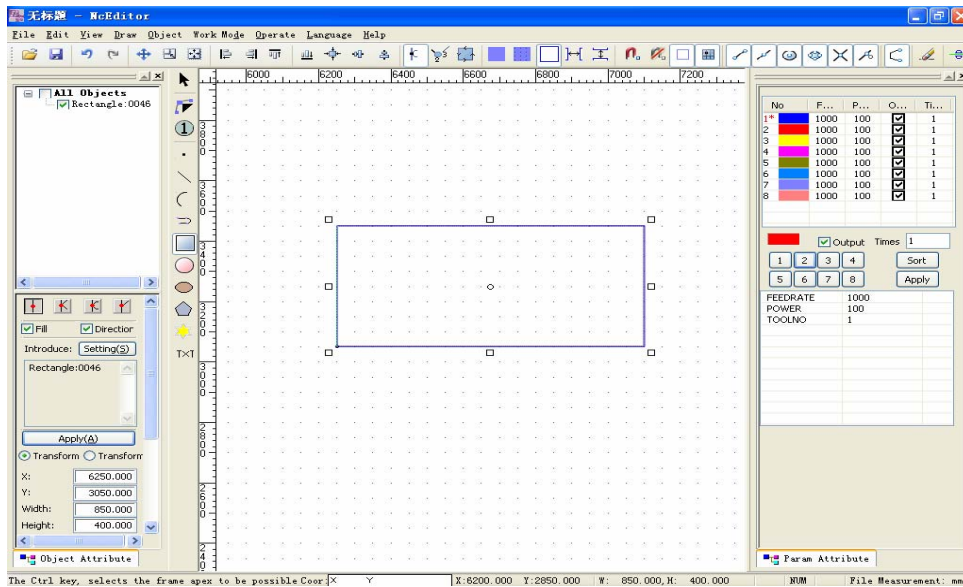

Fig. 10-69 Draw a selected rectangle

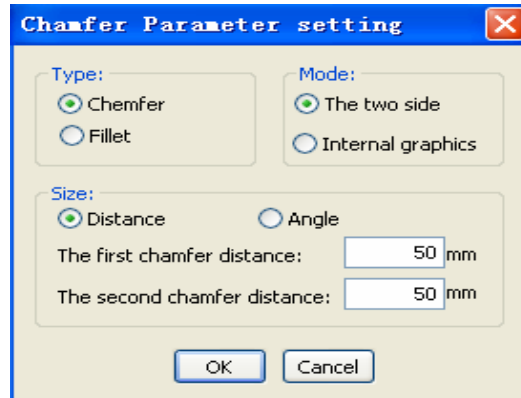Choose [Chamfer] under [Object] menu, the following dialogue will pop up.

Fig. 10-70 Setting chamfer parameters

There are two modes for both fillet and chamfer, [The two side] and [Internal graphics]

◆ **[The two side] mode**

[The two side] mode is ensuring that the chamfer is the angle contained by two sides. See Fig.10-70, under the chamfer mode, there are two modes for setting size, [Distance] and [Angle]. Under [Distance] mode, you can input the distance of two sides, as Fig.10-70, for instance, input 5mm into the input frame → press "OK", cursor reshaping. And the tip will prompt at the bottom of NcEditor. See Fig.10-71.

Fig. 10-71 Tip of selecting the first edge

After the first side of the object is selected, the tip will prompt as Fig.10-72. When the second side, which is adjacent to the first side, is selected, the rectangle will change into the figures as Fig.10-73.

Fig. 10-72 Tip of selecting the second edge
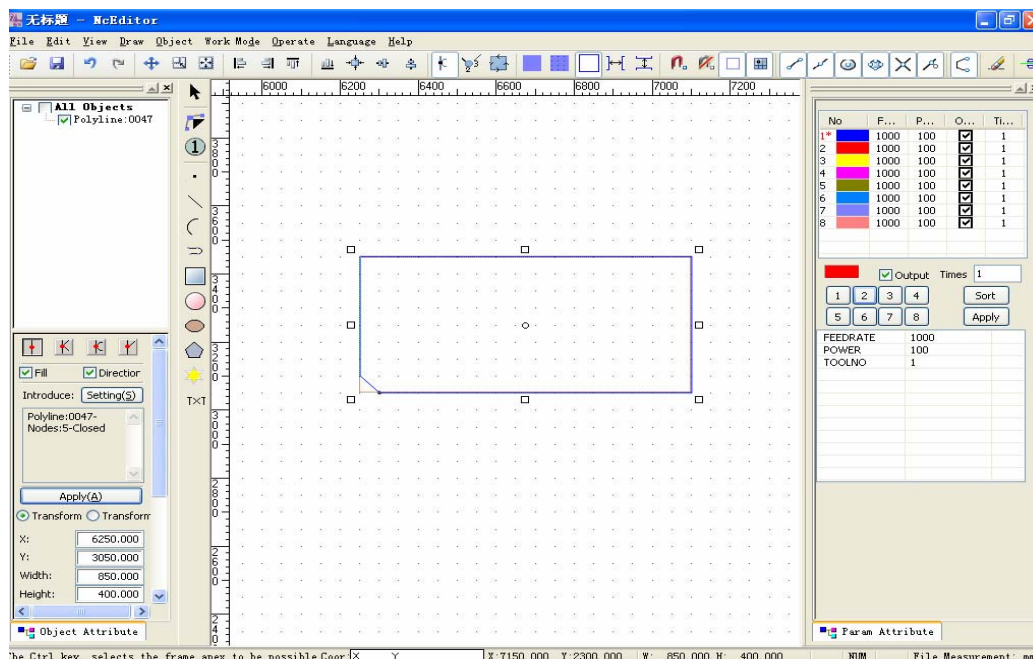
Fig. 10-73 Output after executing [Chamfer]

Input the following data under [Angle] mode of [Chamfer], as Fig.10-74.
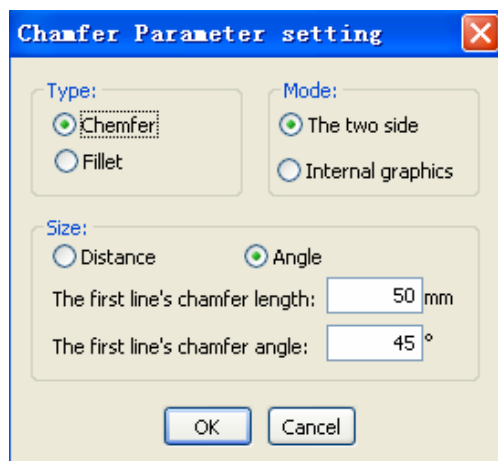
Fig. 10-74 [Angle] mode under [Chamfer] dialogue

After setting the [Chamfer], please select the two adjacent sides, the manipulation is the same as that under [Distance] mode. And the output is the same as in Fig.10-73.

Setting parameters under [The two side] of [Fillet] is used to set the rounded angle of two adjacent sides. The only way for setting the size is [Radius] mode, and input data as Fig.10-75. Press "Ok" and select the two adjacent sides, the output is as Fig.10-76.
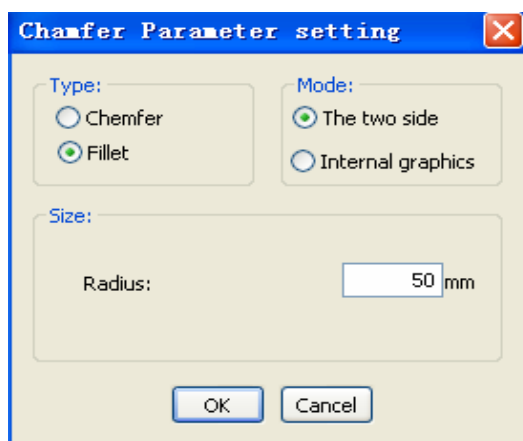
Fig. 10-75 Setting chamfer parameters
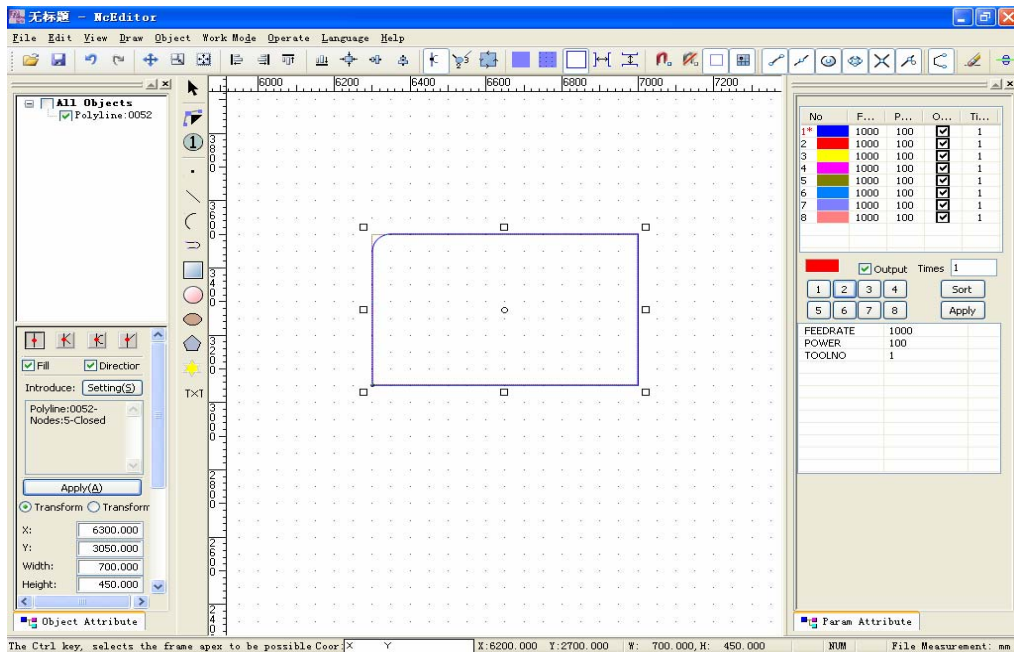
Fig. 10-76 Output after executing [Fillet]

