WEIHONG® FIVE AXES WATERJET CUTTER'S MOTION CONTROL SYSTEM

NCSTUDIO™-V10

Users' manual

Dec, 2010

Shanghai Weihong Electronic Technology Co., Ltd.

Preface

Thank you for choosing our products.

This manual will help you acquaint the product of our company and understand the information of system constitution and configuration and usage.

It introduces the process of system installing and various functions. Before using this system, please read this manual in details. You will find it is beneficial for you to use it.

For the hardware and software update continuously, it is possible that the software and the hardware you have received differ from the statement of this manual, for which we apology to you.

Here we list company address, telephone number and Web site. Any questions, please feel free to contact with us. You will be always welcome.

Company: Weihong Electronic Technology Co., Ltd.

Address: Building No.29, lane 2338, Duhui road, Shanghai, China (201108)

Tel: 021-33587550   Fax:021-33587519

Website: Http://www.weihong.com.cn

E-mail: support_weihong@188.com; sales_weihong@188.com

Shenzhen Agency
ADD: 1- 601, Yalan Ting Residential Dist., 7 Shangchuan RD., Bao'an District, Shenzhen
Phone: (00)86-755-29021726

Jinan Agency
ADD: 3-1804, Bldg.3, Baoli Furong Residential Dist., East Minghu RD., Lixia District, Jinan
Phone: (00)86-531-88966551   Mob: (00)86-13793118200; 13658601716

Hefei Agency
ADD: 5-605, Yikang Yuan Residential Dist., Yaohai Dist., Hefei
Phone: (00)86-551-2111322   Mob: (00)86-13866729681; 13866729689

Changzhou Agency
ADD: A-1003, Bldg.3, Xuefu Mingmen Garden, 15 Middle Mingxin Rd., Changzhou
Phone: (00)86-519-81662091

Ningbo Agency
ADD: 2-504, Shidai Jiayuan Residential Dist., North Xingning RD., Ninghai County, Ningbo
Mob: (00)86-18858015451

Wuhan Agency
ADD: 2-1002, Bldg.4, Kangqiao Residential Dist., Hongshan Dist., Wuhan
Mob: (00)86-18963954543

Attention:

1). It is recommended that the machine tool should be surrounded by baffles to prevent workers from waterjet shoot.

2). The wiring direction of servo driver for A/ B axis must be correct.

3). The mechinanical origin of A, B axis is regarded as reference for the corresponding workpiece origin of A, B axis.

4). Please don't plug or pull out the cable connected with the computer when power on.

5). Please don't plug or pull out the card and any other hardware device when power on.

6). In order to guarantee safety and reduce disturbance, the shell of computer and engraving machine must be grounded.

7). When machine is not in use, please shut off the power.

8). Please pull out the BNC connector plugs, when the machine is not used for a period of time.

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 has 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 power supply 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 origin switch connected is normally opened, 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 connection between terminal board and machine tool. This method can greatly shorten the debugging time.

➢ Debugging Signals

Open the computer and run NcStudioTM software. Open [Diagnoses]→[IOPort] window, 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 modification has been finished, exit NcStudio software and restart it. Your modification will become effective. Supply power to the electrical box. Open the computer and run NcStudioTM software. Open [Diagnoses]→[I/O Port] window, you will see many mapping InPorts and OutPorts. Dots which are in front of signals of proceeding start, proceeding stop, tool calibration signal, zero points of the three axes and E-STOP InPort should be in red which indicate all these signals are invalid. If the dots show in green, you must 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 sure the dots in front of the signals above show 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 being pressed and in red when being released. If no change occurs to this dot, please turn to step one to check if LED on terminal board functions normally, check DB37M/F cable and make sure whether wiring conjunction of DB50M/F cable with the card and the terminal board. 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 parameter [pulse equivalent] of manufacturer parameters. The smaller the pulse equivalent is, the higher the resolution is. But if the pulse equivalent is too small, it will influence the maximum feedrate. Users of model machine can generally set the pulse equivalent as 0.001 mm/P (The

corresponding maximum feedrate is 9600 mm/min) or as 0.005mm/P (The corresponding maximum feed speed is 4800mm/min); For users who are not critical about the accuracy, pulse least increment 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^{TM} abiding by right-hand coordinates system.

That is to say, rightward movement of X-axis is the positive direction, and upward movement of Z-axis is the positive direction, while the positive direction of Y-axis is to moving far away from the operator. (If Y-axis movement is the machine tool 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 into function. Set [mechanical reference point] of manufacturer parameters on the basis of sensor installation position of the three axes, execute [back to the 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 mechanical reference point].

➤ Load processing File and Start Machining (or perform manual Machining).

Contents

1 Summarization

The software of NcStudio™, waterjet cutting machines' motion control system, is independently designed by Shanghai Weihong Electronic Technology CO., Ltd. 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 standard style of Windows interface is simple and dependable, and easy to learn.

Apart from functions of automatic mode, manual mode, MDI and backing to the reference point, this numeric control system can also do simulation, forecast processing time, load processing track in advance, dynamic track, breakpoints resume (program perform with skip segment), and so on.

1.1 Software Features

Ncstudio system included the following functions:

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

■ 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 agilely adjusted.

■ Support special user data input function. User can directly input G code by online and immediately execute it.

■ Support processing wizard. To execute functions of milling frame and special lines bottom, user only needs to input several parameters.

■ Support single block mode. User can set the program being selected to execute as [Single Block], which provides very good support for diagnose and fault resume.

■ Support advanced auto function of breakpoint memory and jump perform of program block.

■ Support save/restore workpiece origin. Support backing to the reference point accurately.

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

■ New function for high-smooth speed connection algorithm. In a usual 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 of processing speed. In accordance with speed value, direction and maximum acceleration, and the function of forward prediction, 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 that has eliminated the speed chatter marks left on the processing surface.

■ Particular function of analog display. Through simple operation, you 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 simulate 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. To satisfy the customers' demands in operation;

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

■ Add an enclosed processing file manager. What the user needs to do is to save the processing file into a specified directory; NcStudio™ will administrate these files in an enclosed file manager.

■ Add an enclosed processing file editor. User can introduce processing file into file editor at any time to edit and modify it.

■ Support to display file processing information. By simulation or actual processing, [file processing information] window can help users to count some important information, such as processing time, processing range, the amount of finished workpiece, etc.

■ Support particular [Parameters Auto backup] function. In this window, system can automatically save parameters' setting 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 can load it at any time.

■ Particular [I/O Port] window can do map simulation, test and regulate polarity.   Users can well realize hardware detection, processing supervision, and fault diagnoses. Add PLC module, through which user can control I/O port.

■ Support intelligent prediction; system will analyze situation in the forward or 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%.

■ Support [auto center]; 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 repetition 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 their needs;

■ Support multi-language, currently, English and Chinese are embedded in the software, other languages will be added in demands;

■ Support parameters transfer between different versions of software;

2 System Installation

It is very simple to install Ncstudio™ application. Advanced user or those who have been familiar with the application can quickly refer to the instruction in 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
Mainboard Expansion Slot:	More than 2 slots of PCI

◆ Operation system

Microsoft Windows 2000 Professional

Microsoft Windows XP Professional

2.2 Installation of Ncstudio^{TM}

If there is already 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 override the formal data.

NcStudio™ includes two parts: the software and motion control card. So, 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 the installation of electrical equipments, about which please refer to "Manufacturer Manual" for detailed information.

2.3 Software Setup of NcStudio^{TM}

Software installation can be divided into several steps as following:

➢ Supply power to the computer, start-up the computer, and the system enters into the Windows operation system automatically. Please setup the operation system at first if you have not done so. After Windows Operation System has been installed, don't forget to exit other applications that are running.

➢ Put the setup CD of NcStudio™ system into the CD-ROM. Double click the "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 V10 version, you

can switch over between English and Chinese. 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. Besides, users can also switch language in [System] window. 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 user to save parameters and delete the old version before going on further installation. Picture in Fig. 2-3 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 user whether to shut down the computer (We recommend to close the computer, and then to install the control card). See Fig.2-5.
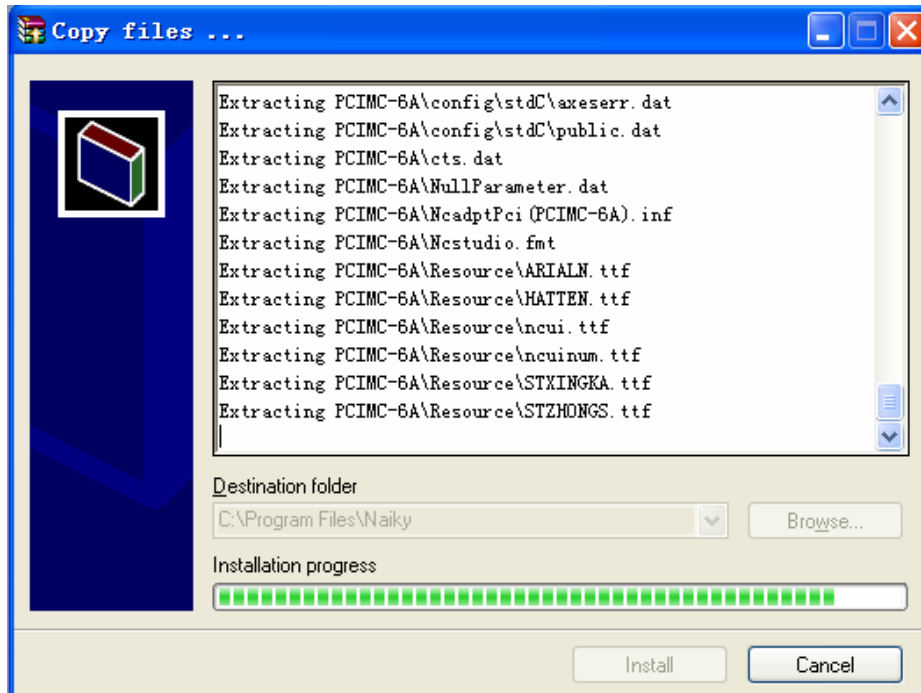
Fig. 2-4 Software Installation Interface

Fig. 2-5 Remind Whether to Shutdown the Computer (Recommend: OK)

Here we have finished the software installation.

2.4 Install NcStudio™ control card & 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 an empty 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 get in well touch with computer baseboard, dependably 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 control card, please insert the peripheral equipments into the related slots or ports and make sure it is firmly fixed without fluttering.

3) Here we end the installation. Restart the computer.

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

Fig. 2-6 Page One of Hardware Update Guide
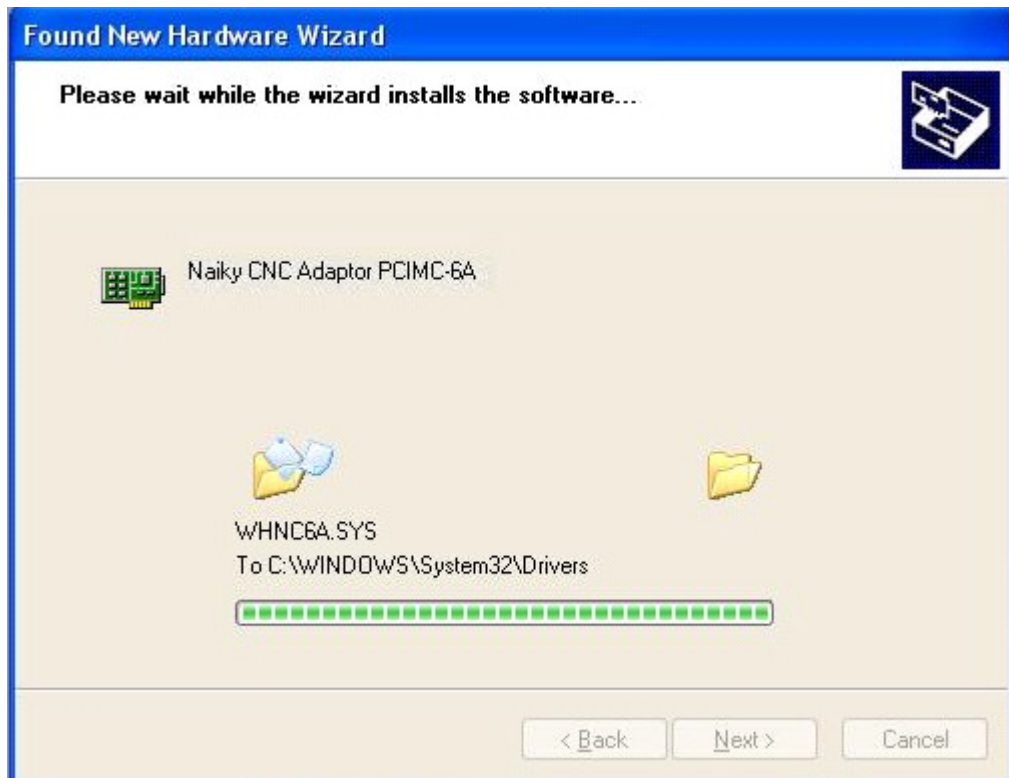
Fig. 2-7 Page Two of Hardware Update Guide

Fig. 2-8 Page Three of Hardware Update Guide

Note:

Figures above may be different with different software version. What we have shown here is only for reference.

2.5 Other Installation problems

If you don't obey the above installation procedure and insert the motion control card before software installation, the computer will remind you to install driver for motion control card installation. The computer will remind you to install driver for motion control card installation. Please cancel the dialog of installing driver, and directly install Ncstudio™ software, restart computer and the operation system will request once again to install driver for the motion control card, at this time you can set according to the hardware guide of the last chapter. After all installation has been finished, user can start to use NcStudio™ software.

2.6 Uninstall NcStudio™ System

NcStudio™ is green software and has the advantage of easy installation and no need to write installation information into registration form. Moreover, to uninstall NcStudio™ system, user only need to delete its document folder: C:\Program Files\Naiky, then, delete [Start] menu→ [Applications] →NcStudio, and the shortcut on the desktop. There will be no residue left on user hard disk.

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, mechanical coordinates system, operation mode, operation state and so on; Unavoidably the user will use these concepts. Therefore, user must 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 (Backing 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 inevitable to back to the reference point. Every time user starts the system, 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. So the processing procedure must be loaded in advance.

◆　Manual Mode

Under manual mode, users can operate with hand appliance, such as hand impulse generator and hand control panel; or users can directly input command to operate the machine. Manual mode is subdivided into: jog, stepping (so-called increment), handwheel, MDI, etc.

➢　Under [Jog] mode, press down the manual button, the machine will keep in motion 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". "Step-size" is divided into four grades: 0.01mm, 0.1mm, 1mm, and 10mm. Moreover, user can customize the step-size. Therefore, user can accurately control the displacement of the machine tool under [Stepping] mode.

➢　Under [Handwheel] mode, users can control the machine tool by handwheel. Each time user boosts ahead the handwheel a lattice, the machine will move forward a stepping 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; it is operation mode and operation state that decides the state of a machine tool.

◆　IDLE State

Idle state is the most common state. Under this state, the machine has no motion to output, and 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

closing waterjet nozzles and cooling 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 (wait instruction), system will enter into PAUSE state and wait for user's further instruction. To suspend the 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 [Auto mode] or [Reset].

◆ LOCK State

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

3.2 Machine's Coordinates System

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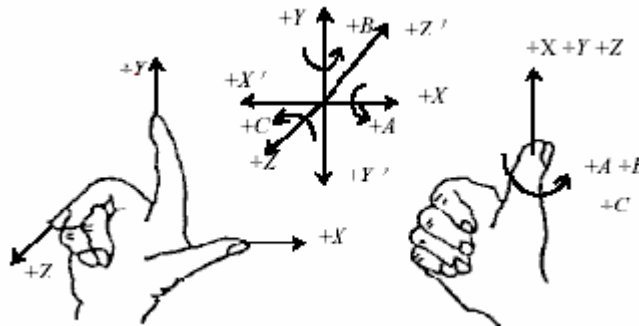

Fig. 3-1 Demonstration for Right Hand Principle

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 cutter 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 face the waterjet nozzle and look 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 Mechanical Coordinates System

Mechanical coordinates system is a set of fixed right-hand coordinates 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 mechanical coordinates system requires the machine being available of function of [back to origin], or this term will only appear in the software.

3.2.2 Workpiece Coordinates System

It is more often to adopt workpiece coordinates system at machining various kinds of workpieces. Generally speaking, the machining position that we mention is a certain point related 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 coordinates system for workpiece machining. That is workpiece coordinates system. Workpiece coordinates system adheres to the right-hand rule as well. Its origin is fixed corresponding to a certain point on the workpiece, which is possibly floating corresponding to the reference point.

4 Operation Interface 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. So, the accuracy and instantaneity of is ensured to the highest extent.

● The system provides powerful visual function, friendly interface, distinct style, and stylish interface pattern. 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 transform, parameters setting and machine stop/ start can be finished by single key, such as single letter, single number, or single function key, which has greatly shortened user's operation time.

● Add simulation operation panel. So many modes to be selected have greatly satisfied users' different requirements.
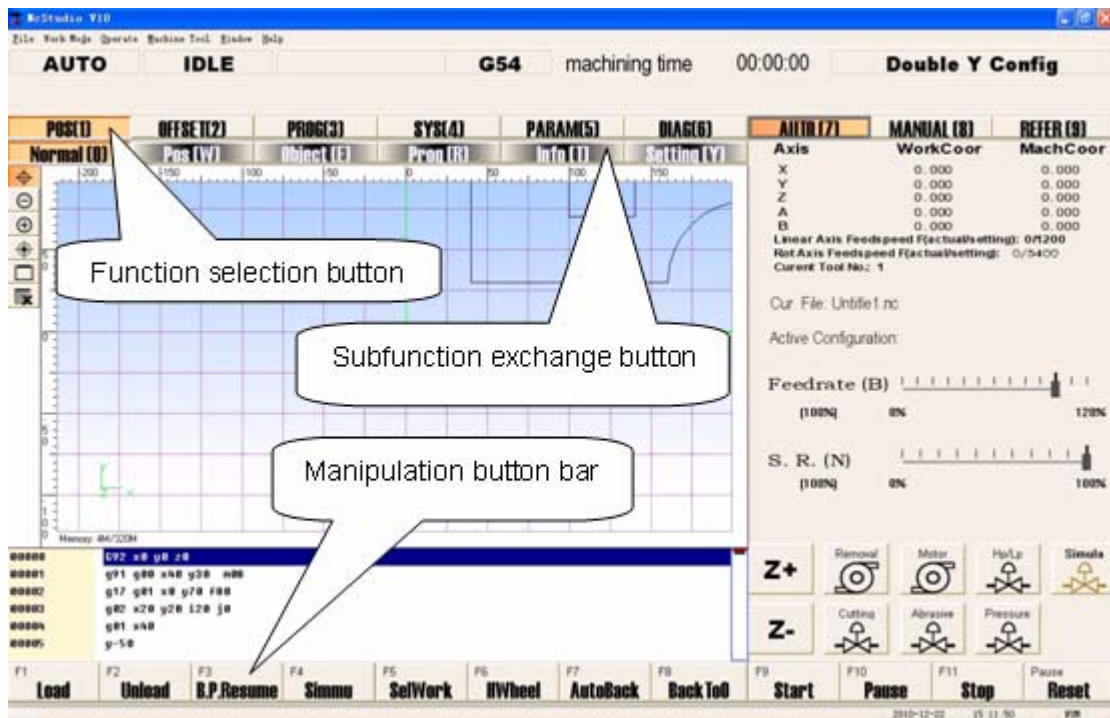
4.1 Integral Interface

Fig. 4-1 The 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 title bar shown in Fig.4-2. It is used to show the name of the application and the name of 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 can accept 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.

The icon in the left of the title bar is a system menu box. It is used to open the window control menu. 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, and 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 be realized by mouse or by keyboard.

On menu bar, there are menu of file, mode, operate, machine tool, windows 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 simultaneously 'Alt' key and hot letter key. 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' can select the item.

◆ Shortcut Key Operation

In pull-down submenu, some items have shortcut keys in 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 direct shortcut key, which can avoid the trouble to enter into multilayer menus.

Some menu option has three dots behind. For example, [File] → [Open and Load…], it indicates that a dialog will automatically pop up. 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 a NC state bar. Please see Fig.4-4.

| AUTO | IDLE | | G54 | machining time | 00:00:00 | Double Y Config |

Fig. 4-4 the NC State Bar

The embedded prompt boxes shown on the NC state bar range from operation mode, operation state, other appended information, and configuration information. In the middle-right, there is the 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 the color 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 a error prompt.

➢ In Black theme interface, if the color of the prompt bar is the same as the color 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 a 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 which leads to operation discontinuance, 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 V10 interface. It is used to display different kinds of function screens, including six powerful function sections and button columns of screen manipulation, such as [Coordinate], [Offset], [Program], [System], [Parameter], and [Diagnose]. Press down function selection 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 subfunction. For example, under the [Offset] function section, there are subfunctions of [workpiece coordinates], [offset], [save & load], etc. Under the [Program] function section, there are screens of [HD List], [ProcessingWizard] and [History record]. To exchange between function screens, please press the [subfunction exchange button]. See Fig.4-7.

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

Fig. 4-7 Function Exchange Button

Note:

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

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

4.7 Manipulation Button Bar

Under the function window is the manipulation button bar, as Fig.4-8.

F1	F2	F3	F4	F5	F6	F7	F8
Load	**Unload**	**B.P.Resume**	**Simmu**	**SelWork**	**HWheel**	**AutoBack**	**BackTo0**

Fig. 4-8 Manipulation Button Column

What it usually shows is the shortcut button relating to the current function screen or the current mode. For instance, under the function screen of [Program]→ [HD List], the manipulation button column has the buttons of [Load], [Unload], [Breakpoint resume], [Simulation], [Selected Machining], [Handwheel Guidance] and [Back To Origin], etc.