◆ **[Internal graphics]**

[Internal graphics] is ensuring all the qualified chamfers inside the graphics. After selecting [Internal graphics] mode, input data as Fig.10-77 and press "Ok" to select any edge of the figure; system will automatically process all the qualified included angles, and the output is as Fig.10-78. Under [Internal graphics], there are two methods, [Distance] and [Angle], which are identical to the above and will be omitted.
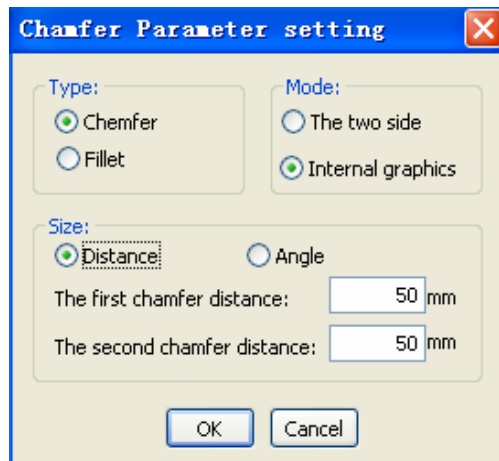
Fig. 10-77 [Internal graphics] mode of [Chamfer]
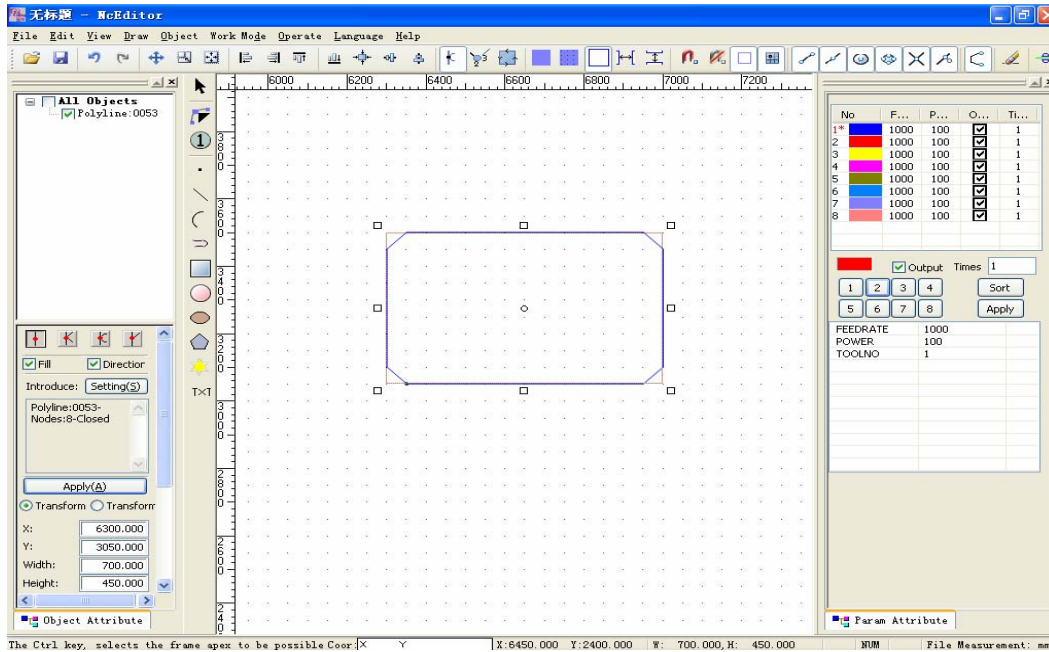
Fig. 10-78 Output after [Internal graphics]

Under the [Internal graphics] mode of [Fillet], system will automatically process all the qualified included angles, inputting radius of 50 mm into the input frame and the output is as Fig.10-79.
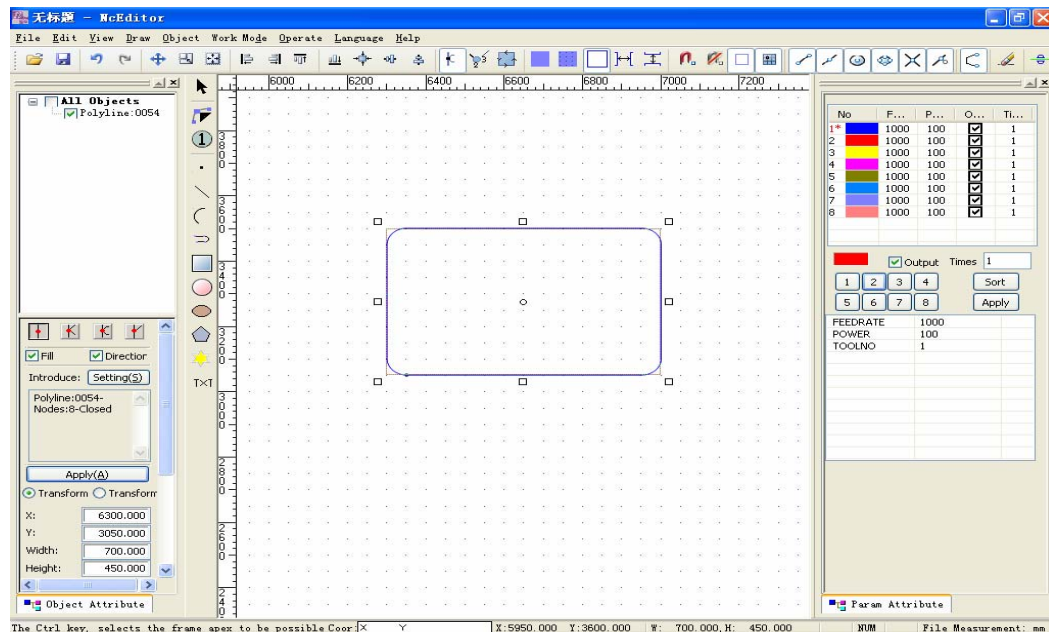
Fig. 10-79 Output after [Internal graphics] mode of [Fillet]

◆ **[Tool Path]**

After selecting an object, you can fulfill [Tool Path] function by following two methods.

➢ Choose the option of [Tool Path] under [Object].

➢ Choose [Tool Path] under right-click menu.

After this option is selected, a dialogue will pop up as Fig.10-80. The default tool compensation is "off"; when the compensation type is "computer", you can input the value of tool diameter,

compensation direction and chord.
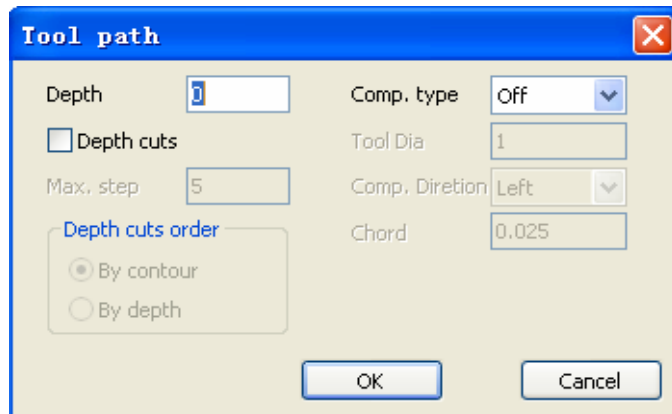
Fig. 10-80 [Tool path] dialogue

When the value of left compensation is "10", the tool path is as following, Fig.10-81.
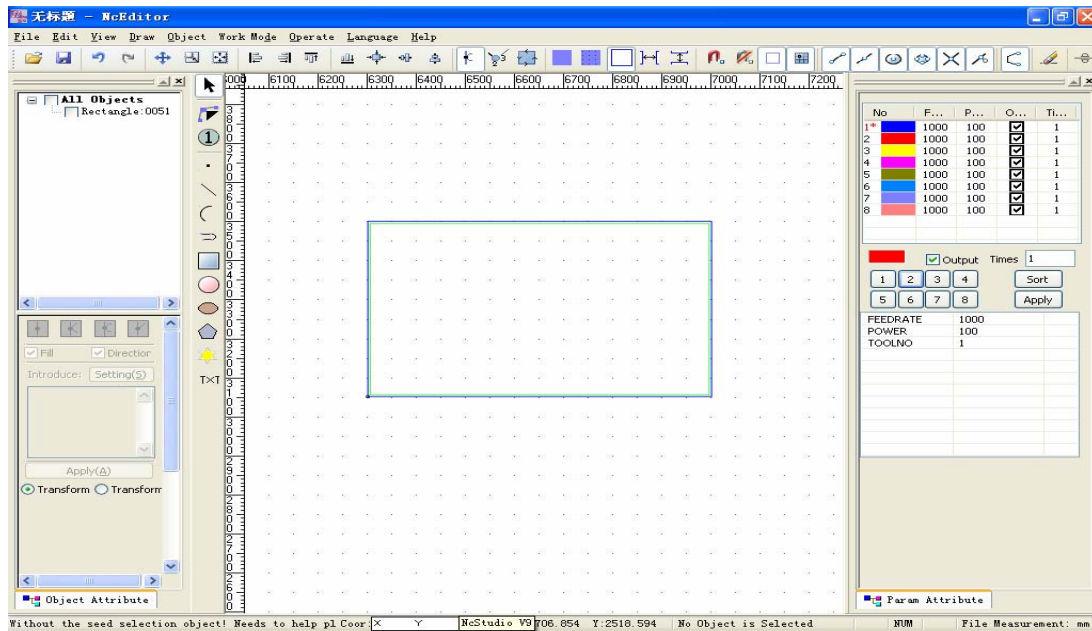
Fig. 10-81[Tool path] of left-compensation as "10"