Users can choose the manipulation button to do the relevant operation following the content displayed on the function screen.

Note:

Unless the conditions are satisfied, some buttons in the manipulation button column is banned to prevent false operation.

4.8 Status Bar

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

		2010-12-22	15:34:10	NUM

Fig. 4-9 the Status Bar

4.9 Operation Panel

According to the mode differences, the operation panel can be divided into automatic mode, manual mode and reference point mode, as Fig.4-10. Above the operation panel, there is coordinates displaying section where you can see the workpiece coordinates and the machine coordinates, the current feed speed, the current spindle rotation speed and the name of the current loaded file. Under the coordinates displaying section is the panel button area. Buttons inside this panel button area will be different under different mode.

AUTO (7)	MANUAL (8)	REFER (9)
Axis	**WorkCoor**	**MachCoor**
X	0.000	0.000
Y	0.000	0.000
Z	976.659	976.659
A	0.000	0.000
B	0.000	0.000

Linear Axis Feedspeed F(actual/setting): 0/1200
Rot Axis Feedspeed F(actual/setting): 0/5400
Curent Tool No.: 1

Cur. File: Untitle1.nc

Active Configuration:

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

S. R. (N)
[100%]   0%   100%

Z+	Removal	Motor	Hp/Lp	Simula
Z-	Cutting	Abrasive	Pressure	

F9	F10	F11	Pause
Start	**Pause**	**Stop**	**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. The advanced function can be fulfilled that breakpoint resume, selection processing, handwheel guidance.

The picture of auto mode is shown in the Fig.5-1, including operation panel and function panel, which will be introduced in detail separately.
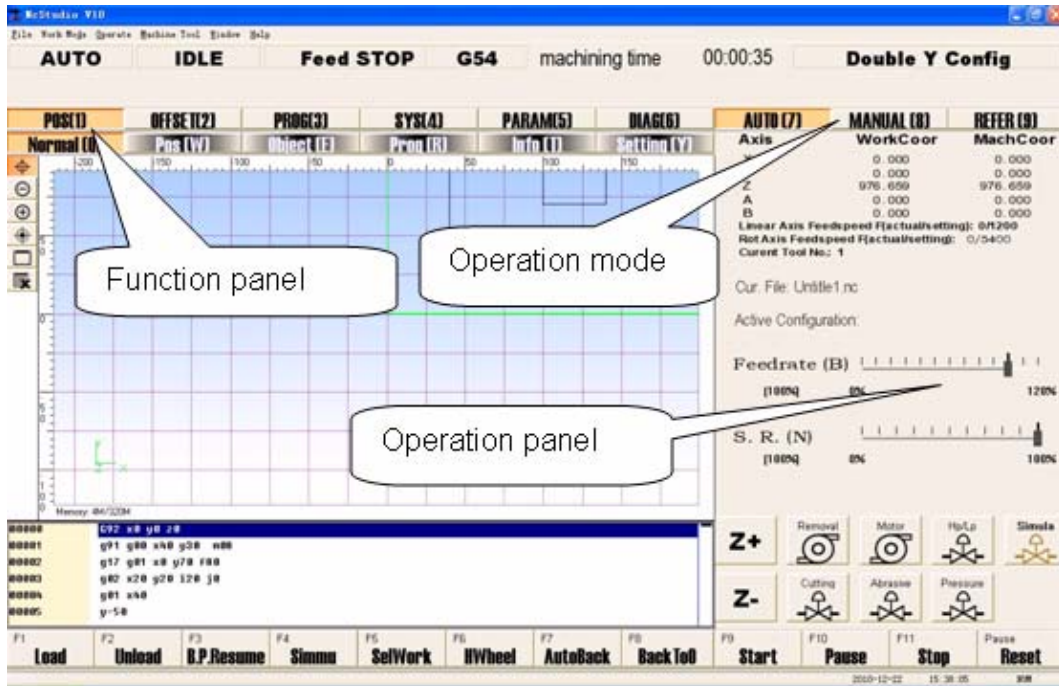
Fig. 5-1 Auto operation mode

5.1 Operation Panel Section

This panel consists of buttons for auto operation of the machine tool, supporting an alternant operation environment for manipulating machine tool automatically. See Fig.5-2.
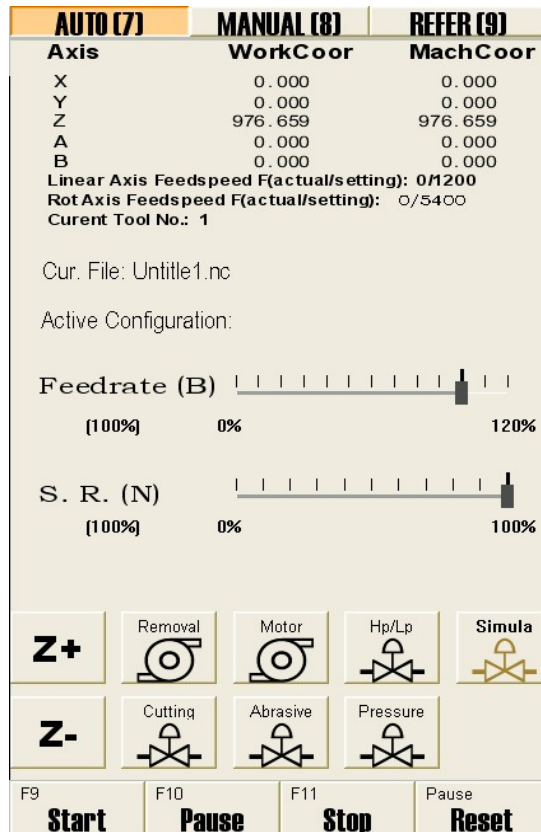
Fig. 5-2 Auto Operation Panel Dialogue

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

Under Auto mode, the button zone includes the following contents:

➢ [Feedrate ratio sliding button]

Shown in Fig.5-3:
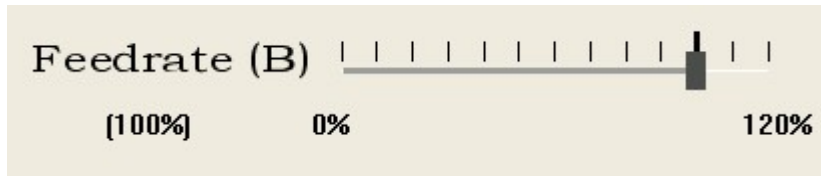
Fig. 5-3 Feed Ratio Sliding Button Dialogue

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

Current feed speed = rated feed value * current feedrate.

The least unit of feedrate is 10% (every grid is 10%). Current feedrate percent is displayed beneath the word "Feedrate". Three ways can be used to revise feedrate percent. Click on the sliding axle; Drag the slide block; Click the slide block to choose it, then press [PageUp], [PageDown] key; Each time press [PageUp] or [PageDown] key, the block moves one case, and the feedrate increases (decreases) 10%. The range of Feedrate percent is 0% ~ 120%.

◆ [Z+], [Z-], [Removal], [H/L], [Pressure], [Motor], [Cutting],[Abrasive], [Simu],

See Fig.5-4.
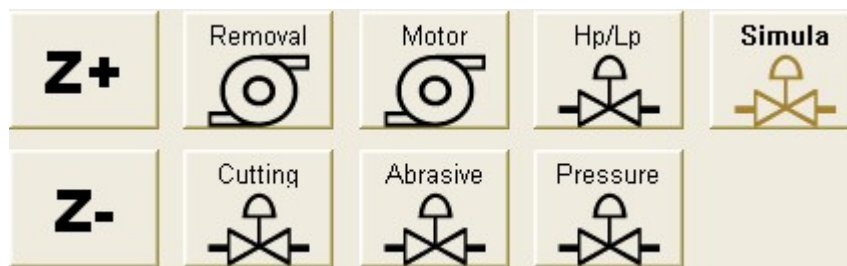
Fig. 5-4 Valve or Pump's Start and Stop Buttons on Operation Panel

These buttons are used to start or stop Removal, Water Pump, Lubricating pump, High Press Valve, Water Valve, and Abrasive Valve.

➢ 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 white. User can distinguish the button's state according to the color of button's background and letter, for example, press down button [Cutting] (Water Valve), its background turns into green, and press the button again, it rises and its background 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 black. User can distinguish the button's state according to the color of button's background and letter, for example, press down button [Cutting] (Water Valve), its background turns into green, and press the button again, it rises and its background restore to black.

◆ [Start]

See: **F9 Start**

Hotkey: F9, only available under AUTO mode.

If there is not loading processing file, the button is disabled;

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

➢ If user has loaded a processing file for the first time, but hasn't executed operations of directly setting or backing to the reference point, system will prompt user to back to the reference point.

The button has two functions:

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

➢ If the system is under "Auto / Pause" state, press the button, system will resume machining from the suspend position and enters "Auto | Run" state. If system is in the state of simulation, the system will start to simulate.

◆ [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 resume machining, press button [Start] or choose other relative menu.

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

Hotkey: F10, only available under AUTO mode.

Note:
System has two ways to enter [Pause] state:
1. When system is executing the 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 G code by users. Press the button, the machine will stop and uplift the cutter to pause all machining tasks, then system enters "Auto | Idle" state. This button is to break off the machining procedure normally.

Hotkey: F11, only available under AUTO mode and MDI (Manual Data Input) 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. [Reset] button terminates machining with E-stop, the state of water pump and high pressure pump will not be changed, there is no deceleration and no cutter 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. 5-5 Position function screen under Auto Mode

5.2.1 Position Function Section

Under AUTO mode, [POS] subfunction section includes as below: Normal, Pos, Track, Prog, Info, and Setting. See Fig 5-6 below:

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

Fig. 5-6 Position subfunction buttons under Auto Mode

◆ Normal Function Screen

See Fig.5-7.

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

```
00000   G92 x0 y0 z0
00001   g91 g00 x40 y30  m08
00002   g17 g01 x0 y70 f80
00003   g02 x20 y20 i20 j0
00004   g01 x40
00005   y-50
```
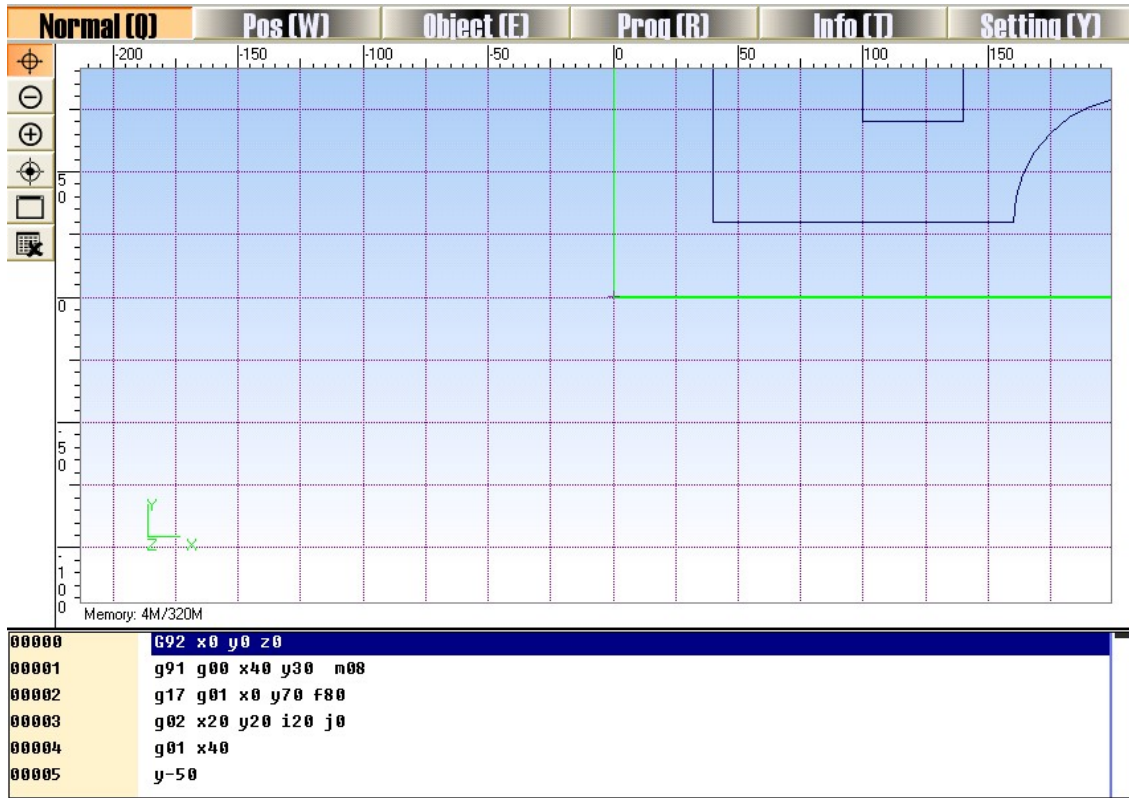
Fig. 5-7 Normal Function Screen

In the processing track window, the tool path displays in real time. User can check tool's track intuitionistically and make sure the machining procedure executed accurately. Besides, in this window, NcStudio supports abundant operating method to give user facilities to look over the processing figure from appropriate position and scaling, such as zoom in, zoom out, etc.

◆ [Icon Button]

On the left of the processing track window, there are icon buttons, see Fig.5-8.

Fig. 5-8 Icon Buttons on the Left of the Processing Track Window

V10 NcStudio adds a tip function for each of these icon buttons. Move the mouse onto the button, a tip will display, which tells user the button's function.

◆ [Move]

Click the icon [Move], move mouse to the display window, the mouse shows as , press down the

mouse's left key, the mouse 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, and then press the direction key to move the whole path towards corresponding direction. 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 their working efficiency if practice perfectly.

◆ [Zoom In] and [Zoom Out]

Click the [Zoom in] or [Zoom out] button on the left of the processing track window, then move the mouse into the display area, when the mouse shows , clicking the left key of the mouse, the figure will be enlarged, when the mouse shows , clicking the left key, the figure will be reduced.

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 processing track window. The current processing point will display in the center of the display window.

Shortcut key: [End] key on the keyboard.

◆ [Adjust to Window Size]

Click button [Adjust to Window Size] on the left of the processing track window. The window will adjust automatically to display all the paths and make the path center in its midpoint. With this method, customer 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 becomes very complex and the temporary folder used to record processing path becomes more and more bulkiness. It will spend a lot of time at redrawing, moving, or revolving the simulation figure. At this time, user needs to clean the tracking window. The actual processing track is cleared after pressing the button. Click the [Clear] button on the left, the simulation figure in the processing track window will be deleted.

Shortcut key: [Delete] key on the keyboard.

◆ [Quick Menu]

Right click the processing track window, and a QuickMenu will pop out. Click the target item to implement the corresponding function. See Fig.5-9.
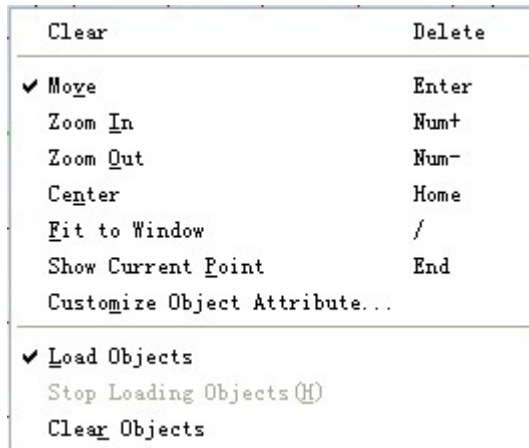

Fig. 5-9 Quick Menu of Processing Track Window

The function of these quick menus will be illustrated separately as following (the above icons with the same function will be omitted)

◆ [Property]

Right click to pop up QuickMenu, choose [Customize Track View], a "Customize Parameter" dialog will appear. See Fig. 5-10 below. User can choose the track color, and decide whether to show the travel range, and so on. For the detailed information, we will introduce as below.
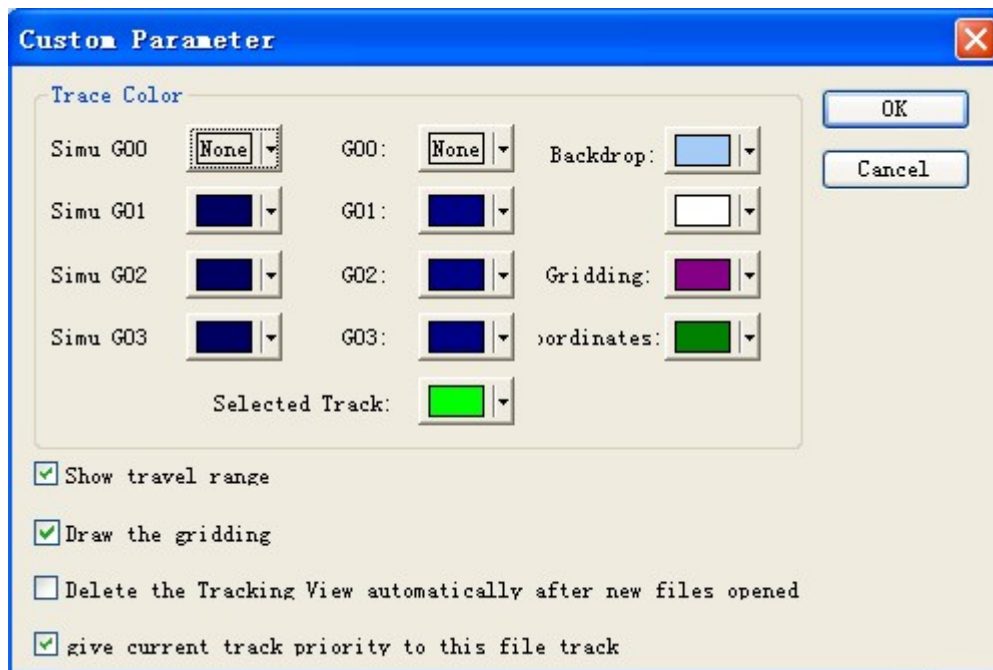
Fig. 5-10 Property Dialogue

Look at the above dialogue, on [Track Color] board, user can set the color of background, gridding, coordinate axis, and the selected track. In V10 NcStudio, we add a new function of loading path in advance, so the path colors are divided into simulation path color and real path color. [Simu G00/G01/G02/G03] is related to simulation path color, and [G00/G01/G02/G03] is corresponding to real path color.

◆ [Show travel range]

This option decides whether to show the simulation worktable in processing track window. Once user selects this item, the simulation worktable will 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.

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

This option is used to decide whether to delete automatically the real path after opening the new processing files. Once user selects this item, the path in processing track window will be deleted automatically.

◆ [Give current track priority to this file track]

After loading processing file and starting machining operation, the actual machining path and simulation path will be all displayed in processing track window. But functions of [Center] and [Adjust to Window Size] are effective only 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.

◆ Processing File Information Display

On the lower part of the Normal Function Screen, there are operation buttons, machine status, 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 beginning auto machining, the current executed instruction will be emphasized with deep blue. See Fig.5-11. The frame in front of the instruction is used to mark the current instruction's position in whole processing file.

Fig. 5-11 The Lower Part of the Normal Function Screen under Auto Mode

● **POS Function Screen**

See Fig. 5-12. This screen is used for looking over the position of tool, including machine coordinate, workpiece coordinate, and remaining distance. When current G code is executed, the distance between current position and destination's position is called remained distance, only available under auto mode.

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

[Position Display]

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

Axis	WorkCoor(G54)
X	0.000
Y	0.000
Z	0.000
A	0.000
B	0.000

MachCoor	Remain
0.000	0.000
0.000	0.000
1306.703	0.000
0.000	0.000
0.000	0.000

Fig. 5-12 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 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. 5-13. [Edit loaded File] button in the lower part of the screen is only available when there is file loaded. Press the button, system will switch to NcEditor, in which user can edit the file.