When the value of right compensation is "10", the tool path is as following, Fig.10-82.
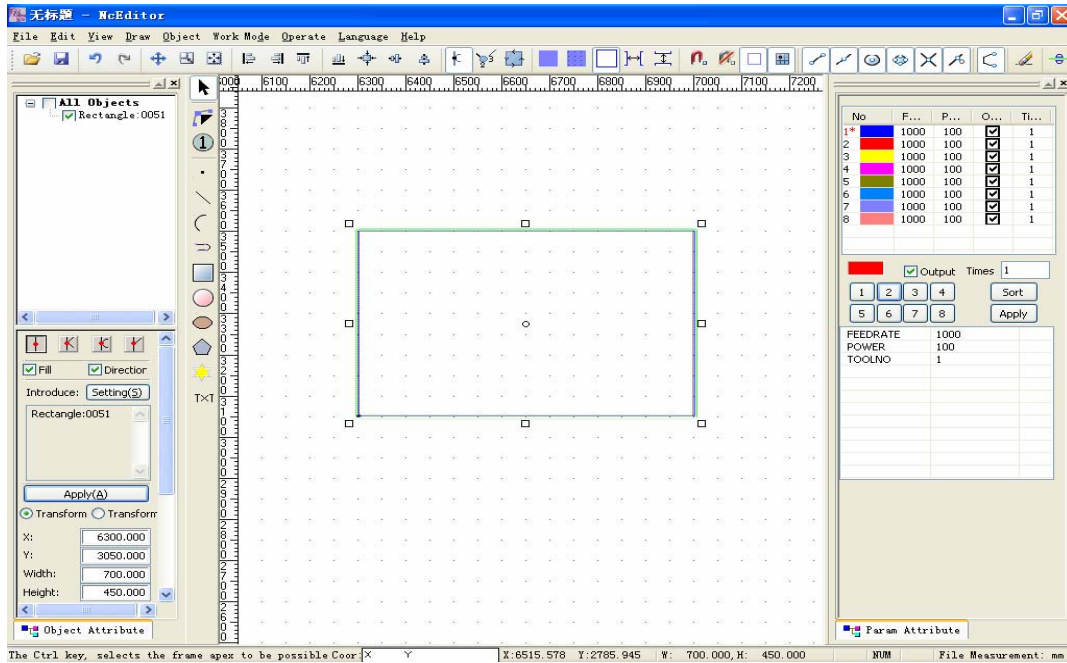
Fig. 10-82 [Tool path] of right-compensation as "10"

◆ **[Expand Bidirectionally]**

In [Tool Path], the compensation direction can be left or right, but not both left and right, however, [Expand Bidirectionally] can fulfill this function (both left and right).

After selecting the object/objects, you can fulfill this function by following two methods.

➢ Choose the option of [Expand Bidirectionally] under [Object].

➢ Choose [Expand Bidirectionally] under right-click menu.

After this option is selected, following dialogue will pop up.
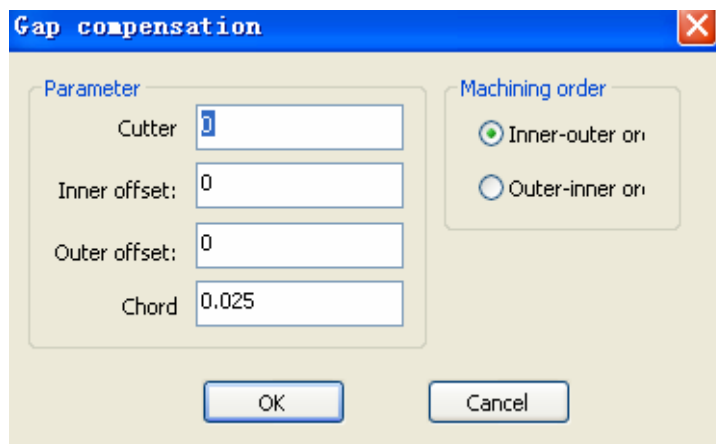
Fig. 10-83 [Gap compensation] dialogue

The parameters of [Cutter] and [Chord] in this function are the same as the ones in [Tool path]. [Inner offset] and [Outer offset] are equal to left compensation and right compensation respectively. After setting the parameters, press "Ok". The view under [Expand Bidirectionally] is as Fig.10-84, when the values of [Inner offset] and [Outer offset] are both "20".
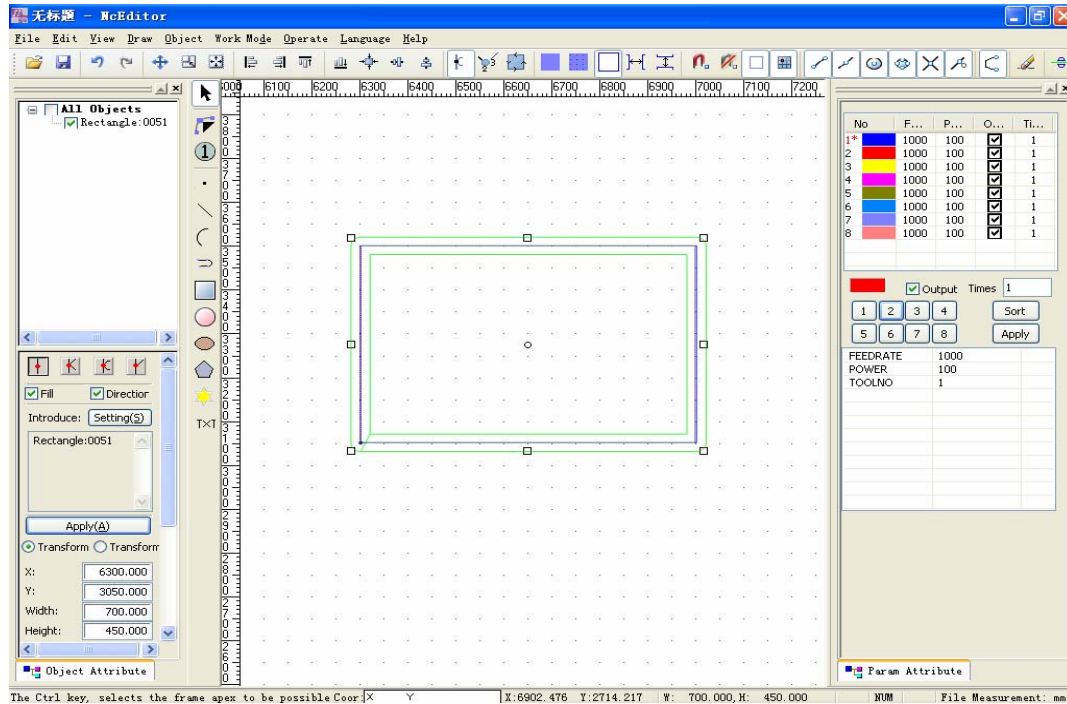
Fig. 10-84 After executing [Expand Bidirectionally]

◆ **[Delete Path]**

After [Tool path] or [Expand Bidirectionally] is executed, [Delete Path] is available. The concrete manipulation is as following:

After selecting the object/objects, you can fulfill this function by following two methods.

➢ Choose the option of [Delete Path] under [Object].

➢ Choose [Delete Path] under right-click menu.

◆ **[Series Machines]**

When you want to machine several graphics continuously, from the last graphic to the next one without lifting tool, you need to connect all the graphics. After [Series Machines] is selected, system will connect the graphics automatically and go on machining continuously.

After selecting two or above two graphics, you can choose [Series Machines] under [Object] menu or under right-click menu.

All the graphics will be connected after [Series Machines] is pressed, as Fig.10-85.
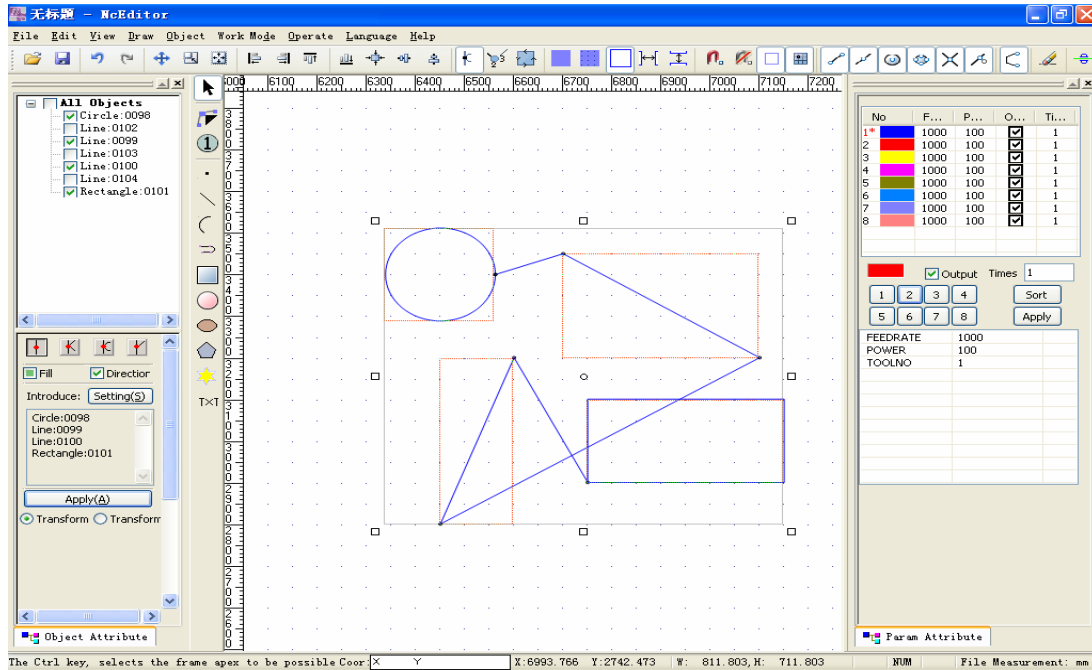
Fig. 10-85 Example of [Series machining]

◆ **[Clear Process Trace]**

When you are machining an object, the red tracks on the object means "processed". After machining is finished, the object will turn to red. Actually, the color is the processing track, rather than the color of the graphic. Therefore, you can select [Clear Process Trace] to clear the track.

After machining is finished, there are several methods to fulfill [Clear Process Trace] as following:

➢ Choose the option of [Clear Process Trace] under [Object];
➢ Choose [Clear Process Trace] under right-click menu;

➢ Press the icon ✎ on toolbar.

Note: [Clear Process Trace] is unavailable during machining.

◆ **[Text Translate to Graphic]**

Press button ⊡ and click in the editor space, as following, Fig.10-86.
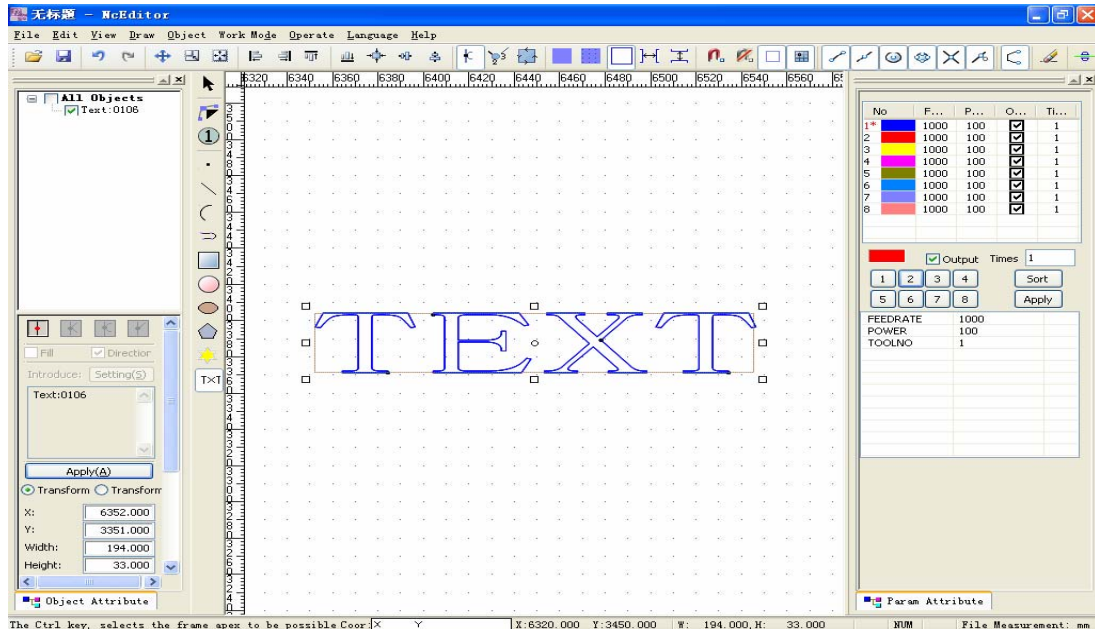
Fig. 10-86 Step one of [Text translate to graphic]
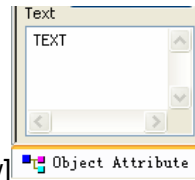
Input the desired text in "Text" of [object attribute window] → press [Apply] → the input text will be displayed in the editor space, as following, Fig.10-87.
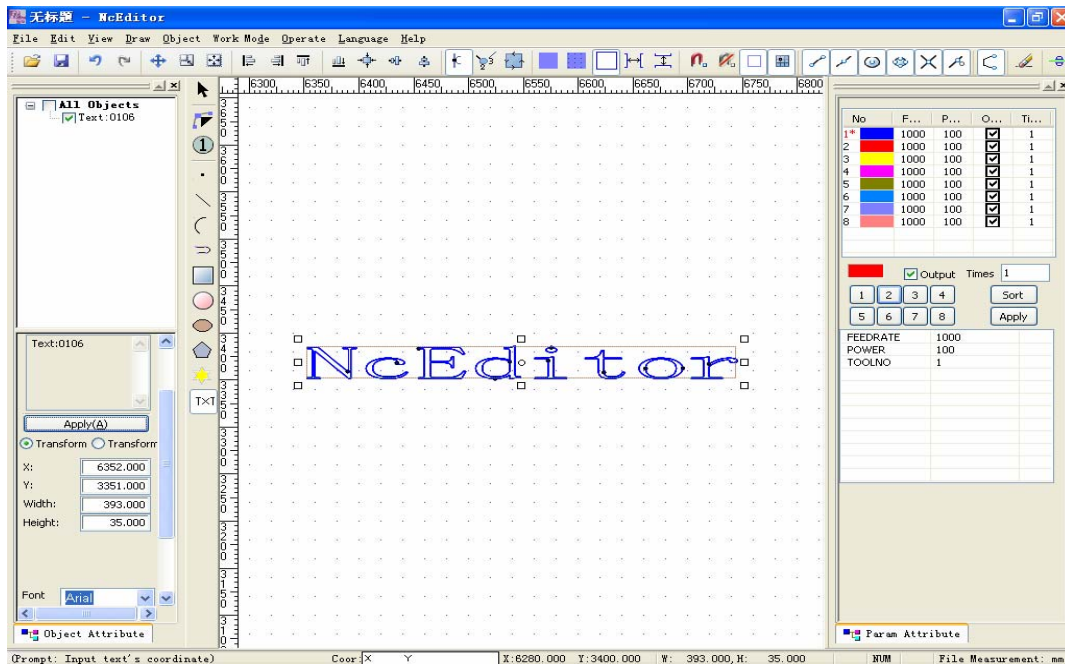
Fig. 10-87 After inputting text

Select this text → click the right mouse button → select the option [Text Translate to Graphic] or select [Text Translate to Graphic] under [Object], the output is as following.
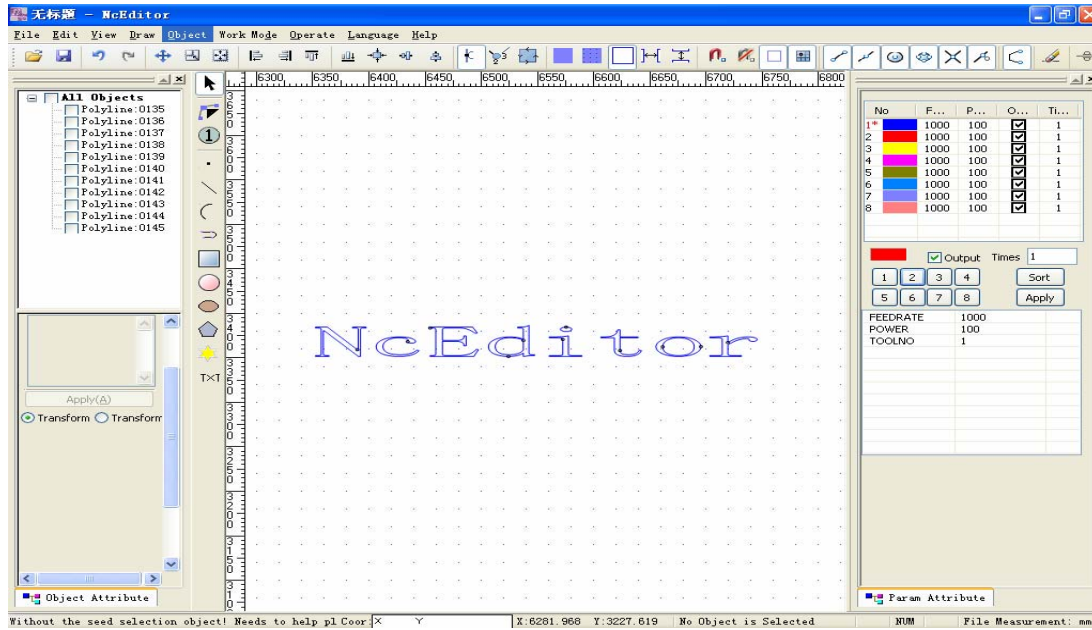
Fig. 10-88 Output after [Text translate to graphics]

◆ **[Dispart CadPath]**

[Dispart CadPath] is assorting with [Add Bridge], which is available after [Dispart CadPath].