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

```
00000    G92 x0 y0 z0
00001    g91 g00 x40 y30  m08
00002    g17 g01 x0 y70 f80
00003    g02 x20 y20 i20 j0
00004    g01 x40
00005    y-50
00006    x40
00007    y50
00008    x70
00009    y-40
00010    g03 x-50  y-50 i0 j-50
00011    g01 x-120
00012    g00 x-40 y-30 m30
```

Edit Loaded File (J)
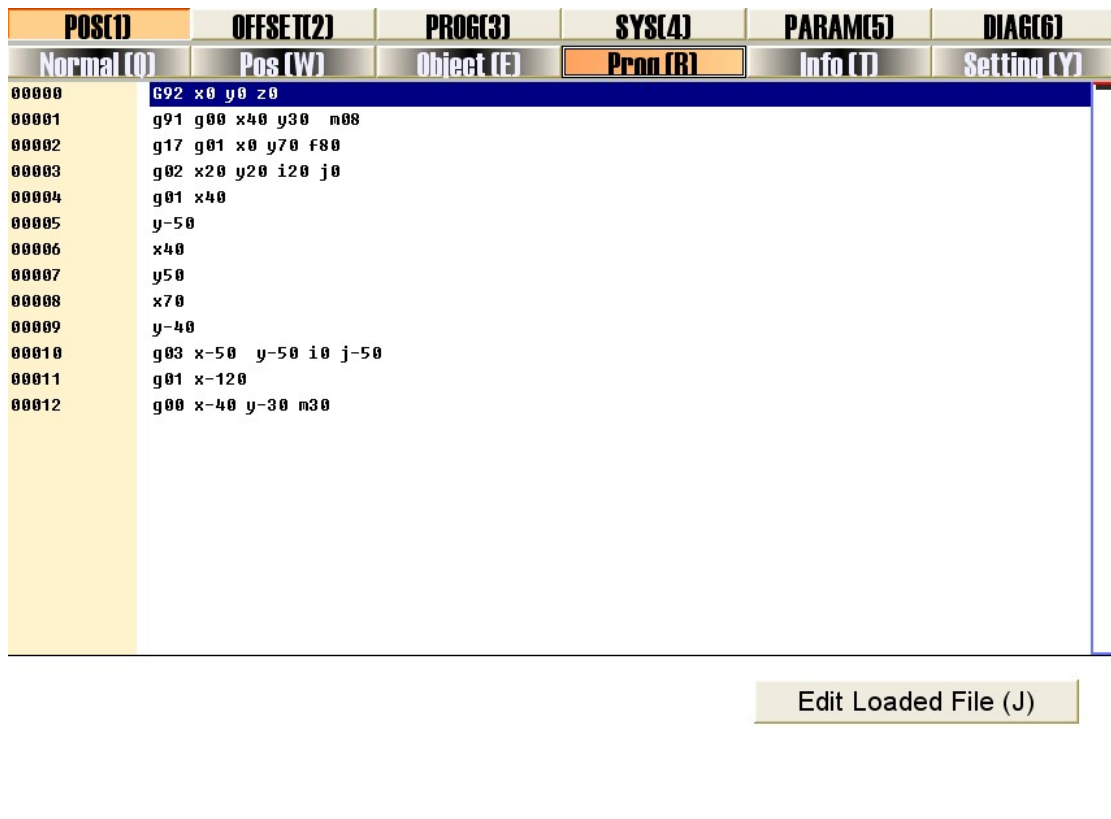
Fig. 5-13 Program Function Screen

◆   Information Function Screen

See Fig.5-14. Information Function Screen is mainly used for displaying various information in machining, such as the maximal processing range, processing time, cutting time, workpieces count, current file line or current executed instruction's position, file length, file path, and incising length, etc. There are some difference between processing time and incising time. Incising time is only the time when tool causes deformation to the workpiece, while processing time includes incising and transverse time. Workpiece count means the times of completing the file and pressing its button will clean the count number. Incising length of the waterjet tool records the whole length of waterjet incision. All the information is totally in user's eyes. Processing time is also displayed in the status bar.
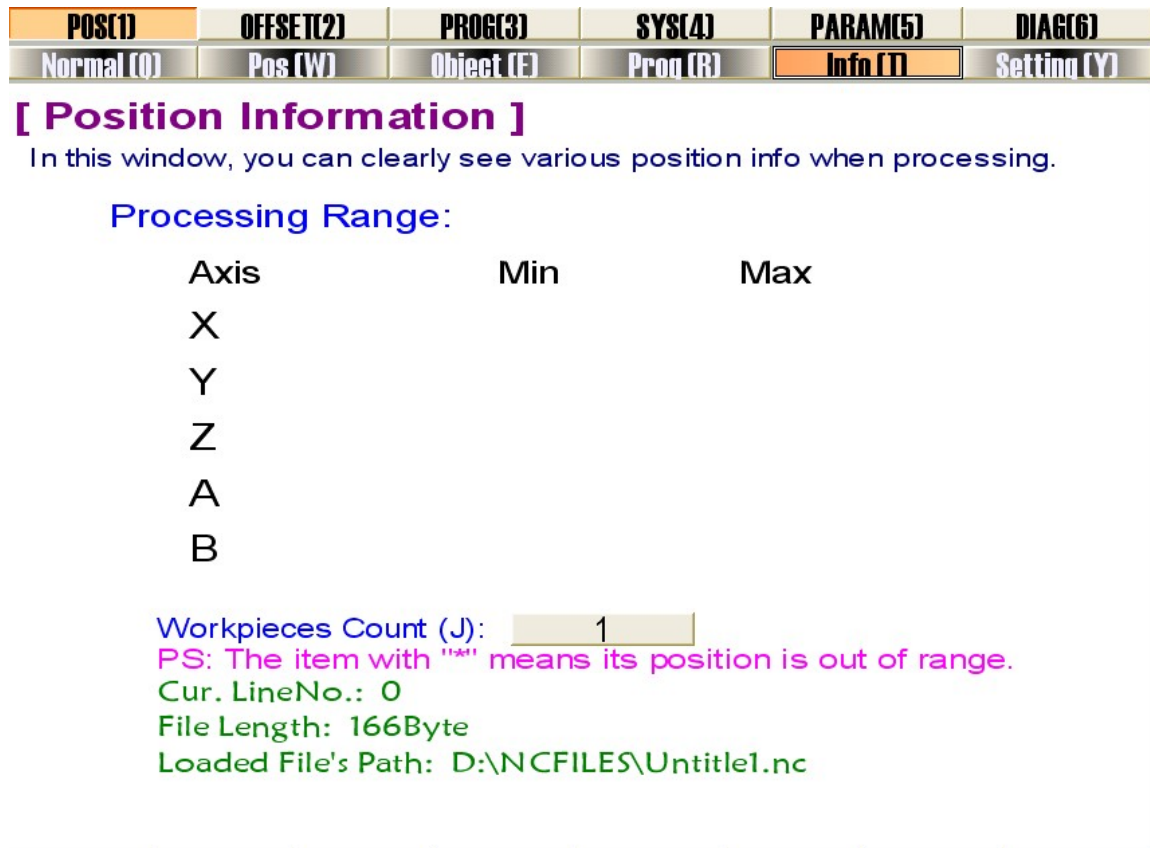
POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Normal (Q)	Pos (W)	Object (E)	Prog (R)	Info (T)	Setting (Y)

[Position Information]

In this window, you can clearly see various position info when processing.

Processing Range:

Axis	Min	Max
X		
Y		
Z		
A		
B		

Workpieces Count (J):　　1

PS: The item with "*" means its position is out of range.
Cur. LineNo.: 0
File Length: 166Byte
Loaded File's Path: D:\NCFILES\Untitle1.nc

Fig. 5-14　Information Function Screen

◆　Setting Function Screen

　　Setting function screen is mainly used for setting parameters under auto mode in common use, such as dry running speed, processing speed, speed mode specified (whether to ignore the feed speed specified by processing file), Cycle times of program, etc. See Fig. 5-15.

Fig. 5-15 Setting Function Screen

◆ [Linear Axis Dry Running], [Rotary Axis Dry Running]

These two items refer to the running speed of linear axis and rotary axis under G00 command, and the range is 1~99999.999. If encroached, system will prompt user. Press the button, system will pop up two Input Boxes, shown in Fig 5-16. It is friendlier than ordinary dialog box, and has functions of copy, paste, expression calculation, even supporting hotkey. User will find it is very convenient, especially when disposing multi-digit number or data needing to calculation.
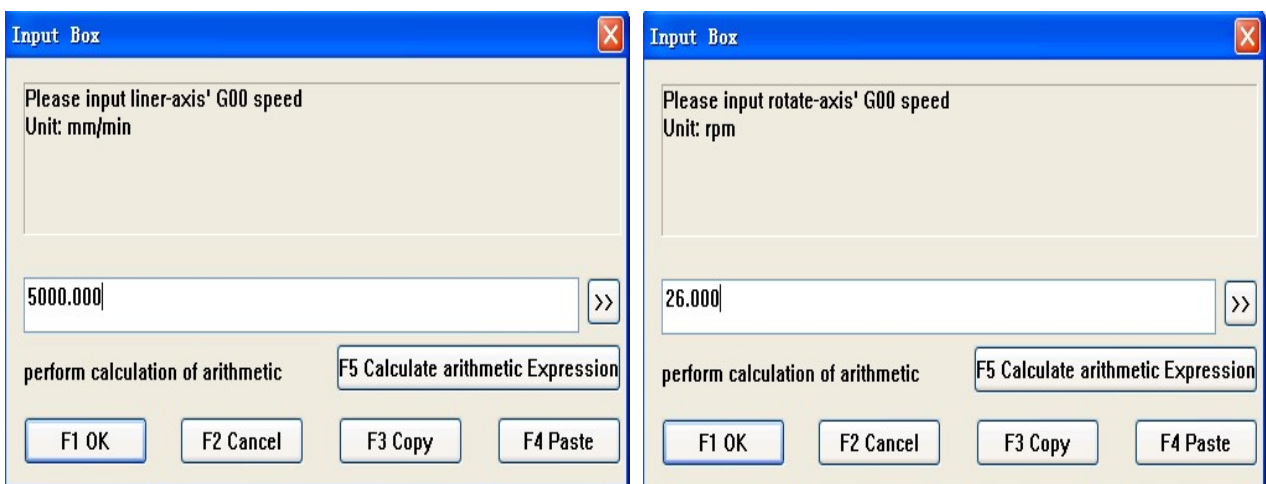
Fig. 5-16 Input Dialogue

◆ [Linear Axis Machining Speed], [Rotary Axis Machining Speed]

These two items refer to the moving speed under G01, G02 and G03 command. Its range is 1~1000. if encroached, system will prompt user.

◆ [Spindle Speed]

This item is used to specify the rotary speed of spindle.

◆ [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. Press the button, system will pop up an input dialogue. User can modify the parameter according to the prompt.
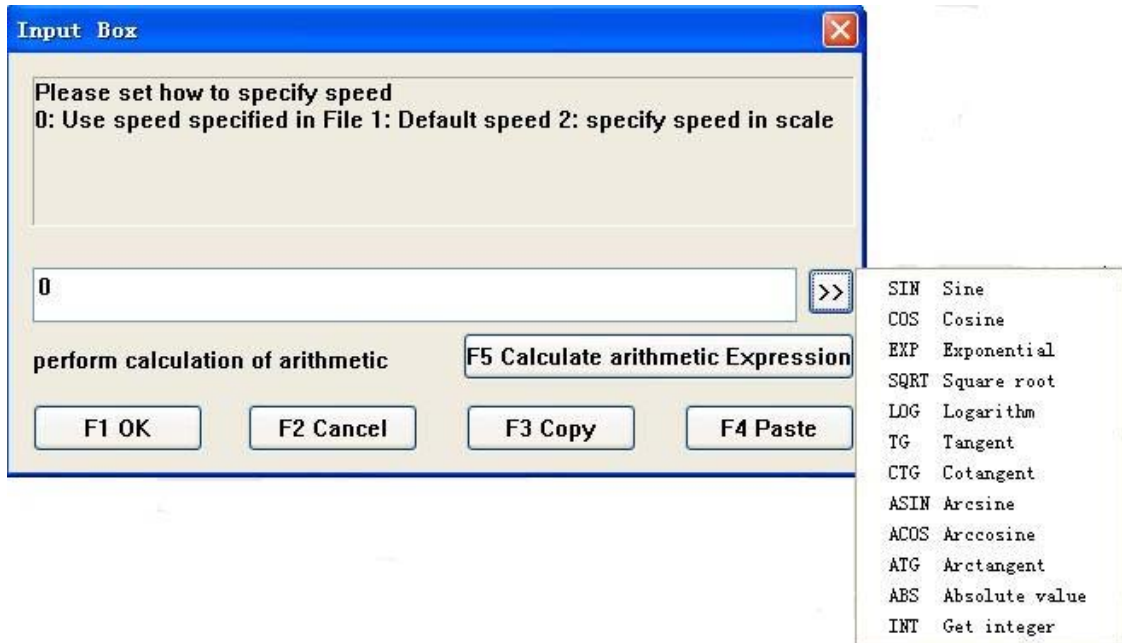
Fig. 5-17 Speed Mode Specified Input Dialogue

◆ [Neglect Spindle Speed in Program]

If this item is set as "Yes", the default spindle speed will be selected, however, if it's "No", the spindle speed in program will be selected rather than default spindle speed.

◆ [Cycle Times of File Processing Task]

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

◆ [Interval 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.

◆ [G00 Feedrate Sliding button]

User can manipulate the cutter traversing speed of G00 instruction by modifying G00 rate. If G00 rate is adjusted to 0%, G00 rate = feedrate percent, and the prompt will be displayed that the "G00FR" is equal to "feed override". The formulation is as following:

Current tool moving speed under G00 instruct = feed speed × machining speed.

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

Current tool moving speed under G00 instruct = dry running speed × G00 rate.

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 on the sliding axle; Drag the slide block; Click the slide block to choose it, then press [PageUp], [PageDown] key; Each time press [PageUp] or [PageDown] key, the block move one case, and the feedrate increases (decreases) 25%. The revised range of G00 ratio is 0%-100%.

● **Manipulation Button Bar**

Under AUTO mode, all of the picture manipulation buttons in the POS function section are the same with each other, shown in Fig. 5-18:

F1	F2	F3	F4	F5	F6	F7	F8
Load	**Unload**	**B.P.Resume**	**Simmu**	**SelWork**	**HWheel**	**AutoBack**	**BackToO**

Fig. 5-18　Manipulation Buttons of Pos Function Section in AUTO Mode

◆　[Load]

Shortcut key: F1, only available under AUTO mode. Pressing down the button will pop up [open and load] dialogue box, user can select needed processing file and load it into parse engine for processing.

◆　[Unload]

Shortcut key: F2, only available under AUTO mode. Its function is to uninstall the loading processing file from NcStudio system.

◆　[Breakpoint resume]

Shortcut key: F3, only available under AUTO mode. This button is not available until power off, E-stop, normal stop by pressing down the stop button. If user can insure accuracy of workpiece coordinate, user can press this button to continue processing. If not, please go back to the origin of mechanical coordinate firstly and restart the processing. The function also can be used under simulation mode.

◆　[Simulation]

Shortcut key: F4, only available under AUTO mode. The function of simulating provides a fast but lifelike simulate processing environment for user. If the processing procedure has been loaded, and the current operating mode is "IDLE", pressing down this button and then "start" button, system will carry out simulating at high speed automatically from the first section. Simulation is just like demo function in CNC system, but it's superior to demo function. Running under the mode of simulating, the system will not drive the machine tool to do the relative actions but to show the processing trace of the cutter in high speed in the trace window. By simulating, the user sees the moving form of the machine tool advance to avoid damaging the machine tool for the reason of mistakes in programming the processing procedure. By simulating, user can also know the additional information. User can look over that information in INFOR function screen under the POS function section.

◆　[Select processing block]

Shortcut key: F5, only available under AUTO mode. This function is used for program perform with skip segment. Pressing down the button, system will pop up the [execute (advanced options)] dialogue box, shown in Fig. 5-19, User can execute any block in processing program by inputting the concrete start line number and the end line number for current procedure. When the user selects [file start] or [file end], the detailed position of the file will be disabled. That function can also be used for simulating.

Fig. 5-19 Select Processing Block Dialogue Box

◆ [Handwheel guidance]

Shortcut key: F6, only available under AUTO mode. Pressing down this button, user can control machine tool only by MPG, and then the buttons on operating panel are invalid.

◆ [Auto Back]

Hotkey: F7, it is available only under AUTO mode. When user found certain block of workpiece is not processed completely, firstly, stop the machine tool and press this button. After finishing "Auto Back" procedure, users can click breakpoint resume to continue machining. This function is only available to current object when there are multiple objects.

Before using "Auto Back" function, the parameter of "Exact Breakpoint Resume" should be set as "True", or the starting point of "Auto Back" will begin with the end point of current track rather than exact current position. When the machining type is not "NCE" format, clicking this button will pop up a dialogue, and the picture will be neglected here.

◆ [Back to origin]

Shortcut key: F8, only available under auto mode. Pressing down the button, the system will control tool to go back to the workpiece origin. Please pay attention to the differences among workpiece origin, fixed point and mechanical origin.

5.2.2 Offset Function Section

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

● Workpiece Coordinate Window

As Fig.5-20, this section includes two parts, [Workpiece Coordinate Setting] and [Auxiliary Function].

◆ [Workpiece Coordinate Setting]

As Fig.5-20, in this screen, user can set the current workpiece coordinate. Click the button in the screen, system will pop up the coordinate input dialog box, please enter the value and click OK button, the system will adjust the coordinate automatically (No actual position movement). And the screen will give prompt to user the moving distance and direction of the waterjet nozzle.
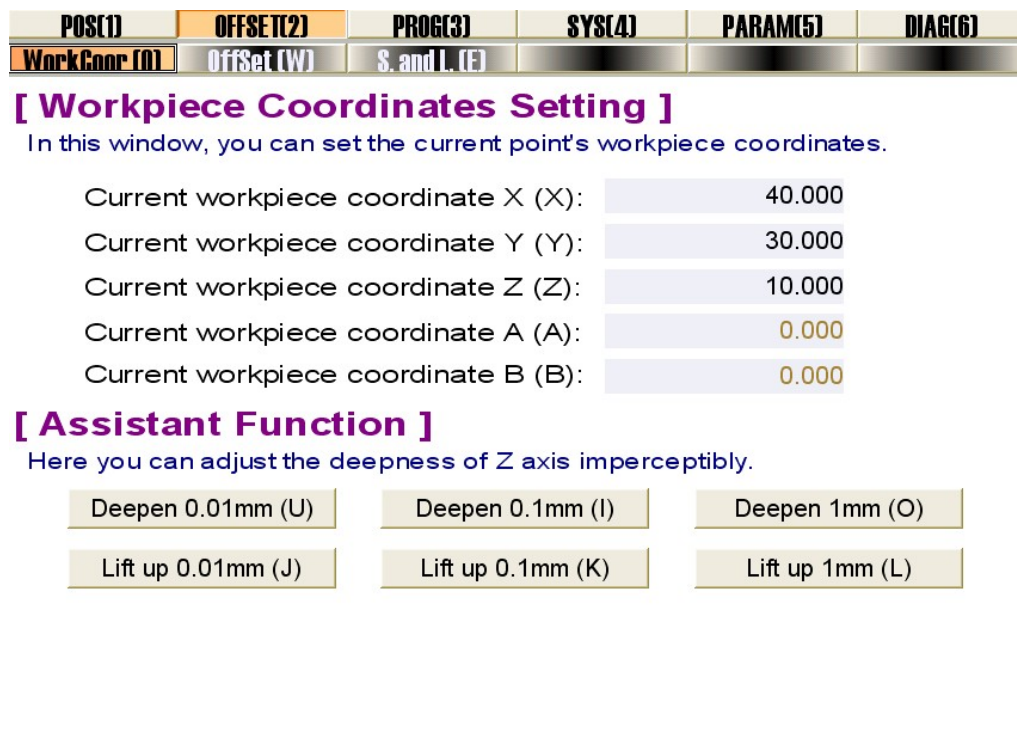
Fig. 5-20 Function Screen of Workpiece Coordinate Setting

The input dialogue is as Fig.5-21.
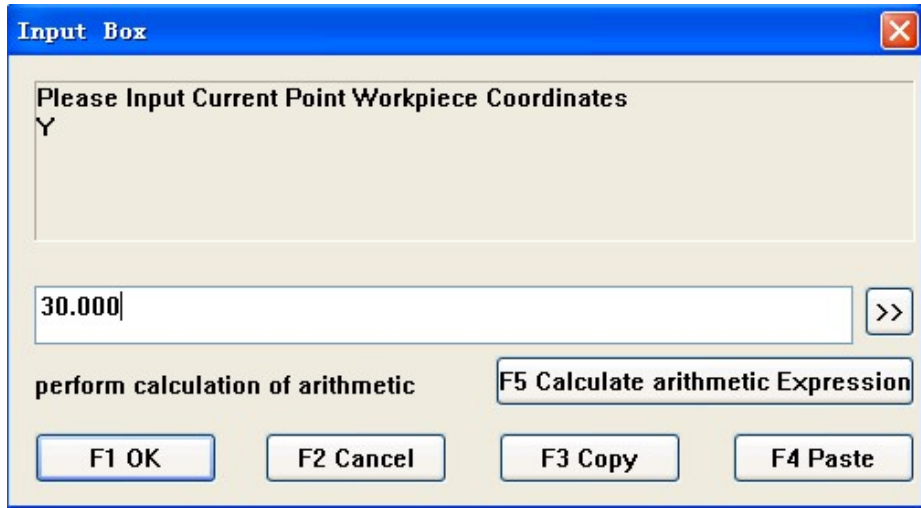
Fig. 5-21 "Input Box" for Workpiece Coordinate Adjustment

◆ [Auxiliary Function]

[Auxiliary function] can adjust Z-axis slightly, and the adjustment range is 0.01mm, 0.1mm, 1mm.