◆ **[[Ellipse translate to polyline]**

Draw an ellipse as following, Fig.10-89.
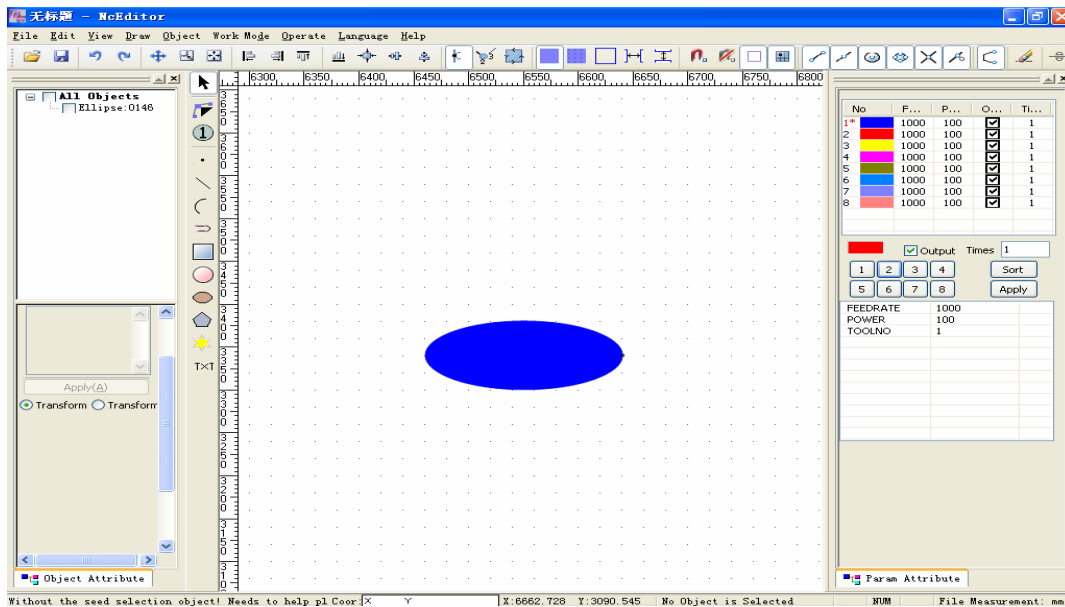
Fig. 10-89 Example of [Ellipse translate to polyline]

Select this ellipse → choose [Ellipse translate to polyline] under right-click menu, as following, Fig.10-90.
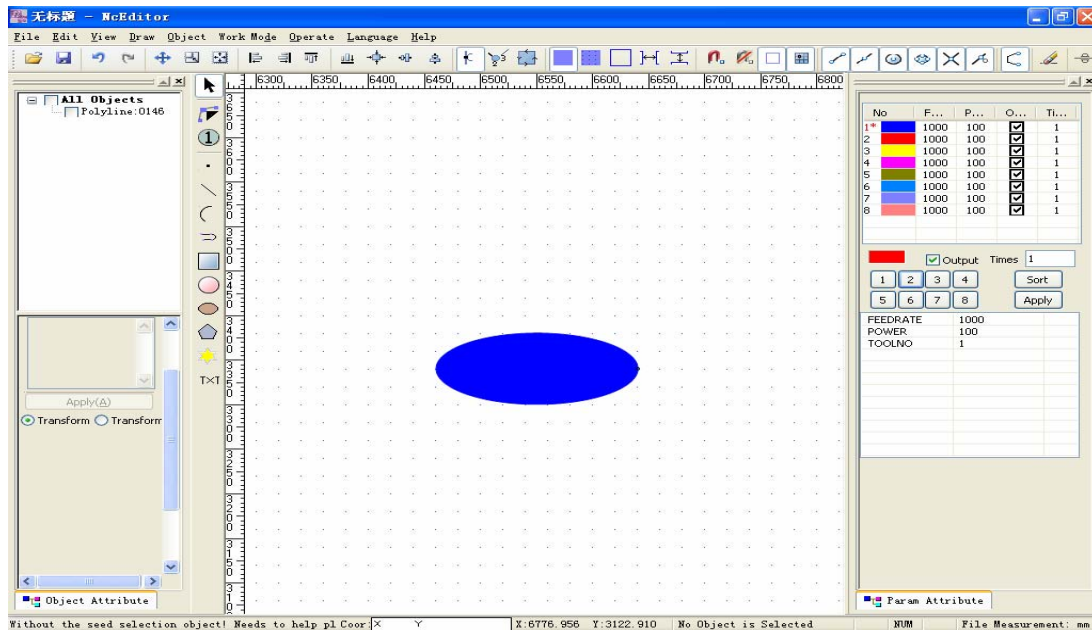
Fig. 10-90 After [Ellipse translate to polyline]

◆ **[options]**

It is used to fulfill some parameters setup in importing a file. After [options] is selected, the following dialog will pop up.
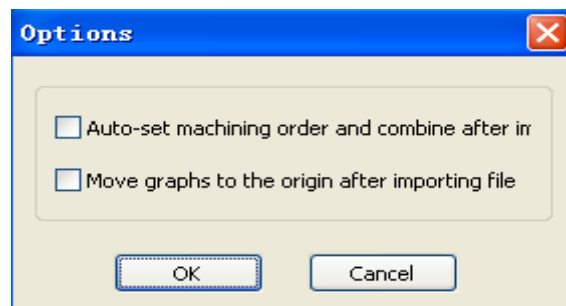
Fig. 10-91 "Options" dialog

10.7.6 Operate menu

Many options in [operate] menu are the same as those in [operate] menu of Ncstudio, and the same options won't be introduced here.

◆ **[Mach. Selected Objects]**

This function is used to machine the selected figures. After finishing machining the selected objects, the software will stop machining, and will not process other unselected figures. You can select one figure or multiple figures.

◆ **[Mach. By Color]**

You can choose the colors of objects in drawing and order the colors. Then the machining can be executed following the order of colors, for instance, first red, then blue.

After [Mach. By Color] is selected, a [Sort color line] dialogue will pop up, as following, Fig.10-92.

Fig. 10-92 [Sort color line] dialogue

In this dialogue, you can set the machining order of different colors.

◆ **[Mach. Current Group]**

If you choose one group and only need to machine this group, this function can be selected. This function is only available for the current selected group. It is unavailable for a single figure.

◆ **[Mach. From Selected Objects]**

There is an order for to-be-machined objects. After [Mach. From Selected Objects] is selected, only the selected objects and its backward objects will be machined, its forward ones will not be machined. For instance, there are 8 objects to be machined, if the sequence of the selected object is "3", then the No. 3 to No. 8 objects will be machined, but No.1 and No.2 won't.

11 Features of Three-axis Waterjet Cutting

Apart from some parameters, the manipulation and functions of NcStudio and NcEditor for three-axis and two-axis waterjet control system are similar. This chapter specializes in introducing some special parameters and functions of three-axis waterjet cutting. For other manipulation and functions, please refer to chapter 1 to chapter 10.

11.1 Integral interface of Ncstudio^{TM}
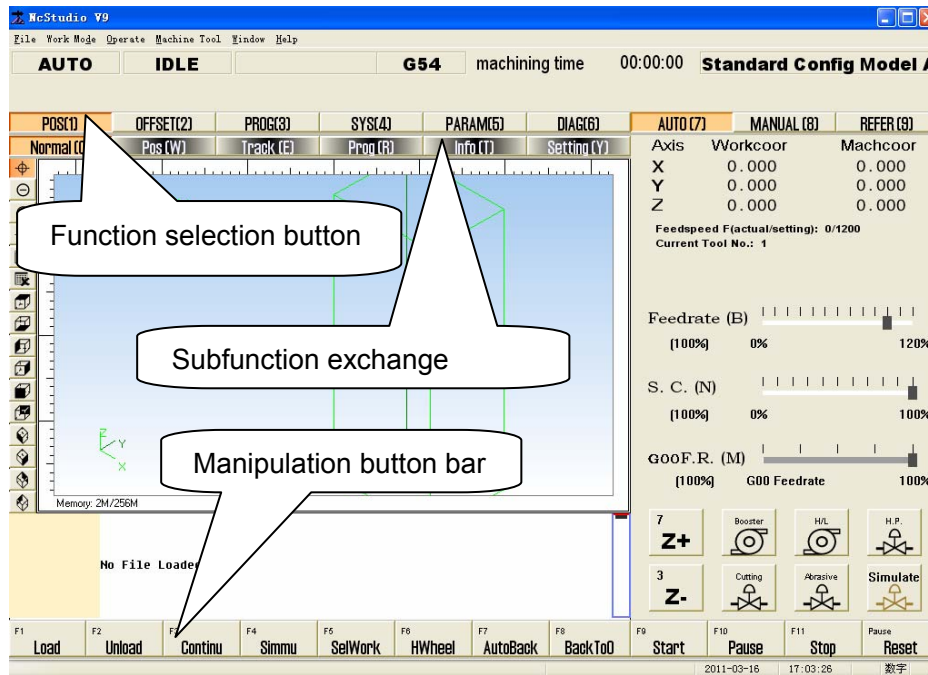
Fig. 11-1 The main operation interface

The main operation interface of Ncstudio for 3-axis waterjet cutting is as Fig.11-1.

As a 3-axis waterjet cutting software, Z-axis is added in its coordinates and functions. And the other functions in the operation panel are the same as those in 2-axis waterjet cutting.

11.2 Operation panel of Ncstudio^{TM}

The functions of 3-axis waterjet cutting, under auto, manual and reference point mode, are almost the same as those of 2-axis waterjet cutting apart from some exclusive ones, which will be introduced in detail as following.

11.2.1 POS under AUTO mode

Under AUTO mode, [POS] subfunction section includes: Normal, Pos, Track, Prog, Info, and Setting. See Fig 11-2 below:

Fig. 11-2 Position sub-function buttons under AUTO mode

◆ **Normal function screen**

See Fig.11-3.

| POS(1) | OFFSET(2) | PROG(3) | SYS(4) | PARAM(5) | DIAG(6) |
| Normal (O) | Pos (W) | Track (E) | Prog (R) | Info (T) | Setting (Y) |
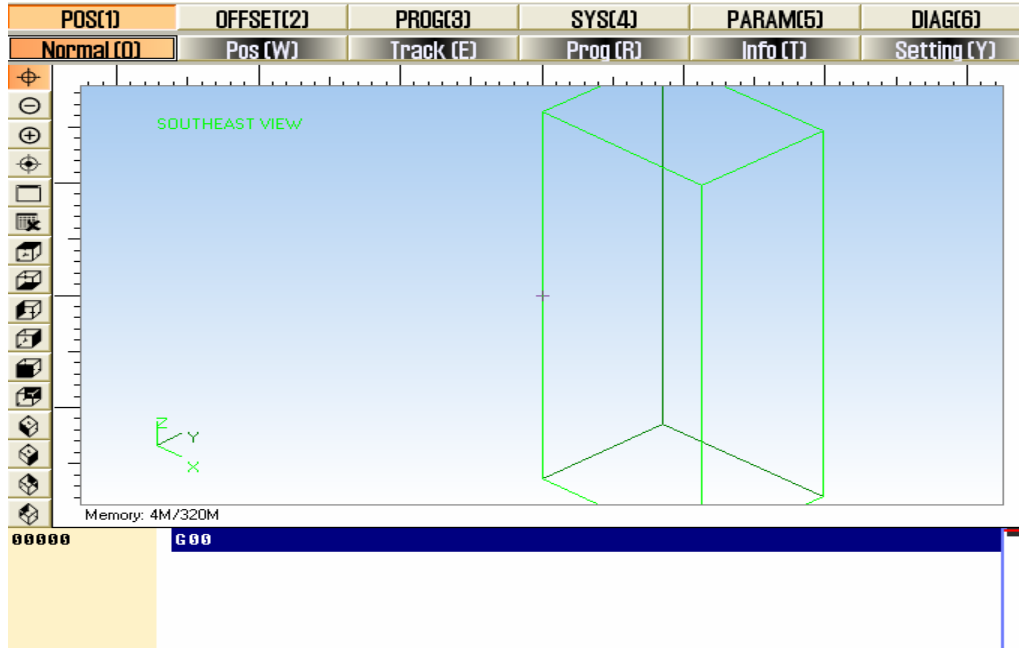
Fig. 11-3 Normal function screen

In the process track window, the path of water jet is in real time. User can check water jet's track intuitionally and make sure the machining procedure executed accurately. Besides, under this three-dimension track mode, user is offered abundant operating methods to look over the processing figure from different position, view angle and scaling, such as zoom in, zoom out, etc.

◆ **[Icon Button]**

On the left of the process track window, there are icon buttons, which are more than those of 2-axis waterjet cutting. See Fig. 11-4.
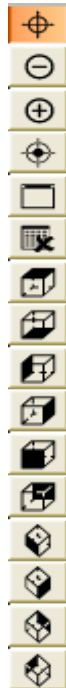
Fig. 11-4 Icon buttons on the left of the process track window

The system provides the top 10 most used groups of angle of view. You can exchange via the letter keys (the letters ahead of these view buttons).

W: top view;

X: bottom view;

A: left view;

D: right view;

S: front view;

B: back view

Z: southwest view:

C: southeast view;

E: northeast view;

Q: northwest view;

11.2.2 OFFSET under AUTO mode

◆ **WorkCoor function screen**

See Fig. 11-5. In this function screen, [Assistant Function] is added, which will help you adjust the feed rate of Z-axis finely.
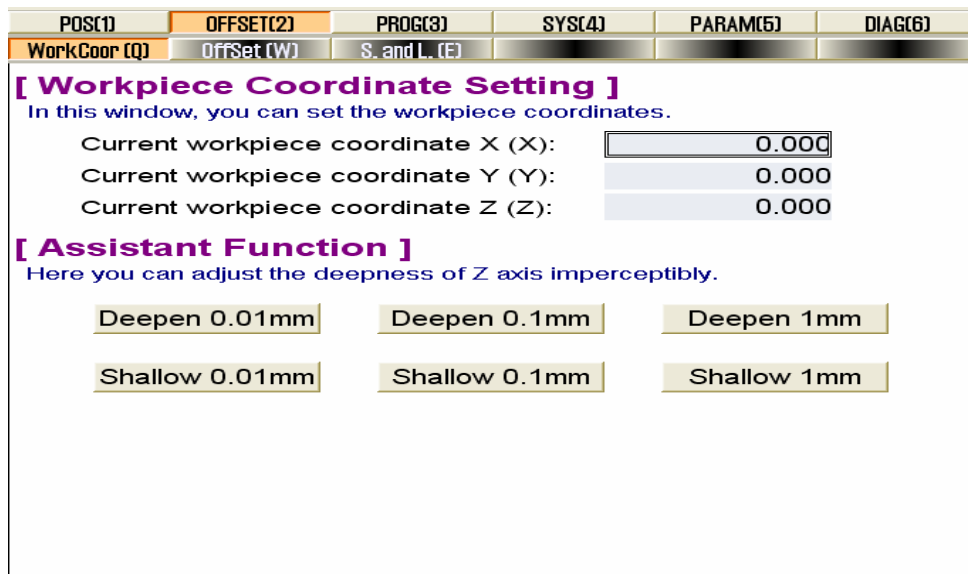
Fig. 11-5 Coordinate function screen in OFFSET function section

11.2.3 PROG under AUTO mode

◆ **[ProcessWiz] function screen**

◆ **[Rectangle Frame Incision Wizard]**

Hotkey: F1, only available in the [ProcessWiz] screen of [PROG] function section. Press the

button, system will change to [Rectangle Frame Incision Wizard] function Screen, as Fig 11-6.

Three parameters have been added here: cutter diameter, carven depth and layer depth.
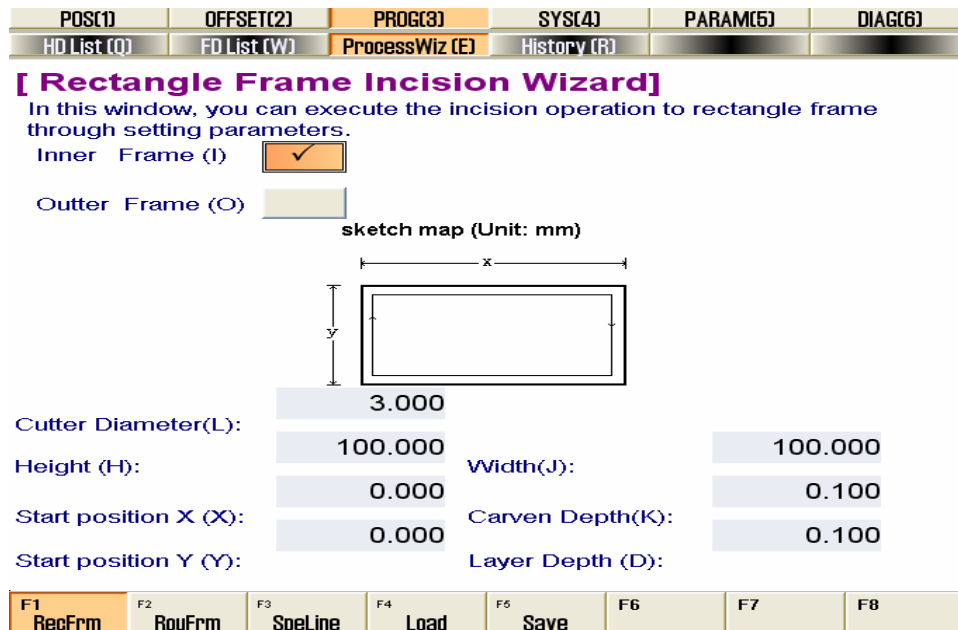
Fig. 11-6 Rectangle frame incision wizard

◆ **[Round Frame Incision Wizard]**

Hotkey: F2, only available in the [ProcessWiz] function screen of PROG function section. Press the button, system will change to [Round Frame Incision Wizard] function screen, as Fig 11-7.