● Offset function section

As Fig.5-22, this picture is used to set the external offset value, and the offset value between mechanical coordinates system and workpiece coordinates system of G54, G55, G56, G57, G58, G59 commands.
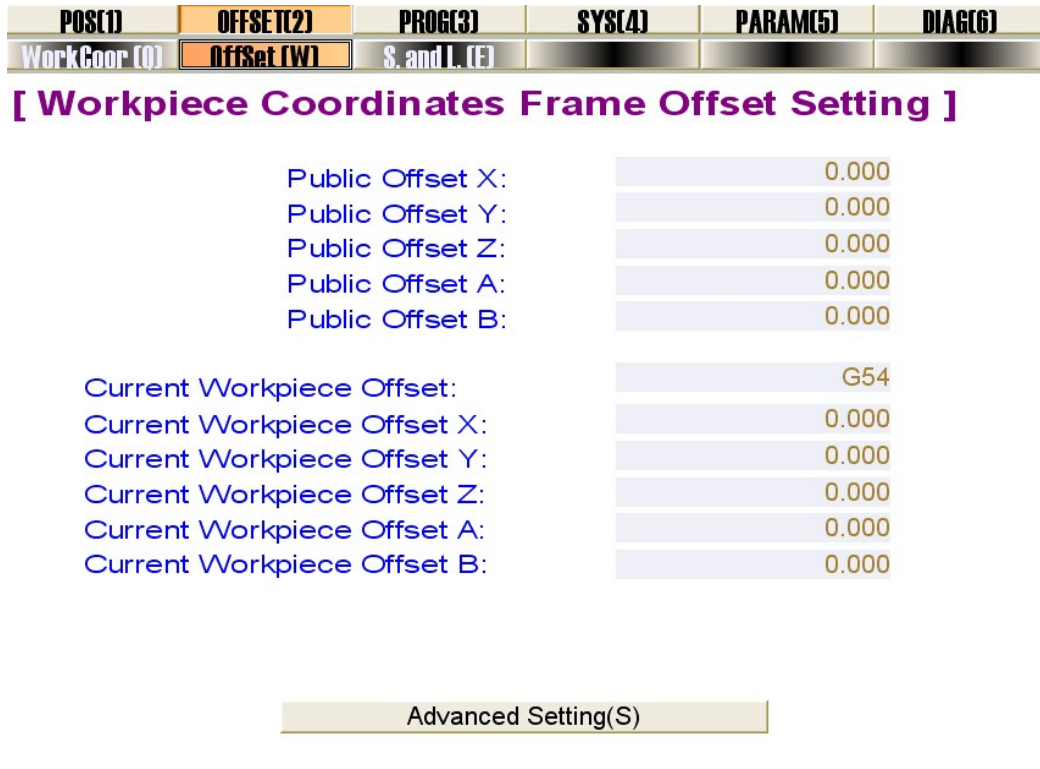
Fig. 5-22 workpiece coordinates system offset setting function screen

Final offset value between workpiece coordinates system and mechanical coordinates system = External offset value + G instruction offset value (including G54、G55、G56、G57、G58、G59) + waterjet nozzle offset value

Click the advanced setting in the dialogue, the offset setting can be specified. See Fig.5-23.
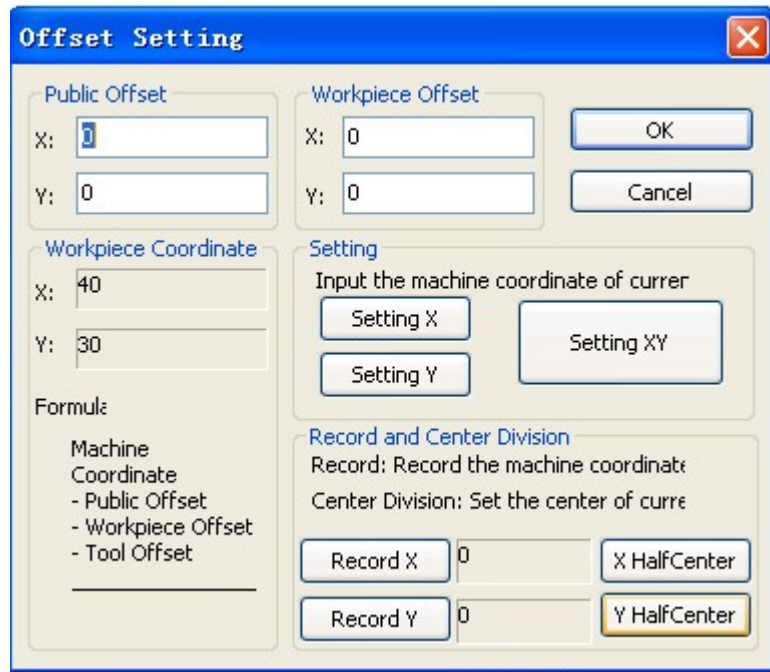
Fig. 5-23 offset setting picture

● S. and L. Function Screen

As Fig 5-24, user can save and load the offset value of workpiece coordinate in this screen. There are 10 saving groups. Each group can supply group no., name of the processing file loaded, and save time. When saving or loading the record, press the corresponding button, system will pop up corresponding dialog box to prompt user.

Fig. 5-24 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 [workpiece coordinate] function under [Offset] section is as shown in Fig. 5-25.

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

Fig. 5-25   manipulation button in [Workpiece Coordinate], [Offset] function screen

◆ [X Clear]

Shortcut Key: F1. Click this button, system will pop up the dialog box to prompt whether to modify the coordinate value to ensure that the workpiece coordinate value of current X-axis is Zero.

◆ [Y Clear]

Shortcut Key: F2. Click this button, system will pop up the dialog box to prompt whether to modify the coordinate value to ensure that workpiece coordinate value of current Y-axis is Zero.

◆ [Z Clear]

Shortcut Key: F3. Click this button, system will pop up the dialog box to prompt whether to modify the coordinate value to ensure that workpiece coordinate value of current Z-axis is Zero.

◆ [XY Clear]

Shortcut Key: F4. Click this button, system will pop up the dialog box to prompt whether to modify the coordinate value to ensure that workpiece coordinate values of both current Y-axis and X-axis are Zero.
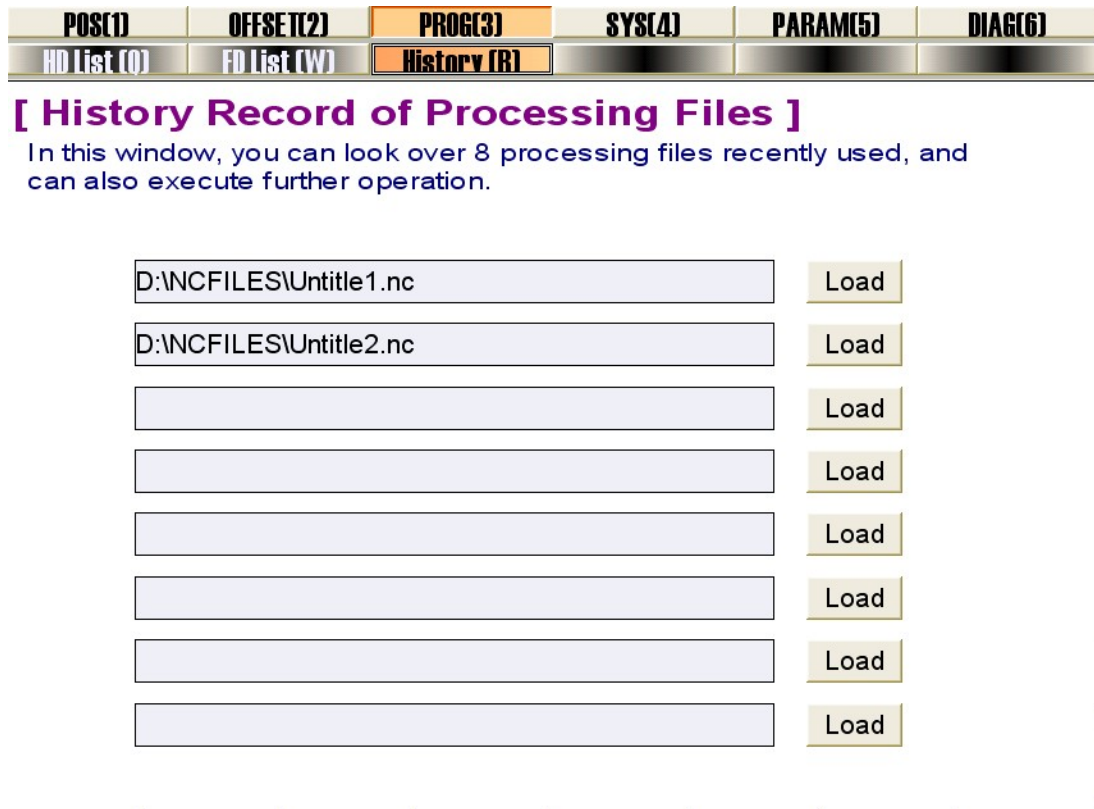
◆ [Back To 0]

Shortcut Key: F8. Press the button, system will control tool to back to the workpiece origin automatically. Note to distinguish workpiece origin, fixed point and mechanical origin.

The manipulation button under [Workpiece coordinate] solely holds "Back to Origin" function, being equal to the "Back to Origin" function under [Offset] section.

The picture manipulation button in [S. and L.] function screen is as shown in Fig. 5-26.

F1 Save	F2 Load	F3	F4	F5	F6	F7	F8
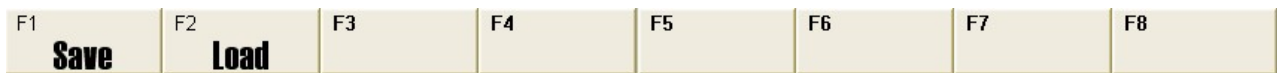
Fig. 5-26 Manipulation button in [S. and L.] function screen

◆ [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 Program Function Section

These screens are mainly used to deal with the related machining operations. [PROG] function section includes [HD List], [FD List] (Floppy Disk List), [ProcessingWiz], and [History] function screen.

● **[Hard Disk List] Function Section**

In this screen, user can lookup processing files under default file path on hard disk, and also can execute functions, such as load, edit, delete, rename, etc. Moreover, user can build a new file under default file path D:\ NCFILES and compile it.

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

HD List (Q)	FD List (W)	History (R)			

File Name	Size	Last Revised Time
Untitle1.nc	166	2010-05-25 14:24
Untitle2.nc	58	2010-05-25 13:51
Untitle3.nc	207	2010-05-27 10:39

```
Current File floder:    D:\NCFILES\
File Name: Untitle1.nc
Size:166          When The File Modified Last Time: 2010-05-25  14:24
File Head:        G92 x0 y0 z0
                  g91 g00 x40 y30  m08
                  g17 g01 x0 y70 f80
                  g02 x20 y20 i20 i0
```

Fig. 5-27 [HD List] Function Screen

The upper part of the screen is a file list; it displays all of the files under the path "D:\ NCFILES". User can save the processing files in common use into this place. It is more convenient for use.

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

Note:

User can save the machining file under the folder "D:\ NCFILES", and then the files can be loaded automatically in the Hard disk list.

● **[Floppy Disk List] Function Section**

In this screen, user can load the files of floppy dick or the soft disk. For instance, the file in the picture is in the floppy disk (Drive: "I"), user can execute a series of operations, such as load, copy, select driver, and safely remove. The lower part of the screen is displaying the basic attribute of selected files, such as file name, size, modified time, etc.
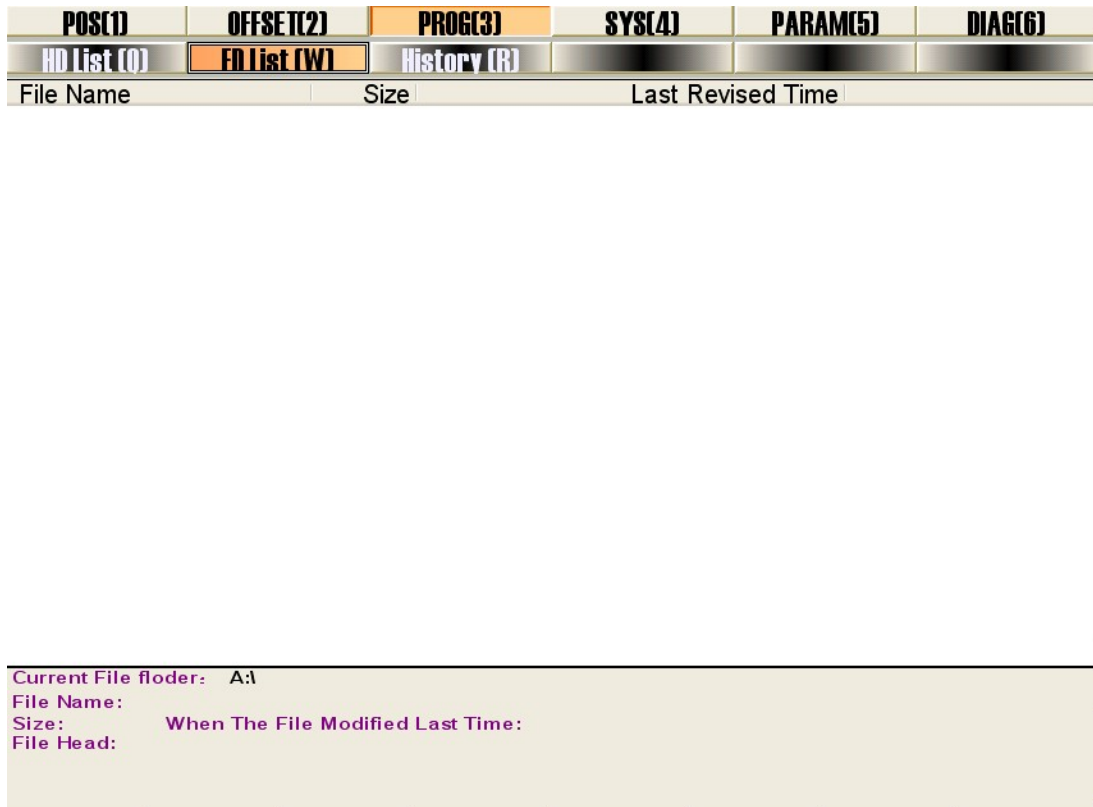
Fig. 5-28 [FD List] Function Screen

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

The lower part of the screen is a prompt frame. After 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.

● [History Function Screen]

History function screen is as shown in Fig. 5-29. If user wants to load one recent fie, please turn to this screen, press [Load] button to load a file. In this screen, it records all the files used recently. User only needs to load the selected file to processing.

| POS(1) | OFFSET(2) | PROG(3) | SYS(4) | PARAM(5) | DIAG(6) |
| HD List (Q) | FD List (W) | 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.

D:\NCFILES\Untitle1.nc	Load
D:\NCFILES\Untitle2.nc	Load
	Load
	Load
	Load
	Load
	Load
	Load

Fig. 5-29 File Record History Screen

◆ Manipulation Button Bar

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

The picture manipulation button in [HD List] function screen is as shown in Fig. 5-30.

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

Fig. 5-30 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; press the button, system will load the file into parse engine automatically. At the same time, the loading progress will be displayed in above information column. After loading process finished, 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; press the button, the loading progress will be displayed in above information column. After loading process finished, system will 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; press the button, a prompt dialog will pop up to remind user whether to delete the file selected. See Fig. 5-31.

Fig. 5-31 Be Sue to Delete Dialog

Note:

If the selected file has already been loaded, or edited, or machined, 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, system will built a new file ".nc" in the list under path "D:\NCFILES". Select the file, Press [Rename] or [Edit] button, user can rename and edit it.

◆ [Rename]

Hotkey: F5, only available in the [HD List] function screen of PROG function section. Select a file in the list; press the button, system will pop up a dialog to prompt user to input the file's name. After inputing the file name, please click [OK] button, see Fig.5-32.
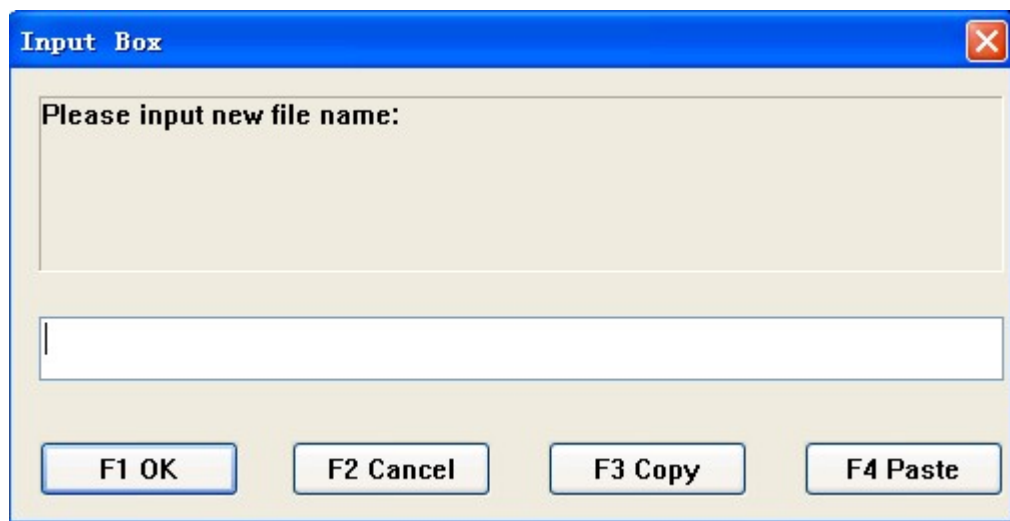
Fig. 5-32 Rename Dialog in [HD List] of [PROG] Function Section

The picture manipulation button in [FD List] function screen is as shown in Fig. 5-33.

Fig. 5-33 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; press the button, system will load the file into parse engine automatically. At the same time, the loading progress will be displayed in above information column. After loading process finished, 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; press the button, system will copy this file to the default path on hard disk.

◆ [Select Drive]

Hotkey: F3, only available in the [FD List] function screen of PROG function section. If there are more than one external memorizers, user can select one of them with the 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 remove the memorizer safely. However, if the current file is under machining or the loaded file is in the external memorizer, the button is invalid.

5.2.4 System Function Section

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

● [Configuration Function Screen]

The screen is as shown in Fig. 5-34. It displays the configuration supported by NcStudio, Press [Select] button to select the corresponding configuration according to actual situation.
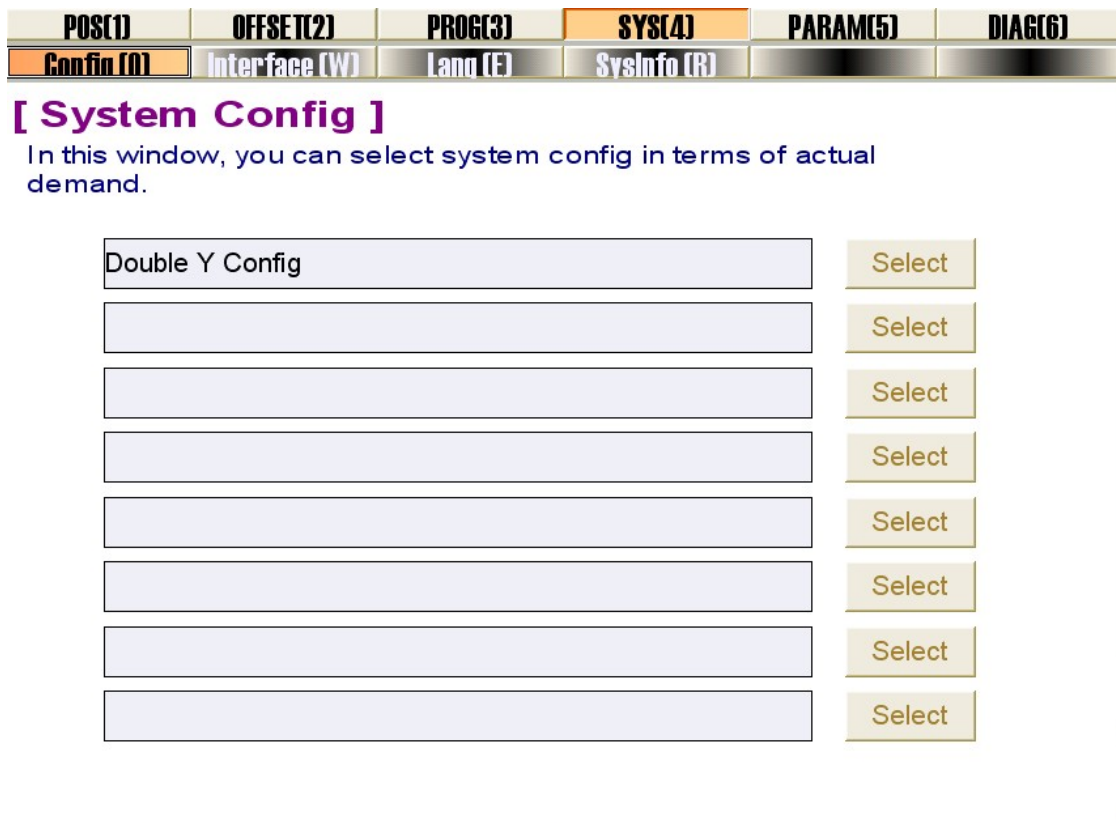
Fig. 5-34 Configuration Selection Function Screen