There are three new parameters: cutter diameter, carven depth and layer depth.
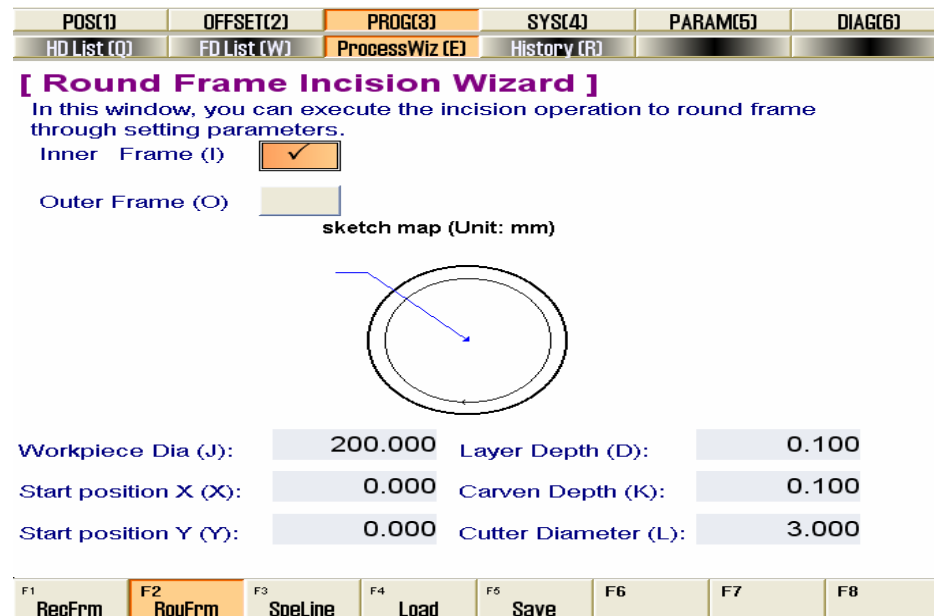
Fig. 11-7 Round frame incision wizard

11.2.4 SYS under AUTO mode

◆ **[Config] function screen**

The screen is as Fig. 11-8. It displays the configurations supported by NcStudio of 3-axis waterjet cutting. Press [Select] button to select the corresponding configuration according to actual situation.

3-axis waterjet cutting only has 3 standard configuration models without revolution configuration models.
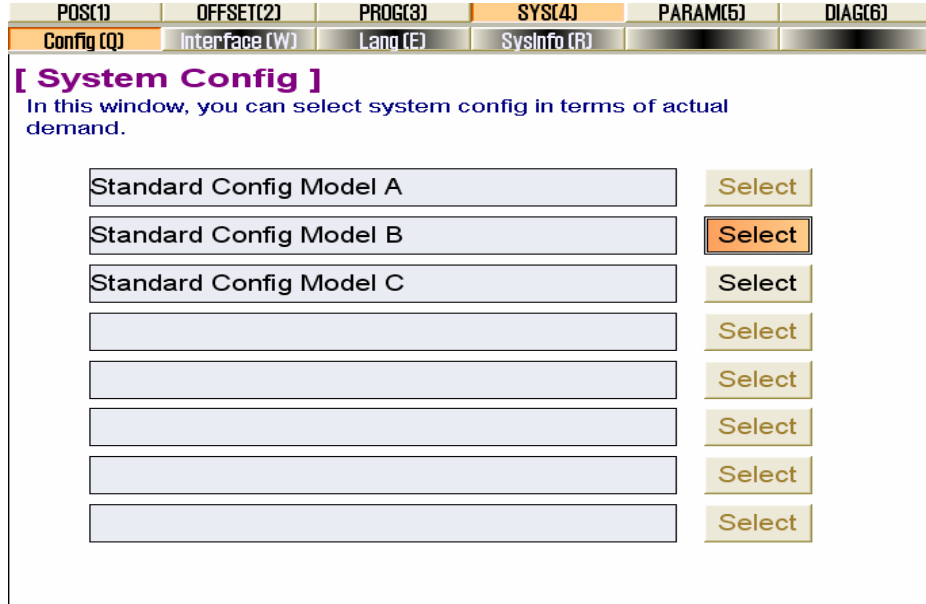
Fig. 11-8 Configuration function screen

➢ Standard Config Model A: it means that the output signals of all the pumps and valves are continuous.

➢ Standard Config Model B: it means that the signal output of oil pump is jog output while the output signals of other pumps and valves are continuous output.

➢ Standard Config Model C: it means that oil pump and high pressure valve is jog signal output while other pumps and valves are continuous signal output.

11.3 Parameters of Ncstudio^{TM}

There are some exclusive waterjet cutting and commonly used parameters in 3-axis waterjet cutting software, which will be introduced in detail as below. For the other parameters, please refer to chapter 9.

11.3.1 General parameters (manufacturers')

1.0. Axes general

◆ **N10002   AXIS_DIR (Z)**

Type:	INT
Unit:	None

Range:	1: Positive Direction -1: Negative Direction
Default:	-1
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the incremental direction of machine coordinates value of Z-axis: "1" indicates the positive increment, and "-1" indicates the negative increment.

◆ **N10012 AXIS_PULSE_FACTOR (Z)**

Type:	Float
Unit:	mm/p（millimeter/pulse）
Range:	0.0000009~999.0
Default:	0.001
Valid time:	Be valid after restarting the system.
Instruction:	Pulse least increment is the displacement or angle produced on the corresponding axis by each control pulse. Specifically, it refers to the least displacement which the motion control card can deal with. In stepping system, it is corresponding to a stepping pulse, then according to transmission connection, the angle displacement of the stepping pulse will be converted into linear value.

◆ **N10022 WORKBENCH_LOWER_LIMIT（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	Z: -1000.0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the lower limit of the travel range of the worktable on the direction of Z-axis.

◆ **N10032 WORKBENCH_UPPER_LIMIT（Z）**

Type:	Float

Unit:	mm（millimeter）
Range:	Lower limit of the worktable travel range (Machine Coordinate)~99999.0
Default:	Z: 1000.0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the upper limit of the travel range of the worktable on the direction of Z-axis.

◆ **N10042　CHECK_WORKBENCH_RANGE（Z）**

Type:	BOOL
Unit:	None
Range:	0 (False): Disabled 1(True): Enabled
Default:	1（True）
Valid time:	Be valid after restarting the system.
Instruction:	It sets whether to check the travel range of worktable.

1.2. Compensations

◆ **N12012　BACKLASH（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~1.0
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the lead screw backlash of Z-axis. It is effective under the condition that parameter "N12001" is set as true.

1.3. Velo/Acc limits

◆ **N13002　AX_MAX_VOL（Z）**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.001~100000.0
Default:	48000.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the maximum allowed acceleration of Z-axis.

5.2. Handwheel

◆ **N52020   SupportNKExKeyset**

Type:	INT
Unit:	None
Range:	0: Null 1: Mini keyboard 2: Extended keyboard
Default:	0
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the type of the extended keyboard.

◆ **N52021   InstallEctendedKeyboardHook**

Type:	BOOL
Unit:	None
Range:	0 (False): Not install 1 (True): Install
Default:	1 (True)
Valid time:	Be valid after restarting the system.
Instruction:	It specifies whether to install the extended keyboard hook.

6.2. G code options

◆ **N62092   G09_TOL（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~99.0
Default:	0.001
Valid time:	Be valid immediately without restarting the system.
Instruction:	It can be used together with the accurate stop command of G09 and with the parameters of pulse equivalent. Regarding the detailed instruction, please refer to the additional introduction of parameters.

◆ **N62410   ENABLE_CUTTER_COMPENSATION**

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default:	0 (False)
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets whether the cutter compensation instruction is valid.

6.5. File translation

File translation parameters include parameters of PLT, DXF and ENG file.

◆ **N65000   TOOL_UP_HEIGHT (PLT_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~99999.0
Default:	0
Valid time:	Be valid after loading processing procedure again.

Instruction:	It specifies the height of cutter lifting in dry running.

◆ **N65001 PLT_UNIT (PLT_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	None
Range:	0.001~99999.0
Default:	40
Valid time:	Be valid after loading processing procedure again.
Instruction:	It specifies the unit of PLT format.

◆ **N65003 CUTTING_DEPTH (PLT_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~0.0
Default:	0
Valid time:	Be valid after loading processing procedure again.
Instruction:	It is used to specify the processing depth of 2D file.

◆ **N65100 TOOL_UP_HEIGHT (DXF_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~99999.0
Default:	0
Valid time:	Be valid after loading processing procedure again.
Instruction:	It specifies the height of cutter lifting in dry running.

◆ **N65101 CUTTING_DEPTH (DXF_FORMAT_CONVERT_PARAM)**

Type:	Float

Unit:	mm（millimeter）
Range:	-99999.0~0.0
Default:	0
Valid time:	Be valid after loading processing procedure again.
Instruction:	It is used to specify the processing depth of 2D file.

◆ **N65102 DEPTH_LAYER (DXF_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~0.0
Default:	0
Valid time:	Be valid after loading processing procedure again.
Instruction:	It specifies the cutting depth of 2D file processing each time.