● **[Interface Function Screen]**

The screen is as shown in Fig. 5-35. 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 of them.
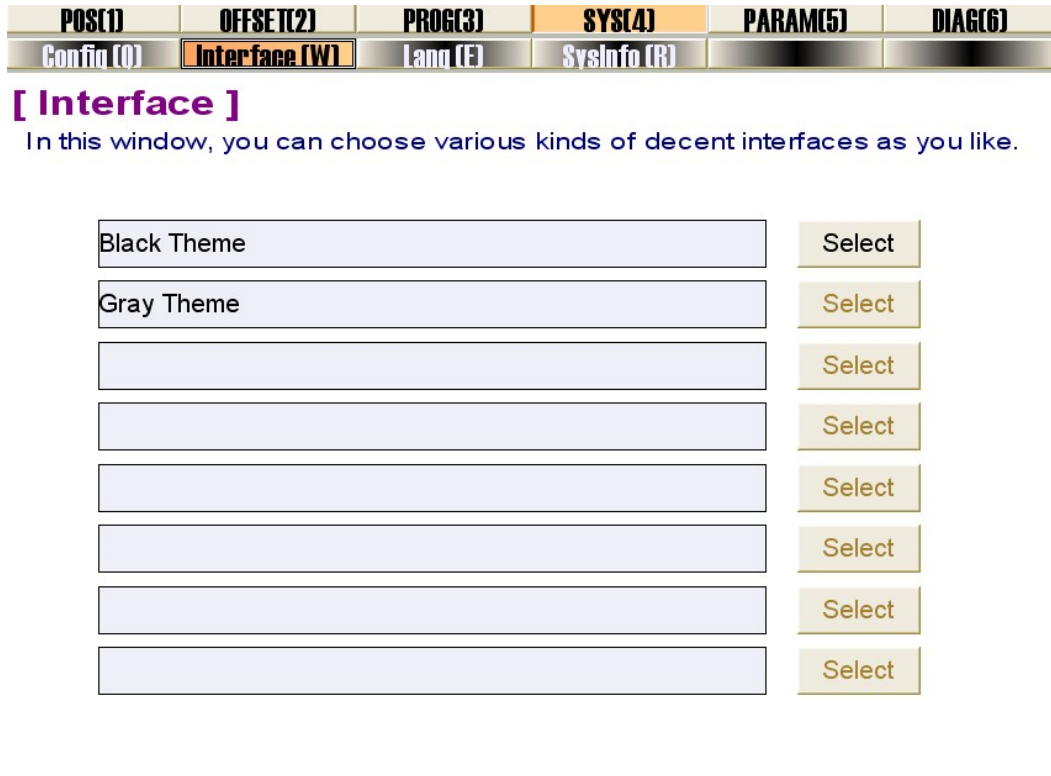
Fig. 5-35 Interface Function Screen

● **[Language Function Screen]**

As Fig. 5-36, 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.
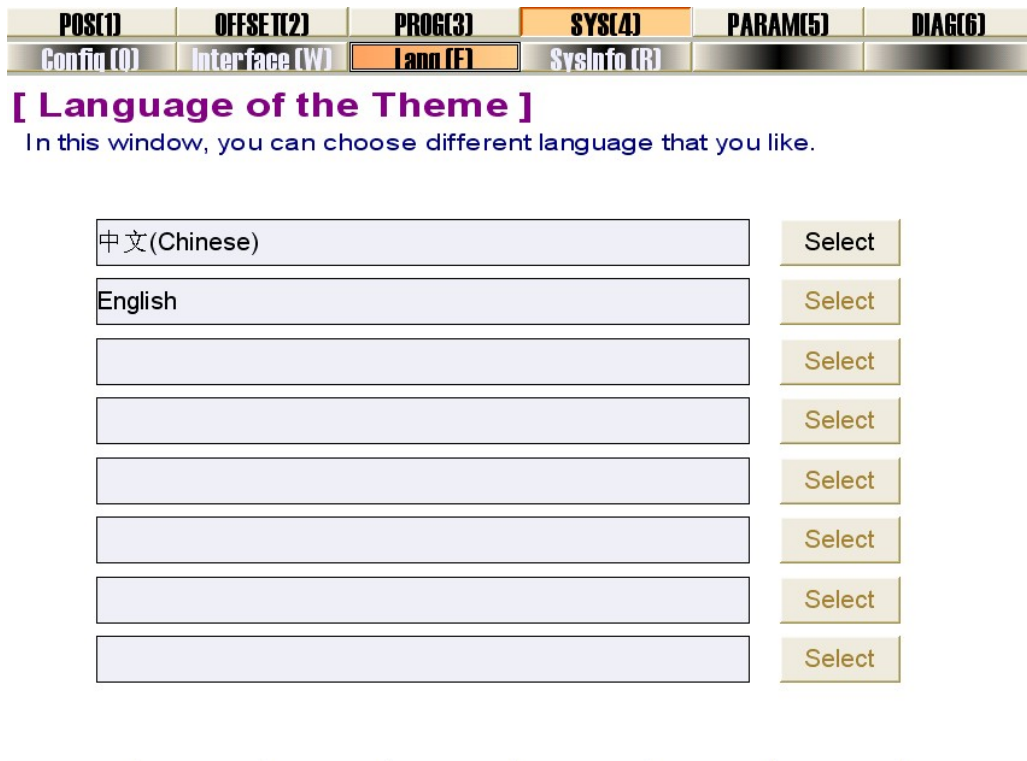
Fig. 5-36 Language Function Screen

● **[System Information Function Screen]**

As Fig. 5-37, user can look over the information of NcStudio system. It includes version, company, name, hardware information, card no., remaining time and self-test information.

Fig. 5-37 SysInfo Function Screen

◆ Manipulation Button Bar

The manipulation button in [SysInfo] screen of [SYS] function section is shown in Fig. 5-38.

F1 Register	F2	F3	F4	F5	F6	F7	F8

Fig. 5-38 Manipulation Button Column under [SysInfo] Function Screen

◆ [Hardware Register]

Hotkey: F1, only available in the [SysInfo] function screen of [SYS] function section. Press the button, system will pop up a "REGISTER" dialog, as shown in Fig.5-39. For new user, infill the register code and press [OK] to accomplish the registration.
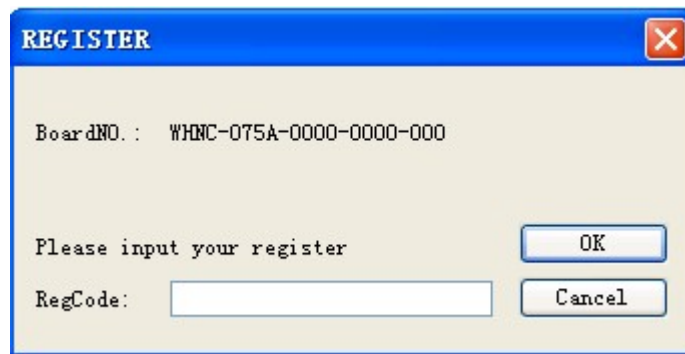
Fig. 5-39 Hardware Register Dialog

5.2.5 Parameter Function Section

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

● [General Parameter Function Screen]

As Fig. 5-40, there are three sub-screens in it. According to the grade of jurisdiction, user can access different sub-screen to set parameters.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Param (Q)	WaterJet (W)	Backup (E)	AutoBackup (R)		

	No.	Name	Value	Unit	Effectin...	De
Operation	**1.0.Axes general**					
		AXIS_DIR			Reboot	The
Axes	N10000	X	1		Reboot	
	N10001	Y	1		Reboot	
Controller	N10002	Z	-1		Reboot	
	N10003	A	1		Reboot	
	N10004	B	-1		Reboot	
Program		AXIS_PULSE_FACTOR			Reboot	The
	N10010	X	0.01	mm/p	Reboot	
	N10011	Y	0.01	mm/p	Reboot	
View	N10012	Z	0.01	mm/p	Reboot	
	N10013	A	0.01	deg/p	Reboot	
Other	N10014	B	0.01	deg/p	Reboot	
		WORKBENCH_LOWER_L...			Reboot	The
	N10020	X	0	mm	Reboot	
All	N10021	Y	0	mm	Reboot	
	N10022	Z	10000	mm	Reboot	

Parameter Name： AXIS_DIR
Value: Unit：
Valid Time：Reboot
Description： The direction of axes. The value available are: 1: Positive, -1: Negative.

F1 Operator	F2 Manufactr	F3 Developr	F4	F5	F6	F7	F8

Fig. 5-40 Operators' Parameter Function Screen

In V10 NcStudio, user can look up 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 it introduces all parameters according to their serial number.

Parameters in V10 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). We offer the type name of the parameters according to the first and second digits.

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

◆ [Operation Parameters]

In [Operation] parameters, it includes parameters which are related to the specific operations. Press the button, and you will see the corresponding parameter list, including N7.1 (Manual), N7.2 (Auto), N7.3 (Suspended), and N7.9 (other operations). In manufacture parameters, it also include N8.3 (Open Door Parameter)

◆ [Axes Parameters]

In [Axes] parameters, it includes parameters which are related to the axis. Press the button, and you will see the corresponding parameter list, including N1.0 (Feed axis routine), N6.4 (Speed Control). Besides N1.0 and N6.4, in manufacturer parameters, it also includes N1.2 (Compensation), and N1.3 (Velocity/ Acceleration Limit), and N7.2 (Parameters that are related to axes under Auto mode).

◆ [Controller Parameters]

In [Controller] parameters, it includes parameters which are related to the connection and setting of external controllers. Press the button, and you will see the corresponding parameter list, including N5.2

(Handwheel). Besides N5.2, in manufacturer parameters, it also includes N5.0 (Controller General), N5.1(Input & Output), N5.3 (Operation Panel). Because only manufacturer and developer have the right to revise the parameter, so in operator parameter, this item is vacant.

◆ [Program Parameters]

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

◆ [Interface Parameters]

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

◆ [Other Parameters]

Under this parameter section, operators' parameters include parameter of N4.29 (Cylinder), but in manufacturers' parameter, apart from N4.29, it also includes N8.6 (Diagnosis Interface).

◆ [All Parameters]

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

User can only access and set parameters belonged to his privilege. Class and Parameters available are shown below. Some parameters only legal user has the right to look over.

◆ Table 1 Privilege and Parameters Available

Class	Parameter Available
Operator	Operator's parameters
Manufacturer	Operator Parameters, Manufacturer Parameters
Developer	Operator Parameters, Manufacturer Parameters, and Developer Parameters

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

● **Waterjet Function Screen**

The screen is as shown in Fig. 5-41. You will see the special parameters of waterjet machine. After pressing the grey frame behind "open valve sequence", user can specify the sequence of the water valve, pressure and abrasive valve. And also the closing sequence of valves can be specified by clicking the grey frame behind "close valve sequence". User can reset them according to their own needs. For detailed information about the parameters please refer to chapter 9.

Fig. 5-41 Waterjet Parameter List

● **[Parameter Backup Function Screen]**

As Fig. 5-42, user can save or restore the parameters' setting in this screen, and avoid the trouble of setting parameters repeatedly. After modifying the parameter, please switch to this screen, press [Save] button, system will pop up an "Input Box", input the name, and click [OK] button. NcStudio supplies 8 backup items. After the parameters have been saved, user can load it momentarily according to the name of saved item.

Fig. 5-42   Parameters Backup Screen

● **[Auto Backup Function Screen]**

The screen is as shown in Fig 5-43. NcStudio supports function of auto parameters backup. If user forgets to save the parameters setting, please switch to this screen, press [Load] button, it is easy to restore all parameters setting of any time from ex-factory date to last time.

POS(1)	OFFSET(2)	PROG(3)	SYS(4)	PARAM(5)	DIAG(6)
Param (O)	Water Jet (W)	Backup (E)	AutoBackup (R)		

[Parameters Auto BackUp]

In this window, you can restore the parameters setting of system auto backup.

LastTime	Load
Yesterday	Load
The Day Before Yesterday	Load
5 Days Before	Load
10 Days Before	Load
Leave Factory Before	Load
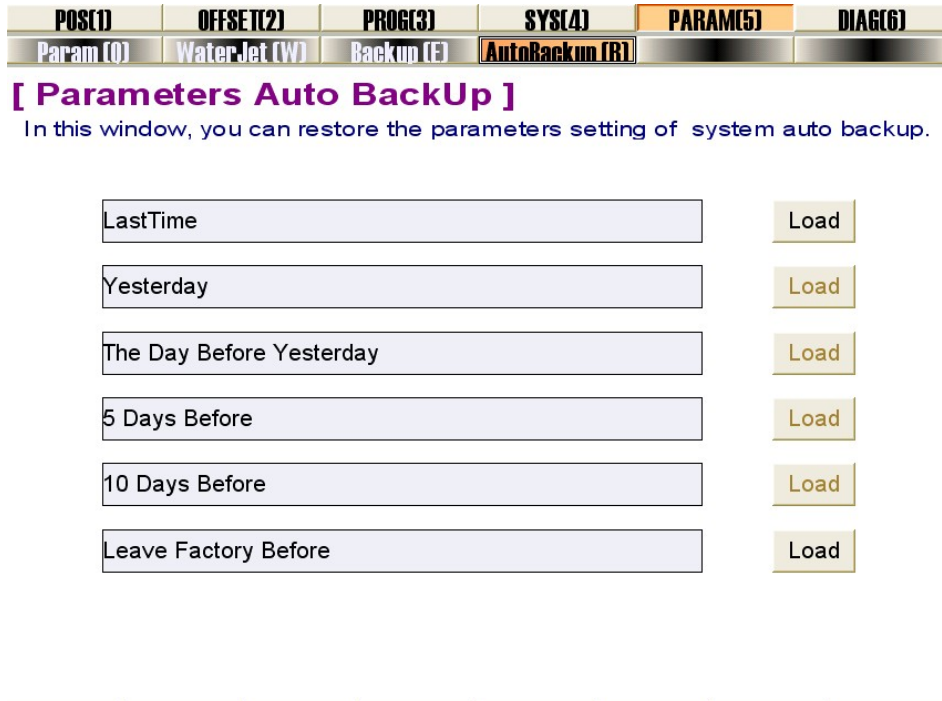
Fig. 5-43 Parameters Auto Backup Screen

5.2.6 Diagnosis Function Section

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

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

● **[Log Function Screen]**

As Fig. 5-44, this screen records users' important operations and system events. In this screen, user can browse not only current log information but also history log information.

Fig. 5-44 Log Function Screen

◆ [I/O Port Function Screen]

As 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 diagnosis.

Fig. 5-45 I/O Port Function Screen

◆ [PLC Function Screen]

As Fig. 5-46, it is used for PLC debug and serves especially system debug person and advanced user. For detailed information please refer to Programme Manual.

POS[1]	OFFSET[2]	PROG[3]	SYS[4]	PARAM[5]	DIAG[6]

Log [Q]	IOPort [W]	PLC [F]			

```
in:  0123456789012345678901234567890 1        spec: 0123456789012345678901234567890 1
00000 ..............................               90000 11111.........................
00100 ..............................               90100 ...........................
                                                    90200 ...........................
out:  0123456789012345678901234567890 1           90300 ...........................
10000 ........11111..............11..1             90400 ...........................
10100 ..............................               90500 ...........................
                                                    90600 ...........................
temp: 0123456789012345678901234567890 1           90700 ...........................
20000 ..............................               90800 ...........................
20100 ..............................               90900 ...........................

---------------------------------- timers -----------------------------------
00[      0,       ] [      0,        ] [      0,         ] [      0,         ]
04[      0,       ] [   1000,    1000] [   1000,     1000] [   1000,     1000]
08[   1000,   1000] [   1000,    1000] [   1000,     1000] [      0,         ]
0C[      0,       ] [      0,        ] [      0,         ] [      0,         ]
---------------------------------- counter ----------------------------------
00[      0,       ] [      0,        ] [      0,         ] [      0,         ]
04[      0,       ] [      0,        ] [      0,         ] [      0,         ]
08[      0,       ] [      0,        ] [      0,         ] [      0,         ]
0C[      0,       ] [      0,        ] [      0,         ] [      0,         ]
```

Fig. 5-46 PLC Function Screen

◆ Manipulation Button Bar

The subfunction screen will vary under [Diagnosis] Function Section, which will be introduced separately.

The screen of Manipulation Button Bar in [Log] function is shown as Fig.5-47.

F1 Refresh	F2 Clear	F3 Info	F4 Warning	F5 Error	F6 System	F7 Nowadays	F8

Fig. 5-47 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 synchronization between the list and system.

◆ [Clean]

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

Pressing the button will clean all the log information.

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

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

Press down one needful button, you will see the corresponding information. They are normal information, warning information, error information, system information, and current information in turn. If you needn't display some type of information, let the right button rise. When [Nowadays] button rises, system will display all the log records.

Manipulation Button Bar under [Hard Disk I/Oport] function is shown as Fig.5-48.

Fig. 5-48 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. It is mainly used for simulating signal for 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.

◆ [Cancel Test]

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

◆ [Cancel All]

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

◆ [Convert Polarity]

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

◆ [Display All]

Hotkey: F6, only available in [I/O Port] function screen. Ports not used are hided in this screen, press down the button, you will see all the I/O ports; let the button rises, ports not used will be hided again.

◆ [Port Attribute]

Hotkey: F7, only available in [I/O Port] function screen. It is used to revise the attribute of the selected port. Press the button, system will pop up a dialog, as shown below. User can set sampling interval, and decide whether to filter and enable, and can also revise port's name and description.
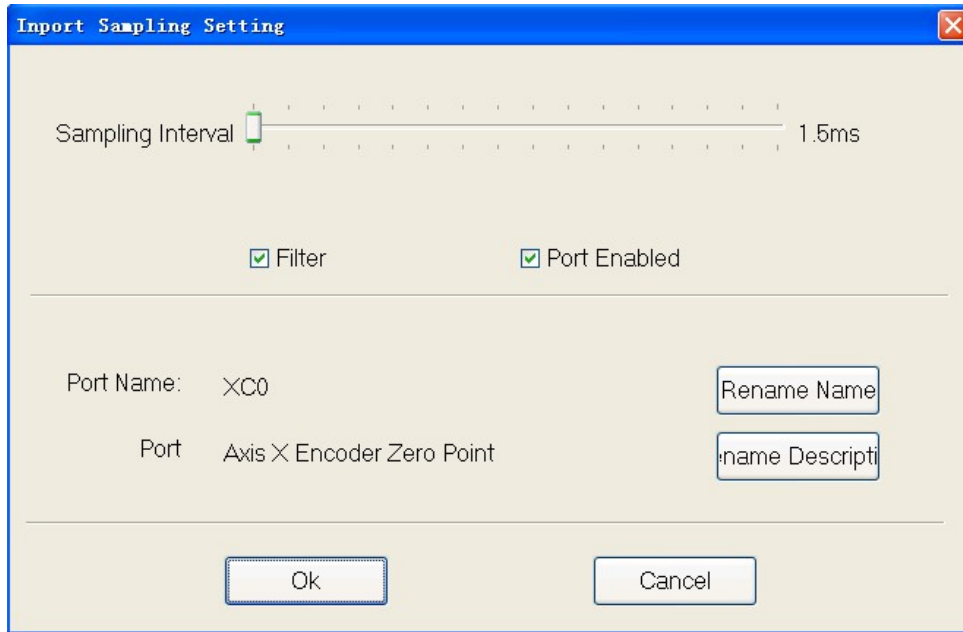
Fig. 5-49 Port Attribute Setting Dialog

6 Manual Mode of Ncstudio™

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

Manual mode is shown in the Fig.6-1.It includes two panels-operation panel and function panel, which will be introduced separately.
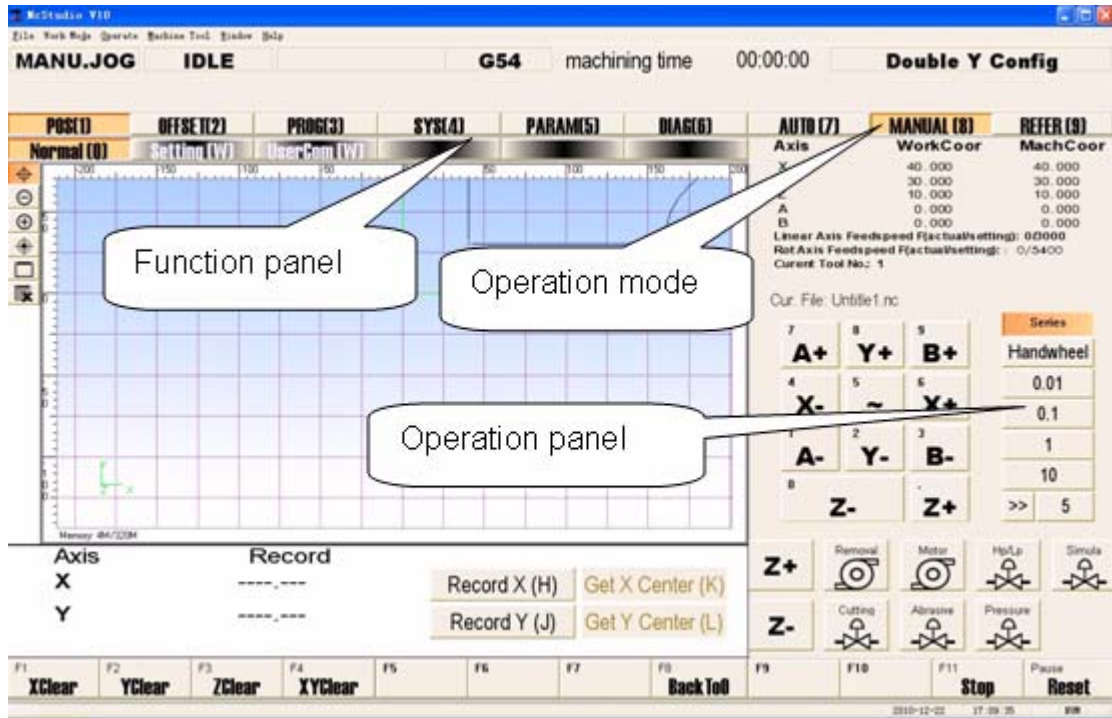
Fig. 6-1 Manual operation mode screen

6.1 Operation Panel Section

On manual operation panel, there are many buttons for controlling the machine manually. See Fig. 6-2. It provides user with a manual alternant operation environment.
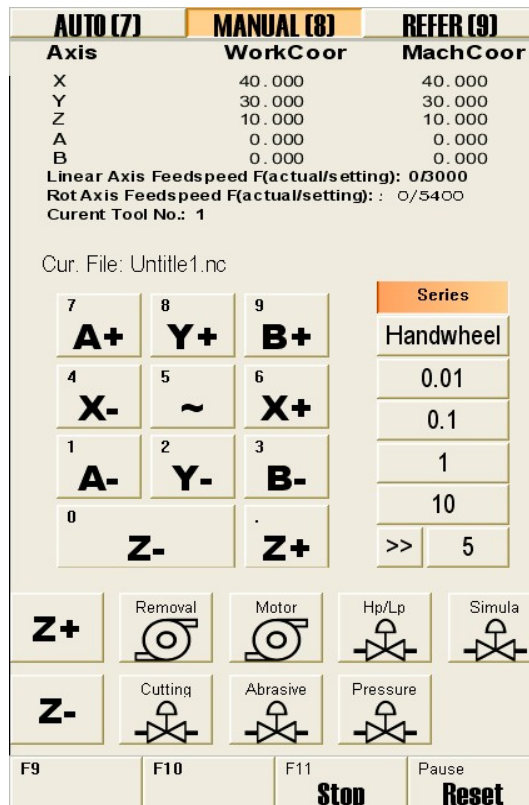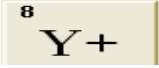
Fig. 6-2 Manual Operation Panel

On the top of the panel, there is a coordinate display zone, which is used to display the current coordinate of the waterjet, current feed speed 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:

They are corresponding to moving direction of - X and + X axis; Hotkey: number key 4 and 6 of mini-keyboard.