◆ **N65104 ENABLE_MACHINE_INDIVIDUALLY (DXF_FORMAT_CONVERT_PARAM)**

Type:	BOOL
Unit:	None
Range:	0(False):Invalid 1(True):Valid
Default:	0（False）
Valid time:	Be valid after loading processing procedure again.
Instruction:	It specifies that one shape will be machined at one time, and machine will go to process the next one when the previous one is finished.

◆ **N65200 TOOL_UP_HEIGHT (ENG_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~99999.0

Default:	0
Valid time:	Be valid after loading processing procedure again.
Instruction:	It specifies the height of cutter lifting in dry running.

◆ **N65201 ENABLE_CHANGE_TOOL_PROMPT (ENG_FORMAT_CONVERT_PARAM)**

Type:	BOOL
Unit:	None
Range:	0(False):Not prompt 1(True):Prompt
Default:	1（True）
Valid time:	Be valid after loading processing procedure again.
Instruction:	It sets whether to prompt when tool change occurs.

◆ **N65204 DeepHoleMachiningWay (ENG_FORMAT_CONVERT_PARAM)**

Type:	INT
Unit:	None
Range:	0: Reciprocating chip removal 1: High speed reciprocating chip removal
Default:	0
Valid time:	Be valid after loading processing procedure again.
Instruction:	It indicates the manner of processing deep hole.

◆ **N65205 RETRACT_VALUE (ENG_FORMAT_CONVERT_PARAM)**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~99999.0
Default:	1
Valid time:	Be valid after loading processing procedure again.
Instruction:	It indicates retract quantity after feeding when drilling deep hole in high speed reciprocating chip removal.

7.2. Auto

◆ **N72012   ENABLE_WORKBENCH_CHECK（Z）**

Type:	BOOL
Unit:	None
Range:	0(False):Invalid 1(True):Valid
Default:	0（False）
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets whether to use workpiece coordinate range check, including the upper & lower limit of coordinates.

◆ **N72022   WORKBENCH_LOWER_LIMIT（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	-10000.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the lower limit of workpiece coordinates on each direction when "ENABLE_WORKAREA _CHECK" is valid.

◆ **N72032   WORKBENCH_UPPER_LIMIT（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	-99999.0~99999.0
Default:	10000.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the upper limit of workpiece coordinates in each direction when "ENABLE_WORKAREA _CHECK" is valid.

7.3. Pause

◆ **N73000 PAUSE_DOWN_VOL**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~100000.0
Default:	600.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets the speed for lowering of roll of Z axis when system resumes processing after pause.

◆ **N73001 PAUSE_UP_VOL**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~100000.0
Default:	600.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets the cutter lifting speed of Z axis when pausing occurs.

◆ **N73002 PAUSE_OPTION**

Type:	INT
Unit:	None
Range:	0: To specified cutter lifting amount 1: To specified workpiece coordinate 2: To specified mechanical coordinate
Default:	0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It sets lifting mode of Z-axis When pausing occurs.

◆ **N73003 Z_WPCOOR_ON_PAUSE**

Type:	Float
Unit:	mm（millimeter）
Range:	None
Default:	0
Valid time:	Be valid immediately without restarting the system.
Instruction:	Be valid when setting the parameter N73002 as "1". Appoint the workpiece coordinates of the position of Z-axis when pausing occurs.

◆ **N73004 Z_OFFSET_ON_PAUSE**

Type:	Float
Unit:	mm（millimeter）
Range:	None
Default:	0
Valid time:	Be valid immediately without restarting the system.
Instruction:	Be valid when setting the parameter N73002 as "0". Appoint the cutter lifting height of Z-axis relative to the position when pausing occurs.

◆ **N73006 Z_MACHCOOR_ON_PAUSE**

Type:	Float
Unit:	mm（millimeter）
Range:	None
Default:	0
Valid time:	Be valid immediately without restarting the system.
Instruction:	Be valid when setting the parameter N73002 as "2". Appoint the mechanical coordinate of the position of Z axis when pausing occurs.

7.4. Bkref

◆ **N74012 REFPT_COOR（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	Worktable stroke lower limit (Machine Coordinate) ~Worktable stroke upper limit (Machine Coordinate)
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	The machine coordinates of the mechanical origin is set according to the parameters of [worktable stroke lower limit] and [worktable stroke upper limit].

◆ **N74022 COARSE_LOCATING_DIR（Z）**

Type:	INT
Unit:	None
Range:	1: Positive direction -1: Negative direction
Default:	1
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the motion direction of coarse positioning during backing to the reference point.

◆ **N74032 COARSE_LOCATING_VELO（Z）**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.001~10000.0
Default:	1800.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies the feed speed of coarse positioning stage during backing to the reference point.

◆ **N74052 FINE_LOCATING_DIR（Z）**

Type:	INT
Unit:	None

Range:	1:Positive direction -1:Negative direction
Default:	-1
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies motion direction of the fine positioning during backing to the reference point.

◆ **N74062 FINE_LOCATING_VELO（Z）**

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.001~10000.0
Default:	60.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It can be used for setting the feed speed of fine positioning during backing to the reference point.

◆ **N74082 BACK_DISTANCE（Z）**

Type:	Float
Unit:	mm（millimeter）
Range:	-1000.0~1000.0
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It refers to the additive moving distance after the completion of the fine positioning stage, which can be positive or negative. If the value is "0", it doesn't move, and if the value is not "0", the mechanical origin will be located on the tracking back distance of the corresponding coordinates besides the zero switch.

7.9. Operation others

◆ **N79000 Z-DOWN_VELO_OPTION**

Type:	INT
Unit:	None
Range:	0: Not available 1: Only the z-axis down processing available 2: Available only when the component of z-axis down processing is included
Default:	0
Valid time:	Be valid immediately without restarting the system.
Instruction:	0: not available 1: only the z-axis down processing available 2: available only when the component of z-axis down processing is included

◆ **N79001 Z_DOWN_VELO**

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.0~100000.0
Default:	480.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It specifies Z-axis down feedrate, which is available when Z down feedrate is selecting "1" or "2"

◆ **N79003 SAFE_HEIGHT**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~1000.0
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	It is used for counting corresponding to the origin of workpiece coordinate. The system deems it is safe for the horizontal movement on the specified height. It is generally used to perform the operation of backing to the origin and breakpoint resuming.

◆ **N79112 FIXPT_POS（Z）**

Type:	Float
Unit:	mm (millimeter)
Range:	-99999.0~99999.0
Default:	0.0
Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter is valid under the condition that the [AutoBack to the FixedPoint Valid after Processing] parameter is valid. It is used to indicate the machine coordinates of the position of the fixed point.

8.7. Show Interface

◆ **N87000 showinterfacet**

Type:	INT
Unit:	None
Range:	0: Only NcEditor is opened. 1: Only NcStudio is opened. 2: Both NcEditor and NcStudio are opened.
Default:	2
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the displaying form of the interface.

11.3.2 Waterjet cutting parameters

9.0. Application general

◆ **N90039 PRE_OPENED_HP_DELAY_TIME**

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	0

Valid time:	Be valid immediately without restarting the system.
Instruction:	This parameter can be used to specify the delay time of pre-opening the high pressure valve.

12 Ncstudio™ Appendix

There are many shortcuts in Ncstudio system for the convenience of user. All the involved shortcuts will be listed in this Chapter.

12.1 Shortcut Key Appendix

Most of the shortcut keys which will be introduced below are global shortcut keys. If it is a local one, it will be marked by '★' and the enabled precondition will also be provided.

Single number key (small keyboard is invalid) can be used to choose function section under the current mode (AUTO, MANUAL, REFER).

List the keys according to the sequence of the screen from left to right.

1	**position function section**
2	**offset function section**
3	**program function section**
4	**system function section**
5	**parameter function section**
6	**Diagnose function section**

Single number key (small keyboard is invalid) can be used to choose mode.

7	**auto mode**
8	**manual mode**
9	**reference point mode**

Single letter key be used to can choose function screen under current function section. List the keys according to the sequence of the screen from left to right.

Q	**the first function screen**
W	**the second function screen**
E	**the third function screen**
R	**the fourth function screen**
T	**the fifth function screen**
Y	**the sixth function screen**

Other single letter keys which can be used to choose the button on the current function screen are listed as following.

X	**when related to X-axis**
Y	**when related to Y-axis**

U, I, O, H, J, K, L

Other shortcut keys

F1 ~ F7	**manipulation buttons (valid under function screen)**
★F9	**start (valid under auto mode)**
★F10	**pause (valid under auto mode)**
★F11	**stop (valid auto mode and MDI state)**
F12	**reset**
Ctrl + O	**open and load**
Ctrl + U	**unload**
Ctrl + N	**create processing procedure**
Ctrl + E	**open and edit**
Ctrl + P	**edit current processing procedure**
★Ctrl + F9	**select processing block (valid under auto mode)**
Shift + F1	**open / close water pump**
Shift + F2	**open / close oil pump**
Shift + F3	**open / close high pressure pumps**
Shift + F4	**open / close water valve**
Shift + F5	**open / close abrasive valve**
Shift + F6	**set the current point as workpiece origin**
Shift + F9	**breakpoint resume**