◆ [Y+], [Y-] manual button, shown as:

They are corresponding to moving direction of - Y and + Y axis; Hotkey: number key 2 and 8 of mini-keyboard.

◆ [~] manual button, shown as:

Acceleration button; only available under "Series" state; correspond to number key "5" of the mini-keyboard; it is used together with X, Y direction keys, with combination key: "5" and direction number key to accelerate waterjet speed, such as "5"+"8" (accelerate along +Y direction).

◆ [Series], [Handwheel], [0.01], [0.1], [1], [10], [>>], see Fig.6-3.
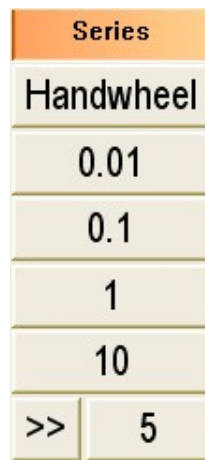
Fig. 6-3 Manual Moving Buttons

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

Jog 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 of 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 from the color of the title bar.

When manual screen is active, number lock is not 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, user must install 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 (0.01, 0.1, 1, 10, >>): It is also called step-manner or increment-manner. Steeping manner is different from the jog-manner, it can control the machine's feed distance more accurately. Every time trigger the direction key, the machine will move a distance of a specified step-size. Press down one button of [0.01], [0.1], [1], [10], each time trigger the direction key, system will move 0.01 mm, 0.1 mm, 1 mm, and 10 mm respectively. Moreover, V10 NcStudio also supports custom step-size. Press down [>>] button, system will pop up an "Input Box" dialog, input a step-size that you want, click [OK], value of the custom step-size will be displayed behind [>>] button; press the custom stop-size button, every time triggering the direction key will make the machine tool move a distance of a custom step-size.

◆ [Stop], [Reset]

These two functions have been already introduced under auto mode, which will not be introduced again.

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, 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.
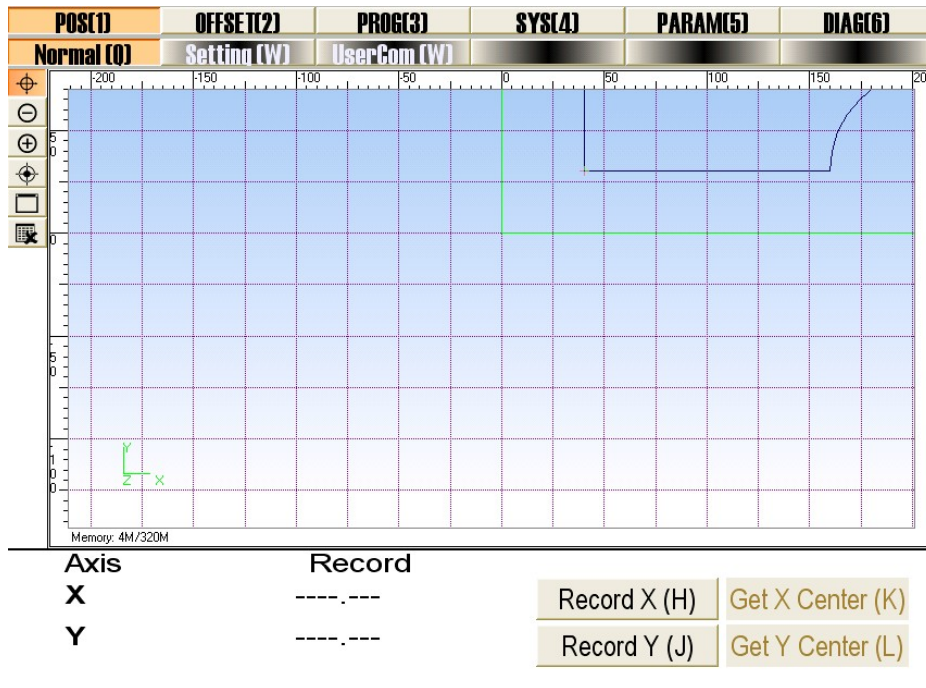
Fig. 6-4 normal function screen under manual mode

◆ [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 relatively getting X Center and getting Y center functions.

● **Setting Function Screen**

Setting function screen under manual mode is as shown in 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 presses the manual direction key, with range: 1 mm/min ~ Manual High Speed. Manual high speed refers to the moving speed when user presses the manual direction 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
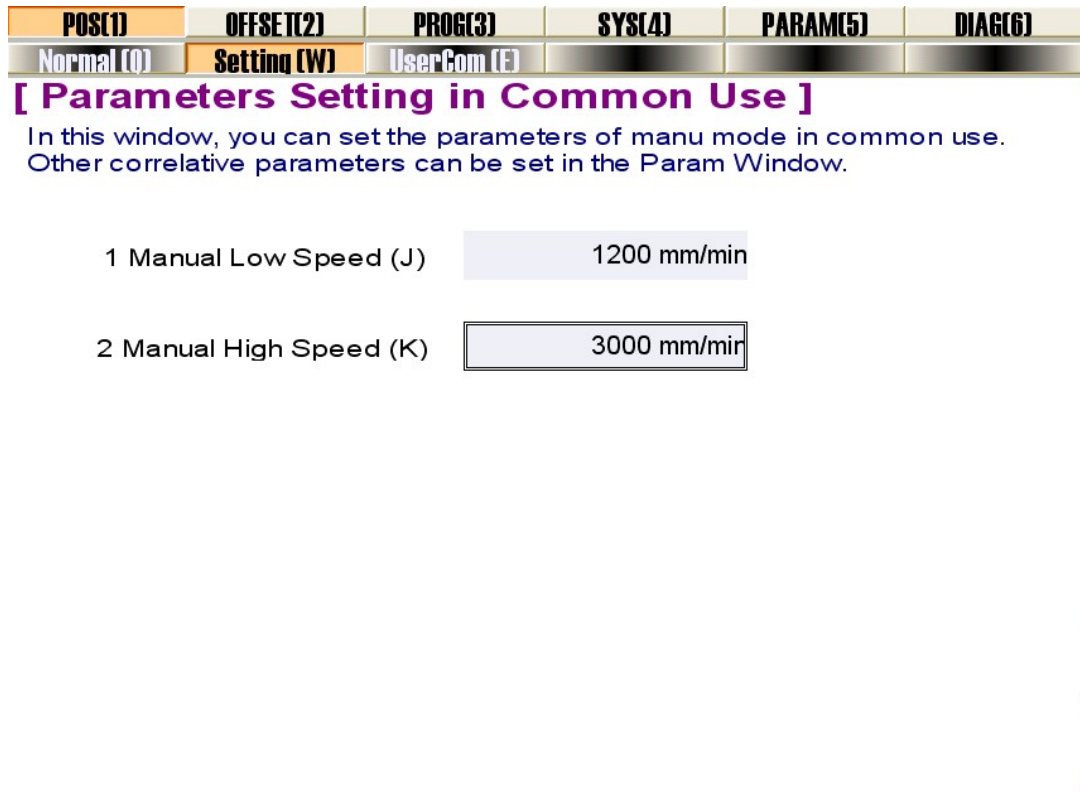
Fig. 6-5 Setting Function Screen of POS Function Section under Manual Mode

● **[Manual Date Input]**

　　Manual Date Input screen is as shown in Fig. 6-6. In this screen, user can input G instruction manually and execute the instruction. The screen supplies ten instruction input items, and every item supports inputting instruction many times. If no instruction is input into the item, the [Execute] button is disabled. Click [Edit] button, system will pop a dialog "Input Box" 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, Picture manipulation button column in the Normal function screen of POS function section is as shown in Fig. 6-7.

◆   [Clear] option

F1	F2	F3	F4	F5	F6	F7	F8
XClear	YClear	ZClear	XYClear				BackToO

Fig. 6-7 Manipulation Button Column of Setting Screen

The dialogue of system will pop up as following in Fig.6-8, when executing [clear] action:

NcEditor

? Are you sure to change the offset as zero by modify current workpiece coordinates (G54?)?

Yes   No

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 **BackToO** F8, system will be back to the setting workpiece origin.

There is only one [Back to 0] in the manipulation 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 and the machine actual coordinates. Therefore, during activation process, it is inevitable to back to the reference point. After starting the system each time, the 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:
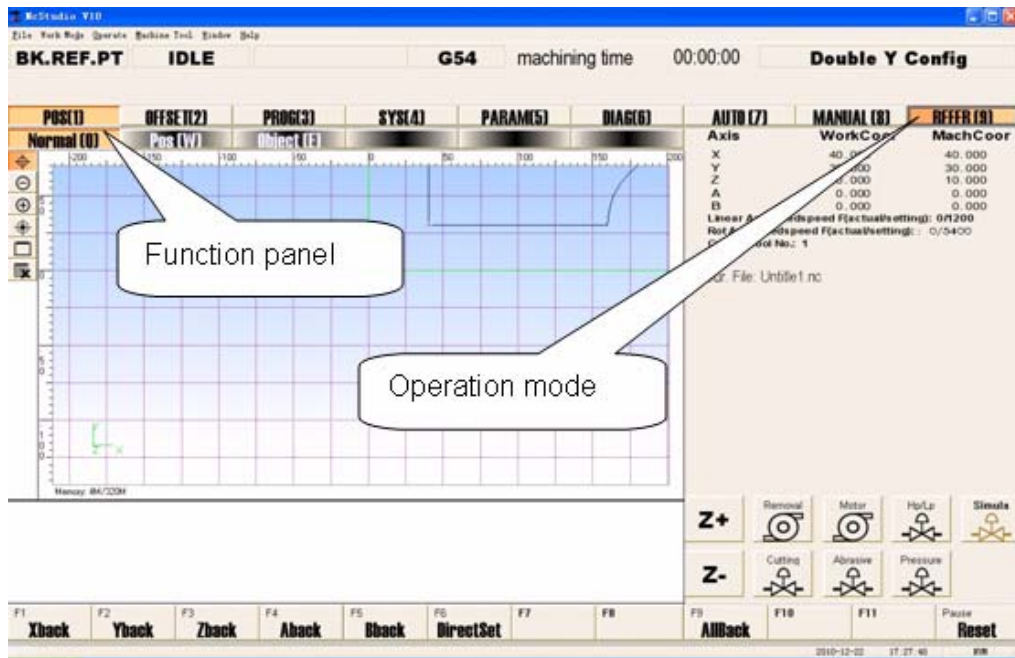
Fig. 7-1 Reference operation panel Dialog

The operation panel of reference point displays the operation button related with mechanical point and also supports 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 that in manual and auto operation panels.

7.1 Operation Panel Section

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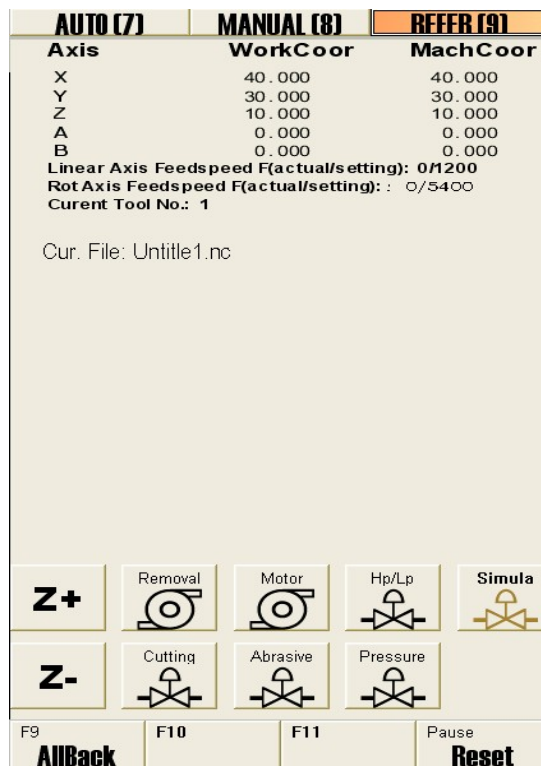
Fig. 7-2 Operation panel Dialog

The operation panel of reference point displays the operation button related with mechanical point and also supports an alternant interface for manipulating machine tool to the origin of mechanical coordinates system.

◆ [All Back]

Shown as:

F9
AllBack
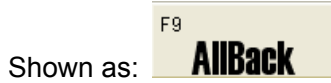
Machine origin (the reference point) is a fixed position on the machine, and is positioned by both machine switch and electrical system. With regard to the Machine Coordinate system, please refer to chapter 3.2. Operation of backing to the reference point requires the support of machine hardware, so 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 after E-stop or program start or program stop.

[AllBack] button is used to call X-axis, Y-axis, Z-axis, A-axis and B-axis to back to the reference point. V10 Ncstudio also supports single-axis back. For detailed information please refer to chapter 7.2.

◆ [Reset]

This function has already been introduced under auto mode, which won't be introduced any more.

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, 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 the same as that in auto and manual mode, only the lower parts of manipulation button bar are different. Therefore, we will no long introduction here. Please refer to the picture as following:
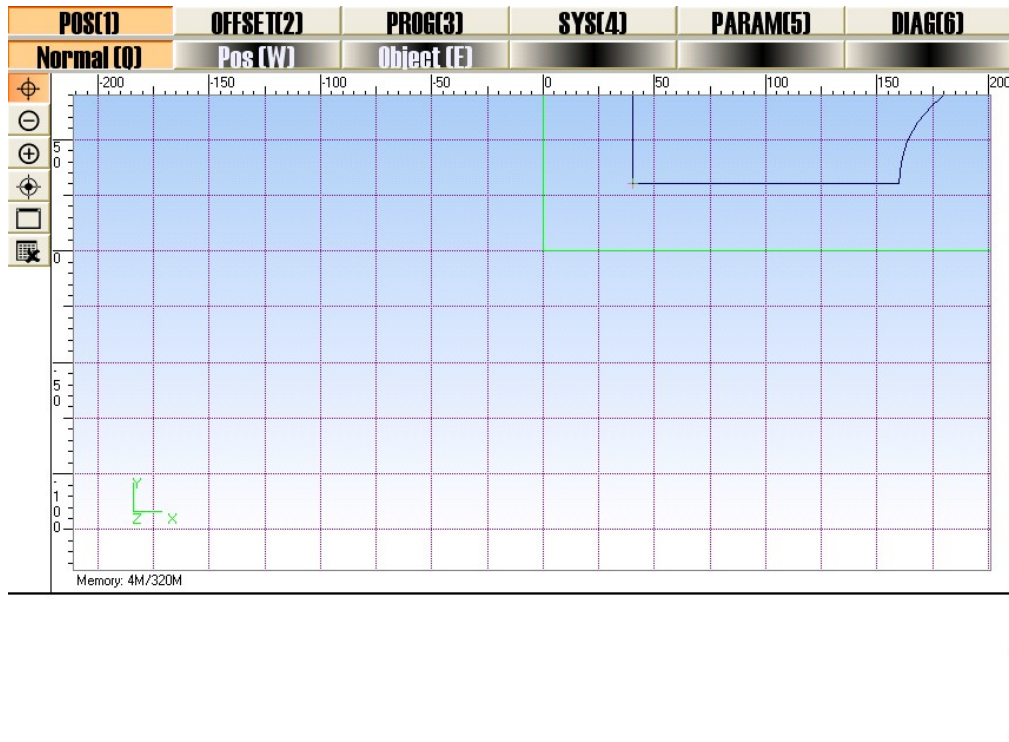
Fig. 7-3 Normal function screen under Reference point mode

● Position Function Section

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

● [Track Function] Screen

The upper part of track function screen is the same as what under auto mode, only the lower parts of manipulation buttons are different. The picture will be omitted.

◆ Manipulation Button Bar

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

Fig. 7-4 Manipulation Button Bar under REFER mode

◆ [X_Back], [Y_Back], [Z_Back], [A_Back], [B_Back]

Hotkey: F1, F2, F3, F4, F5 respectively, it is available only under REFER mode. Press the button, system will control the related single axis to go back to the machine origin. When the operation 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.

◆ [DirectSet / Direct Setting]

Hotkey: F6, it is available only 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 "Shut" and "E-stop", click the button to directly set the current machine coordinate as the accurate machine coordinate. When the operation 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 after X and Y axis.

8 Menu System of Ncstudio™

Menu operation, keyboard operation, and shortcut key operation constitute the entire operation framework for NcstudioTM. Menu operation is one of the most important manners to operation NcstudioTM and provides user with very affluent menu commands, which you will have a profound witness in the following contents.

8.1 File Menu

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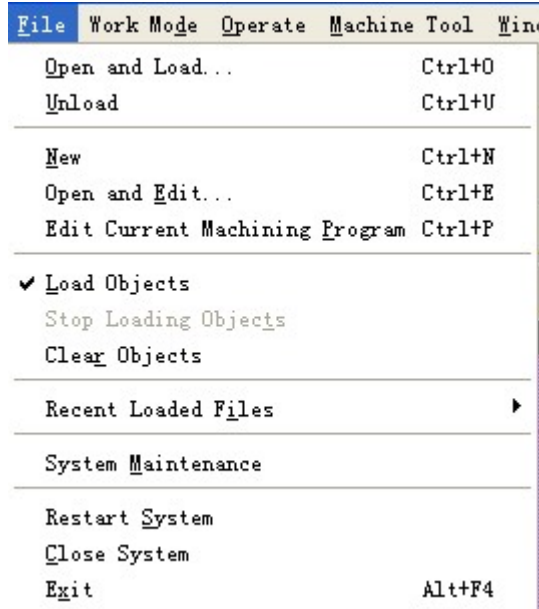
Fig. 8-1 File Menu

◆   [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 parse engine will parse the file into G instructions, and transfer these instructions to CNC card.

Choose the menu item; system will flip out the "Open and Load" dialog box. Show in Fig.8-2. After loading the processing file, the operation panel will display the name of processing file loaded.
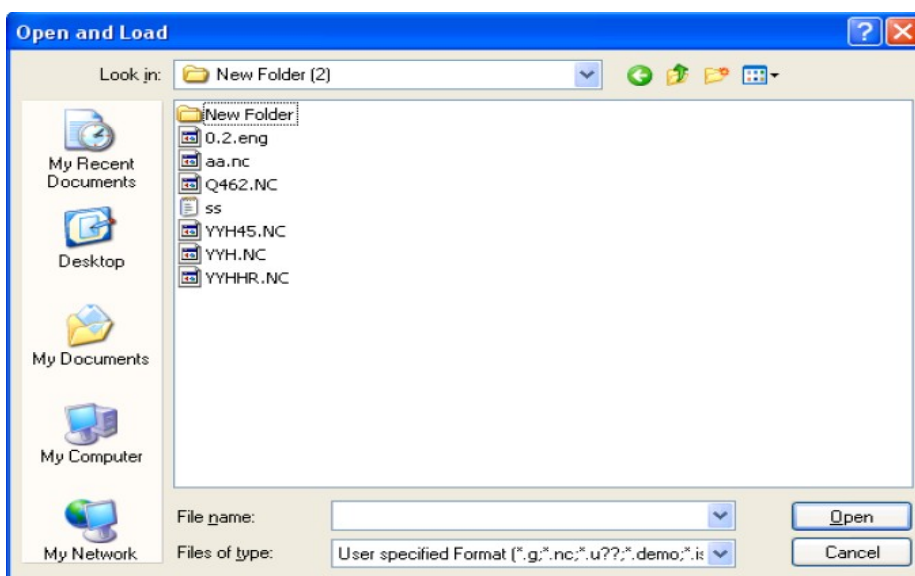
Fig. 8-2 Open and Load Dialog

Once a processing file is loaded into NcStudio system, it can not be modified. So if user want to modify the processing file loaded, they must unload the file first, then modified in NcEditor.

Note: When the menu item with symbol "…" behind is chosen, the system will pop up a dialog.

◆　[Unload]

This menu item has the same function as the 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.

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

◆　[New]

Choose this menu item; the system will create a new file 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 file saved in disk and loading it into NcEditor. User can edit the processing file in it. Clicking the menu option will pop up [open and edit] dialogue box. User can operate it according to the prompt of the dialogue box, please refer to the detail in NcEditor manual.

◆　[Edit Current Machining Program]

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

◆　[Load Objects]

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 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 the opposite option is [Clear Tracks].

◆　[Stop Loading Objects]

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

◆　[Clear Objects]

The menu item is used for clearing the loaded track and forbidding continuing loading track. After clicking this option, it 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 can display the recent loaded processing file. If users need to load these files, they can click the file name directly, it is more convenient for user to load the processing file.

◆　[System Maintenance]

The menu item is mainly used for making a backup, resuming, repackaging and etc. Selecting this menu item will close the NcStudio for a while. Shown in Fig.8-3 below, and then system will pop up an option selecting dialog box. The dialog includes: key data backup, key data resume, system installation,

and generate installation packet.
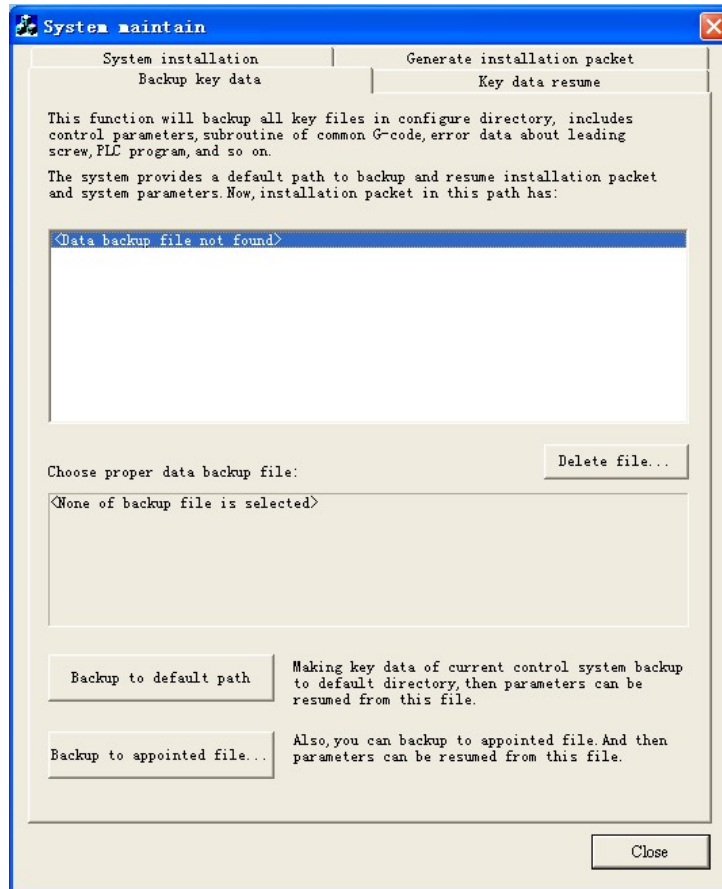
Fig. 8-3 System Maintenance Dialog

➢ Under the "Backup key data" option card, user can back up key files in default directory, including controlling parameters, subprogram of common-G code, error data of lead screw, PLC program, and so on. If there are backup files under the default directory, they will be displayed in the window below. User can delete or preserve them, and also can check the property of the backup by clicking the backup in the window below. User can select whether to back up the files under the default directory. If not, user can appoint another directory for files backup.

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

➢ "System installation" Option Card: User can install a NcStudio system once again. To be careful, 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 before installation. If there are already any installation packages under the default directory (It is the same as package generate directory), they will be displayed in the window. Click one package, its attribute will be displayed in the lower part of the window. User can delete or install it. Press direction-key to select other installation package, click button [Start Installation], 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 back up or save a stable system version. If there are already installation packages under the default directory (it is the same as system installation package directory), they will be displayed in the lower part of the window. Click one package, its attribute will be displayed in the lower part of the window. User can delete it or make it into a new installation package. Press the direction-key to select other installation package, and click the button [Generate to default path] or [Generate to appointed path], a "Generate install packet" dialog will pop up, then click button [Start Creating].

◆ [Restart System]

User can restart Ncstudio^{TM} CNC system by selecting this item, after clicking this item, a dialogue will pop up, user can choose "Yes" to restart the system.

◆ [Shut Down System]

User can close Ncstudio^{TM} CNC system by selecting this item, after clicking this item, a dialogue will pop up to prompt user whether to shut down system, user can choose "Yes" to close the system.

◆ [Exit]

The menu item is used to close NcStudio^{TM} system. If there is any file being edited, system will prompt user to save it. And user can follow the prompt to execute the related operation.

8.2 Mode Menu

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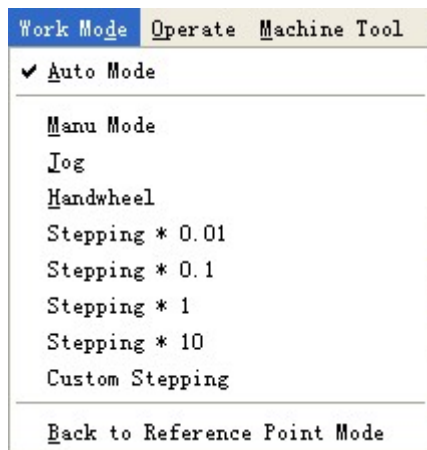
Fig. 8-4 Working Mode Menu

◆ [Auto Mode]

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

◆ [Manual Mode]

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

◆ [Series]

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

◆ [Handwheel]

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

◆ [Stepping * 0.01], [Stepping * 0.1], [Stepping * 1], [Stepping * 10]

It corresponds to step-size buttons on operation panel. Selecting one menu item will appoint stepping mode as current manual manner, and user can select the corresponding step-size.

◆ [Back to Reference Point Mode]

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

8.3 Operation Menu

[Operator] menu mainly includes the items about manipulation, as shown in Fig.8-5.
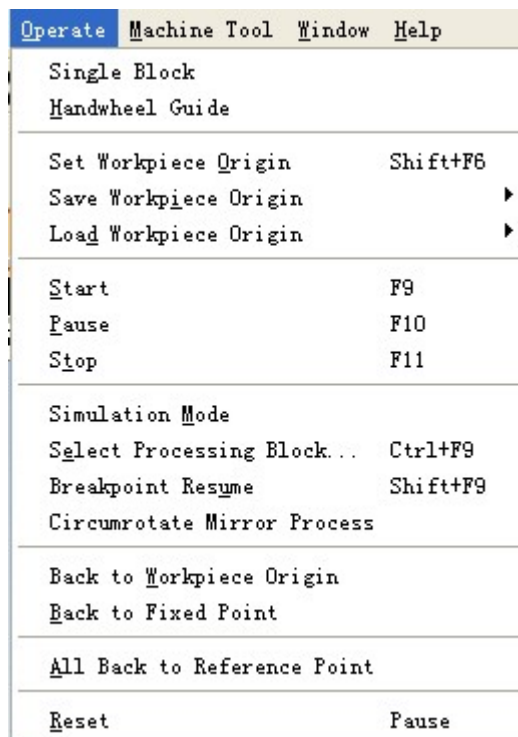
Fig. 8-5 Operation Menu

◆ [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 false diagnose and fault recovery action. 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 waterjet, install the handwheel first, and then select this menu item, system will change into handwheel mode.

◆ [Auto Back]

The menu item is a new function of V10 NcStudio and only available under AUTO mode. It is used for previous gradation during machining process. When user needs to return to a foregone position, select this menu item, 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] section.

◆ [Save Workpiece Origin]

This menu item is used to save current workpiece origin into system machining file. Regarding workpiece origins that are often used, user can save them for future use. There are totally 10 groups of workpiece origin to be saved. After saving a workpice origin for the first time, if you want to locate to this workpiece origin again, please save the workpiece origin.

◆ [Load Workpiece Origin]

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

◆ [Start], [Pause], [Stop]

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

◆ [Simulation Mode]

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

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 waterjet tool path with high speed in the processing trace window. By simulation, user can know about cutter's movement in advance to avoid machine tool damage (e.g. overtravel) which is caused by mistake in writing procedure, and get other additional information.

Once simulation starts, this menu item will change into "Stop and Exit Simulation", select this item again, system will exit simulation.

◆ [Select Processing Block]

The menu item corresponds to button [SelWork] of [POS] screen under [AUTO] mode. It is used for program block skip execution and only available under AUTO mode. Its shortcut key is Ctrl + F9. It is also applied to simulation mode. Choose the menu item, system will pop up a "Execute (Advance Options)" dialog, as shown in Fig. 8-6 below. Use can operate according to the prompt.

Fig. 8-6 Execute (Advanced Options) Dialogue

◆　[Breakpoint Resume]

The menu item corresponds to button [Continu] of [POS] function screen under [AUTO] mode, only available under AUTO mode. Its shortcut key is Shift + F9. Select this menu item, system will resume machining from the interrupting point of last time. It is also applied to simulation mode.

◆　[Rotary Mirror Machining]

This item will set workpiece origin as the center and begin rotary mirror machining, the dialogue will pop up as Fig.8-7 after selecting this item.
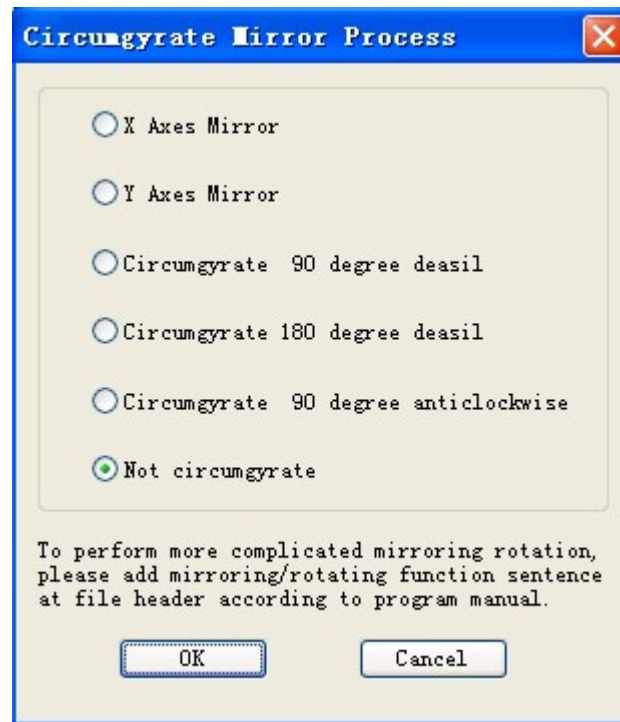
Fig. 8-7 Rotary Mirror Machining Dialogue

◆　[Back to Workpiece Origin]

Workpiece origin is the zero point of workpiece coordinate system; generally speaking, it is the 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].

Select the menu item, nozzle will return to the workpiece origin automatically from current position.

[Back to Workpiece Origin] have same function with button [Back to 0] of [POS] function screen under AUTO and MANUAL modes.

◆　[Back to Fixed Point]

User can set a fixed point in advance, under the situation that there is any damage to the nozzle or the nozzle alteration is needed, select this menu item, system will uplift the nozzle and back to the fixed point for you to change tool.

◆　[All Back to Reference Point]

This menu item is used to call all axes to go back to reference point. We suggest user to use this function before machining stars or when E-stop occurs.

◆　[Reset]

The menu item corresponds to button [Reset] on operation panel, with shortcut key F12. Select 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 will terminate the machining with E-stop, and the state of water valve and high pressure will not be changed, there is no deceleration and no cutter uplift.

8.4 Machine Tool Menu

[Machine Tool menu] mainly includes items about valve, pump and machine origin; it can realize the functions of machine origin checkout, start/ stop valve or pump, etc. See Fig.8-8.
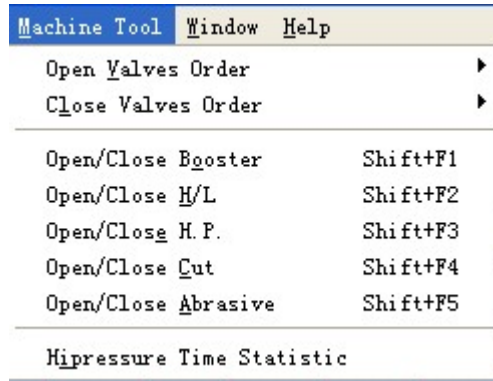
Fig. 8-8 Machine Tool Menu

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

The two menu items can be used to set the sequence to open/close valves, pumps. And their submenus supply all the combination sequences. Select one group of them, system will operate automatically according to the selected sequence.

◆ [Open/Close Removal], [Open/Close Hp /Lp], [Open/Close Cutting], [Open/Close Abrasive], [Open/Close Pressure]

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

◆ [High Pressure 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 broken easily. So machine manufacturer needs to maintain it regularly. Select this item, system will pop up a "High Pressure Time Statistic" dialog, as shown in Fig. 8-9. Button [Edit…] is used to clear the time. User needs passwords to enter it. Typical user can only have right to look over it.
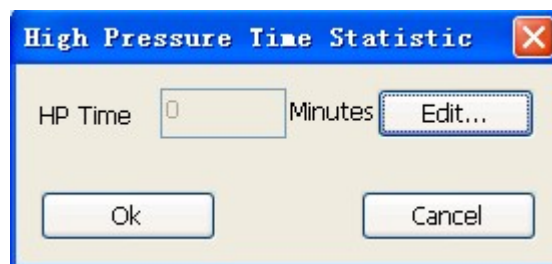
Fig. 8-9 High Pressure Time Statistic Dialog

8.5 Window Menu

[Window] menu is used for interface navigation. It marks all visible windows under current mode. Some screen may be different under different mode. See Fig.8-10.
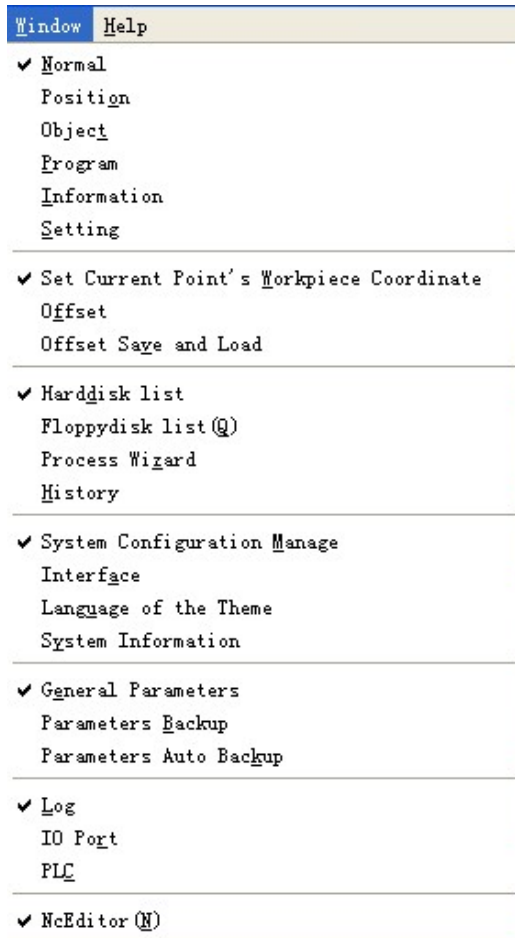
Fig. 8-10 Window Menu

8.6 Help Menu

The menu is used for displaying system and registration information. See Fig. 8-11.

Fig. 8-11 Help Menu

◆   [About NcStudio]

Select the menu item, system will pop up a dialog "About NcStudio". See Fig.8-12.
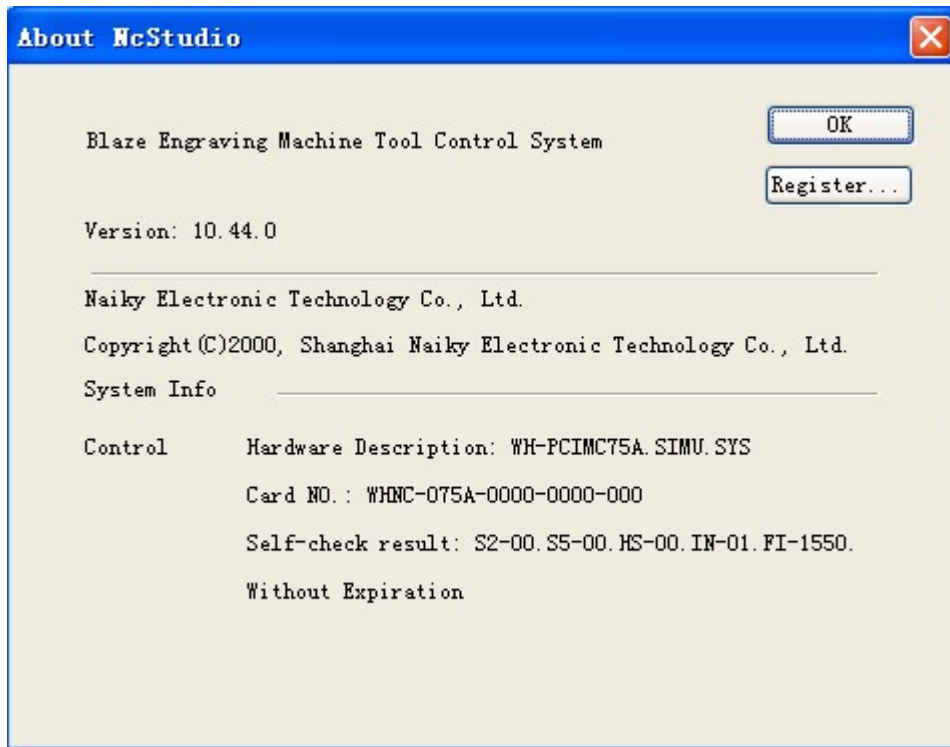
Fig. 8-12 "About Ncstudio" Dialogue

From above figure, user can look up the information about version, motion card, and manufacturer. Click button [Register…], system will pop up a "REGISTER" dialog, as shown in Fig. 8-13, user needs to input correct register code to finish registration.
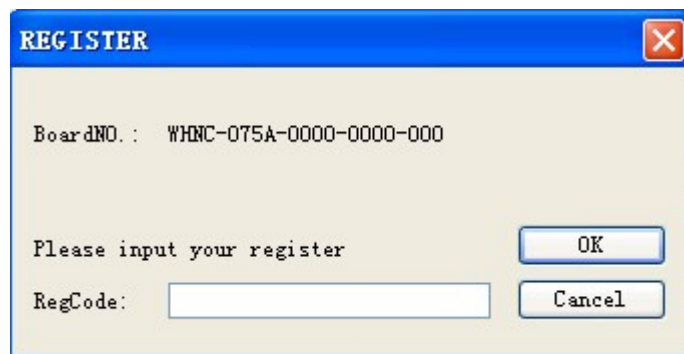
Fig. 8-13 REGISTER Dialogue

9 Parameters of Ncstudio™

NcStudio™ has abundant machining parameters. With the aid of these parameters, NcStudio™ is competent for various machining tasks. This chapter will specialize in introducing the parameters involved in NcStudio™ system. We divide the parameters into two parts: common parameters and waterjet parameters. Common parameters can be subdivided into three parts, operators' parameters, manufacturers' parameters and developers' parameters; Waterjet parameters are specially configured for different cutting machine tool. To look over or modify some types of parameters, user must have enough jurisdictions.

9.1 Methods for Parameter Modification

At first, 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 parameter that user want to modify. 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", user 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 user how to look over the complete content of the parameters: Look at the lower part of the parameter interface, this window will show user the complete information about the parameter selected; 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 operator's 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 manufacturers' parameters.

Fig. 9-2　The Manufacturers' Parameters

Parameters in V10 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). We offer the type name of the parameters according to the first and second digits.

In the following, we list all operators' parameters and entire manufacturers' parameters according to the coding number of the parameter. User can look up the parameters according to its coding number.

1.0. General Parameters of the Feed Spindle

◆ N10000~N10004　Spindle Direction（X\Y\Z\A\B）

Type:	INT
Unit:	None
Range:	1: Positive Direction -1: Negative Direction
Default:	X/Y/A:1 Z/B: -1
Valid time:	Be valid after restarting the system.
Instruction:	It specifies the motion direction of the specified axis: "1" indicates the positive increment; "-1" indicates the negative increment.

◆ N10010~N10014　Pulse equivalent（X\Y\Z\A\B）

Type:	Float
Unit:	X/Y/Z　Unit: mm/p（millimeter/pulse） Z/B: deg/p
Range:	0.0000009~999.0
Default:	0.01
Valid time:	Be valid after restarting the system.
Instruction:	It refers to the displacement or angle produced on the related 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～N10024　Worktable Stroke Lower Limit（X\Y\Z\A\B）

Type:	Float
Unit:	X/Y/Z: mm（millimeter） A/B: deg
Range:	-99999.0~99999.0
Default:	X/Y:0.0　　Z:-10000 A:0.0　　　B:-8
Valid time:	Have to restart the system after modification.
Instruction:	It refers to the lower limit of the workspace stroke on the direction of each axis under the situation that "worktable stroke detection" is valid

◆　　N10030～N10034 Worktable Stroke Upper Limit（machine coordinates）（X\Y\Z\A\B）

Type:	Float
Unit:	X/Y/Z: mm（millimeter） A/B: deg
Range:	Lower limit of the worktable travel range (Machine Coordinate)~99999.0
Default:	X/Y/Z: 10000 A: 10 B: 8
Valid time:	Have to restart the system after modification.
Instruction:	It refers to the upper limit of the travel range of the

	worktable on the direction of each axis under the situation that "worktable stroke detection" is valid

◆　N10040～N10044　Worktable Stroke Detection _Valid（X\Y\Z\A\B）

Type:	BOOL
Unit:	None
Range:	0 (false): Disable. 1(true): Enable.
Default:	1（true）
Valid time:	Be valid after restarting the system.
Instruction：	It sets whether to check the worktable stroke

1.2. Compensation

◆　N12000　Lead Screw Error Compensation Valid

Type:	BOOL
Unit:	None
Range:	0(false): Disable 1(true): Enable
Default:	1（true）
Valid time:	Have to restart the system after modification
Instruction:	The parameter specifies whether the lead screw error compensation is valid, including compensation of pitch error and backlash.

◆　N12001　Only Backlash Compensation Valid

Type:	BOOL
Unit:	None
Range:	0(false):Disable 1(true): Enable
Default:	1（true）
Valid time:	Have to restart the system after modification.
Instruction：	It specifies whether to execute error compensation according to backlash

◆　　N12010～N12014　Backlash (X/Y/Z/A/B)

Type:	Float
Unit:	X/Y/Z: mm（millimeter） A/B: deg
Range:	0.0~1.0
Default:	0.0
Valid time:	Need to restart the system after modification.
Instruction:	It specifies the backlash of each axis, which is valid when "only backlash compensation valid" is true

◆　　N12020　Quadrant_Errors_compensate_valid

Type:	BOOL
Unit:	None
Range:	0(false): Disable. 1(true): Enable.
Default:	0（false）
Valid time:	Have to restart the system after modification.
Instruction:	It specifies whether the circular across quadrant taper angle compensation is valid.

Across Quadrant Compensation Parameters are mainly dealing with the compensation parameters of the sharp-angle and circular across quadrant, which are 12 groups of parameters from group "0" to group "11", each of them includes four parameters, time, compensation value, delay and intensity. Take group "0" as an example, others can refer to group "0".

◆　　N12030　Time (Across Quadrant Compensation Parameter)

Type:	Float
Unit:	Sec（second）
Range:	0.0~0.1
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	The bigger the value, the longer the range influenced by the compensation. Generally, 0.02 second is recommended.

◆　　N12031　Compensation Quantity (Across Quadrant Compensation Parameter)

Type:	Float
Unit:	Mm（millimeter）
Range:	0.0~999.0
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	The larger the value is, the more obvious the compensation effected is. But be noted, If the value is too large, the curve would be projecting inward, and if too small, it would reduce the height of curve; 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 (compensation effect and compensation time are related to compensation intension).

◆　　N12032　Compensation Delay Time（Across Quadrant Compensation Parameter）

Type:	Float
Unit:	Sec（second）
Range:	0.0~0.1
Default:	0.0
Valid time:	Be valid after restarting the system.
Instruction:	Owing to the different mechanical properties of different machine, the position that the sharp-angle appears may not be at the quadrant conversion point, but some distance far away from the quadrant conversion point. Therefore, please estimate the time corresponding to this distance and set this distance as the value of the parameter.

◆　　N12033　Compensation Intensity（Across Quadrant Compensation Parameter）

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 has an influence on the compensation effect. The larger the value, the more obvious the compensation effect is.

1.3.Speed / Acceleration Limit

◆　N13000～N13004　Maximum Acceleration of Each Axis（X\Y\Z\A\B）

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.01~1000000.0
Default:	48000．0
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	It is used to specify the maximum acceleration which is available to each axis.

4.2.Cylinder

◆　N42000　Red light prompts after processing ends

Type：	BOOL
Unit：	none
Range：	0: false 1: true
Default：	0(false)
Valid time:	Be valid after restarting the system.
Instruction：	Whether to use red light for tips after processing ends until there is external input by mouse or keypad

5.2.Handwheel & Keypad

◆　N52001　StrictCount_of_HandwheelPulse

Type:	BOOL
Unit:	None

Range:	0(false): Disable. 1(true): Enable.
Default:	0（false）
Valid time:	Be valid after restarting the system.
Instruction:	If the parameter is set valid, the moving distance of lathe will be strictly count corresponding to 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 invalid, the system may have rapid respondence time to the handwheel rotation. But when the handwheel moves too fast, the moving distance of the machine tool may not consist with the rotation distance of the handwheel.

◆　N52002　Handwheel Direction

Type:	Int
Unit:	None
Range:	1, Maintain the original moving direction of the machine during the motion of handwheel. -1, Maintain the original moving direction of the machine during the reverse motion of handwheel.
Default:	-1
Valid time:	Be valid after restarting the system.
Instruction:	It sets the positive and negative moving direction of the lathe when the handwheel rotate. 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 counter-clockwise, the machine tool will move in the negative direction.

◆　N52003　Handwheel Pulse Ratio×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 every pulse produced by the handwheel as the actual one pulse that has been received.

◆　N52004　Handwheel Ratio×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 "1" pulse produced by the handwheel as the actual 10 pulses that have been received.

◆　N52005　Handwheel Ratio×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　HandWheel Guide Ratio (numerator / denominator)

Type:	Int
Unit:	None
Range:	1~1000
Default:	1
Valid time:	Be valid after restarting the system.

Introduction:	When using handwheel to control the machine tool, 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   Forbid reverse function in handwheel guide

Type：	Bool
Unit：	None
Range：	0(false): The reverse function isn't forbidden. 1(true): The reverse function is forbidden.
Default：	0 (False)
Valid Time:	Be valid after restarting the system.
Instruction：	It specifies whether to forbid reverse function in handwheel guidance.

◆ N52012   HandWheel Acceleration

Type:	Float
Unit:	mm/s2（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 use the handwheel.

◆ N52020   Extend keyboard style

Type：	Bool
Unit：	None
Range：	0~2
Default：	0

Valid Time:	Be valid after restarting the system.
Instruction：	It specifies the type of extend keyboard. "0": None; "1": Mini keyboard; "2": extend keyboard

◆   N52021   Install extend keyboard hook

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 indicates whether to install extend keyboard hook

6.2.G Code Options

◆   N62000   Deceleration Distance

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~999.0
Default:	5.0
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	To protect the cutter, the machine tool will decelerate when it is close to a target position on the process of positioning (at approaching speed). The parameter indicates the distance away from the target position where the machine begins to decelerate

◆   N62001   Approaching Speed

Type:	Float
Unit:	mm/min（millimeter /minute）
Range:	0.001~99999
Default:	300.0
Valid time:	Be valid immediately. Needn't to restart the system.

Instruction：	In the process of positioning, it indicates the feed speed when the waterjet approaches the workpiece. (Distance between the cutter and the workpiece is smaller than the distance of deceleration)

◆ N62020 Arc_IJK_IncrementalManner_Valid

Type:	BOOL
Unit:	None
Range:	0（False）Disable 1（True）Enable
Default:	1（true）
Valid time:	Be valid immediately. Needn't to restart 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 which corresponds to circle center, please refer to the program manual for detail.

◆ N62021 Arc Radius Tolerance

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~9999.0
Default:	1
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	In the IJK incremental manner of G02 and G03, it will count twice the radius. 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. Recommendation tolerance is 100 micron.

◆ N62090～N62094 G09 Tolerance（X\Y\Z\A\B）

Type:	Float
Unit:	Mm
Range:	0.0~99.0

Default:	0.001
Valid time	Be valid immediately.　Needn't to restart the system.
Instruction:	It is used together with the accurate stop command of G09 and with the parameter of pulse equivalent. Regarding the detailed instruction, please refer to the additional introduction of parameter.

N62090～N62094　G09 Tolerance（X\Y\Z\A\B）Additional Introduction of Parameter:

G09 Accurate Stop, not in simulation mode. The instruction is only effective with the specified procedure block. To execute this command, the waterjet will decelerate at the end of the procedure block, and implement the arriving check. 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 cutter will decelerate to stop at the end point of the procedure block. In the meantime, system will compare the cutter actual position which 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〔Maximum accurate time], N11320～N11324[Direction of the accurate stop encoder], N11330～N11334〔Pulse equivalent of the accurate stop encoder], N11340～N11344[Accurate stop encoder gain].

6.3. Track Control

◆　N63001　Turning Angle Tolerance.

Type:	Float
Unit:	mm（millimeter）
Range:	0.0~0.1
Default:	0.0
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	For the smoothness of the whole workpiece, at the connection point of two procedure blocks, the cutter may not move to the specified position accurately. When the difference between the actual cutter position and the specified position is equal to the value of this parameter, the system will regard the complement of this procedure block.

◆　N63006　Track Smooth Time

Type:	Float
Unit:	S（Second）
Range:	0~0.064
Default:	0.01
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction：	The bigger the value of the parameter is, the smoother the surface of the workpiece is. But if the value of the parameter is too large, it will influence the dimension of the workpiece.

6.4.Speed Control

◆ N64000 Start-up Speed

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~ the maximum speed
Default:	0.0
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	It is corresponding to the jump frequency of the servo motor, referring to the achieved motor speed starting from zero without acceleration, which is the minimum speed of the machine motion.

◆ N64020 Linear Axis Rapid Transverse Speed

Type：	Float
Unit：	mm/min（millimeter/minute）
Range：	0.0~10000.0
Default：	6000.0
Valid Time：	Be valid immediately. Needn't to restart the system.
Instruction：	It is the default running speed (not the processing speed) of the linear axis in positioning (under G00 command)

◆ N64030 Rotary Axis Rapid Transverse Speed

Type：	Float
Unit：	rpm
Range：	0.0~10000.0
Default：	26
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the default running speed (not the processing speed) of the rotary axis in positioning (under G00 command)

◆ N64031 Rotary Axis Machining Speed

Type：	Float
Unit：	rpm
Range：	0.0~10000.0
Default：	15
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the default machining speed (not the positioning speed) of the rotary axis

◆ N64080 Rotary Axis Angular Acceleration

Type：	Float
Unit：	deg/ s^2
Range：	0.0~10000.0
Default：	300.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the default agular acceleration of rotary axis in machining

◆ N64101 Linear Axis Rapid Transverse Acceleration

Type：	Float
Unit：	mm/s^2（millimeter/ second^2）
Range：	0.001~100000.0

Default：	400.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the maximum accelerateon (not the processing speed) of each feed axis in positioning

◆　N64102　Z-axis Acceleration

Type：	Float
Unit：	mm/s^2（millimeter/ second^2）
Range：	0.001~100000.0
Default：	400.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the maximum accelerateon (not the processing speed) of Z-axis in machining

◆　N64120　Linear Axis Turning Acceleration

Type：	Float
Unit：	Mm/s^2（millimeter/second^2）
Range：	0.001~100000.0
Default：	3000.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the max acceleration of the feed motion on the adjacent axes. We recommend to set the value as 1~2 times of the acceleration of the single axis.

◆　N64150　Linear Axis Plus The acceleration

Type：	Float
Unit：	Mm/s^3（millimeter/second^3）
Range：	0.001~100000.0
Default：	5000.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It describes the growth rate of the acceleration. (acceleration's acceleration)

◆ N64203 Interpolation algorithm selection

Type：	INT
Unit：	None
Range：	0: Trapezoidal Algorithm 1: S Type Algorithm
Default：	1
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	Please choose the suitable interpolation algorithm to reduce the error and debug before selecting

◆ N64210 Stable Processing_ Valid

Type：	Bool
Unit：	None
Range：	0：invalid 1：valid
Default：	0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It specifies whether the system will take stable processing. When this parameter is set as valid, the machine will be more stable during processing, but the processing speed will be slowed down.

◆ N64211 Current efficiency strategy group

Type：	INT
Unit：	None
Range：	0-4
Default：	0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is used to specify the efficiency strategy group.

◆ N64212、N64217、N64222、N64227、N64232 Turning acceleration regulatory factor

Type：	Float
Unit：	None

Range：	
Default：	Group 0: 1 Group 1: 1.5 Group 2: 2 Group 3: 2.5 Group 4: 5
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It specifies the Turning acceleration regulatory factor

◆ N64213、N64218、N64223、N64228、N64233　single axis processing acceleration regulatory factor

Type：	Float
Unit：	None
Range：	
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. Needn't to restart the system.
Instruction：	This parameter is used to specify single axis processing acceleration regulatory factor

◆ N64214、N64219、N64224、N64229、N64234　Arc speed limitation regulatory factor

Type：	Float
Unit：	None
Range：	
Default：	Group 0: 1 Group 1: 1.2 Group 2: 1.6 Group 3: 2 Group 4: 999
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It specifies the arc speed limitation regulatory factor

◆ N64240　smooth time unit factor

Type：	Float
Unit：	None
Range：	0.01-10
Default：	1
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It specifies the ratio of unit processing time to control periodic time, when the speed is smooth.

6.5.File Translation

PLT File Translation Parameter

Following are the needed parameters in the PLT format file machining

◆ N65000   Cutter Lifting Height (PLT File Translation Parameter)

Type：	Float
Unit：	Mm（millimeter）
Range：	0.0~99999.0
Default：	1
Valid Time:	Be valid after loading processing procedure again.
Instruction：	This parameter specifies the cutter lifting height of dry run.

◆ N65002   Space_ between_ Cutters _in_ PLT Area（PLT File Translation Parameter）

Type：	Float
Unit：	Mm（millimeter）
Range：	0.0~99999.0
Default：	0.025
Valid Time:	Be valid after loading processing procedure again.
Instruction：	The space between cutters needs to be confirmed by the diameters of cutters. To fulfill the processing, parts of the adjacent cutters must be overlapped. The overlapping path is the space between cutters in PLT area.

◆ N65003　Two Dimension Processing Depth（PLT File Translation Parameter）

Type：	FLOAT
Unit：	Mm（millimeter）
Range：	-99999.0~0.0
Default：	-1
Valid Time:	Be valid after loading processing procedure again.
Instruction：	It is used to specify the processing depth of 2D file.

DXF File Translation Parameter

DXF file translation parameters are used for the translation of DXF file, which include the following parameters: cutter lifting height of dry run movement, the processing depth of 2D file, processing mount of each time, taking the starting point as the zero point, the shape separate processing valid, bottom processing valid, and so on.

◆ N65100　Cutter Lifting Height（DXF File Translation Parameter）

Type：	FLOAT
Unit：	Mm（millimeter）
Range：	0.0～99999.0
Default：	1
Valid Time:	Be valid after loading processing procedure again.
Instruction：	It specifies the cutter lifting height of dry running

◆ N65102　Processing Amount Of Each Time（DXF File Translation Parameter）

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 cutting depth of 2D file processing.

◆ N65103　Taking_ Starting Point_ as_ Zero Point（DXF File Translation Parameter）

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 Bottom processing valid (DXF file translation parameter)

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 only enables threshold manipulation on the surface of workpiece (three-dimensional cutting)

◆ N65106 Forced identification DXF file as metric size

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 refers to whether to force identification DXF file as metric size

ENG File Translation

Following are the needed parameters in the ENG format file machining

◆ N65200 Cutter Lifting Height（ENG File Translation Parameter）

Type：	FLOAT

Unit：	Mm
Range：	0.0～99999.0
Default：	1
Valid Time:	Be valid after loading processing procedure again.
Instruction：	It specifies the cutter lifting height of dry running.

◆ N65203　Use_ ENG_ File_ to_ Select_ Tool _for _Processing（ENG File Translation Parameter）

Type：	Bool
Unit：	None
Range：	0（False）：not in use 1（True）：under use
Default：	0（false）
Valid Time:	Be valid after loading processing procedure again.
Instruction：	When using tool selection function, once you open a ENG file, a dialogue will pop up and require the user to select a tool to do processing (The tool used in Eng file is not the default tool in system settings)

7.1.Manual

◆ N71000　Manual mode low speed

Type:	Float
Unit:	mm/min（millimeter/minute）
Range:	0.0~ Manual High Speed Value
Default:	3000.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction:	It specifies the max default system moving speed in pressing the direction keys under manual mode

◆ N71001　Manual mode high speed

Type：	Float
Unit：	mm/min（millimeter/minute）

Range：	0.001~100000.0
Default：	5000
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	This parameter specifies the default running speed of system when user presses down both direction keys and acceleration key under manual mode.

◆ N71002 Manual mode low speed of Rotary axis

Type:	Float
Unit:	rpm
Range:	0.0~ manual high speed value of rotary axis
Default:	15
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction:	It specifies the default speed of rotary axis when users press direction key under manual mode

◆ N71003 Manual mode high speed of Rotary axis

Type：	Float
Unit：	rpm
Range：	0.0~100000.0
Default：	26
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It specifies the default speed of rotary axis when user presses down both the direction keys and the acceleration key under manual mode.

◆ N71020 Prohibit _Moving_ in_ Limit Direction_ when _Limit _Effects

Type：	Bool
Unit：	None
Range：	0（False）：not Prohibit 1（True）：Prohibit
Default：	1（true）

Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It prohibits the spindle to move towards limit direction when the limit effects. And machine will be damaged if machine moves towards limit direction during limit.

7.2.Auto

◆ N72001 Select feed speed style

Type：	INT
Unit：	None
Range：	0: select file speed 1: select default speed 2: specify speed in proportion
Default：	0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It sets the selection of style of the feed speed

◆ N72006 Cutter lifting height

Type：	Float
Unit：	mm（millimeter）
Range：	0.0~99999.0
Default：	5
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It sets the cutter lifting height of dry running

◆ N72010～N72014 Check _the_ Workpiece Coordinates_ Valid（X\Y\Z\A\B）

Type：	Bool
Unit：	None
Range：	0（False）：invalid 1（True）：Valid
Default：	0（False）
Valid Time:	Be valid immediately. Needn't to restart the system.

Instruction：	It specifies whether to use the workpiece coordinates range check. The range is constituted by the upper limit and the lower limit of the workpiece coordinates.

◆ N72020～N72024 Lower Limit _of_ the_ Workpiece Coordinates（X\Y\Z\A\B）

Type：	FLOAT
Unit：	mm（millimeter）
Range：	-99999.0~99999.0
Default：	-10000.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is used to specify the lower limit of the workpiece coordinate when workpiece coordinate check is valid

◆ N72030～N72034 Upper Limit_ of_ the_ Workpiece Coordinates（X\Y\Z\A\B）

Type：	FLOAT
Unit：	mm（millimeter）
Range：	workbench stroke lower limit (Machine Coordinate) ~99999.0
Default：	10000.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is used to specify the upper limit of the workpiece coordinate when workpiece coordinate check is valid

7.3.Pause

◆ N73000 The_ Speed _for _Lowering _of _ Roll When Continues

Type：	FLOAT
Unit：	mm/min（millimeter/minute）
Range：	0.001~99999.0
Default：	10000.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It sets the speed of Z axis in processing after pause when lowering at cutting point before pause

◆ N73001 The_ speed_ of_ Cutter Lifting_ when_ pausing

Type：	FLOAT
Unit：	Mm/min（millimeter/minute）
Range：	0.001~99999.0
Default：	600
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It sets the Lifting speed of Cutter when pausing

◆ N73002 Positioning Mode of Z-axis When Pausing

Type：	INT
Unit：	None
Range：	0: specified cutter lifting amount 1: specified workpiece coordinate
Default：	0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It sets lifting mode of Z-axis in pausing, "0": to the specified cutter lifting amount; "1": to the specified workpiece coordinate

◆ N73003 The_ position_ of_ Z-axis when pausing

Type：	FLOAT
Unit：	Mm（millimeter）
Range：	0-9999
Default：	10
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	Be valid when setting the parameter N73002 to "0". It appoints the workpiece coordinate of Z-axis position when pausing.

◆ N73004 The specified cutter lifting distance of the Z-axis when pausing

Type：	FLOAT
Unit：	Mm（millimeter）
Range：	0-9999

Default：	10
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	Be valid when setting the parameter N73002 to "1". It appoints the cutter lifting height of Z-axis referring to the position before pausing.

7.4.Reference Point

◆ N74000   ESTOP_ Cancel_ Back To Reference Point Mode

Type：	Bool
Unit：	None
Range：	0（False）：Not cancel 1（True）：cancel
Default：	1（true）
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	If this parameter is set to be "True" to cancel the reference point, once there is ESTOP, the current symbol reference point will be canceled.

◆ N74001   Back_ to_ Mechanical Origin _before_ Processing

Type：	Bool
Unit：	None
Range：	0(False): Needn't to back to the reference point 1(True): Need to back to the reference point
Default：	0 (False)
Valid Time:	Be valid immediately. Needn't to restart 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, we need to select a reference point, which is the mechanical origin. During the processing, if an ESTOP occurs or power off, an error of displacement may arise when to restart the machine. With the function of backing to the mechanical origin, the system can retrieve the workpiece coordinates system and can ensure the accuracy to continue the processing.

◆ N74010～N74014 Position_ of_ the_ Reference Point（X\Y\Z\A\B）

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. Needn't to restart the system.
Instruction：	The mechanical coordinates of the mechanical origin position are fixed in accordance with the setting of [worktable stroke lower limit] (Machine Coordinate) and [worktable stroke upper limit] (Machine Coordinate)

◆ N74020～N74024 Rough_ Positioning _Phase Direction（X\Y\Z\A\B）

Type：	INT
Unit：	None
Range：	1：Positive direction -1：Negative direction
Default：	X/Y/A/B : -1 Z: 1
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It can be used for the rough positioning motion direction during backing to the reference point.

◆ N74030～N74034 Rough_ Positioning_ Stage Speed（X\Y\Z\A\B）

Type：	Float
Unit：	mm/min（millimeter/minute）
Range：	0.001~10000.0
Default：	X: 1800.0 Y: 1800.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It can be used for the rough positioning of stage feed speed during backing to the reference point.

◆ N74050～N74054 Exact _Positioning_ Stage Direction（X\Y\Z\A\B）

Type：	INT
Unit：	None
Range：	1：positive direction -1：negative direction
Default：	X/Y/A/B: 1 Z: -1
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It can be used for stage direction of the exact positioning during backing to the reference point.

◆ N74060～N74064 Exact_ Positioning_ Stage Speed（X\Y\Z\A\B）

Type：	Float
Unit：	X/Y/Z: mm/min (millimeter/ min) A/B: rpm
Range：	0.001~10000.0
Default：	60.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It can be used for feed speed of the exact positioning stage during backing to the reference point.

◆ N74080～N74084 Trace Back_ Distance（X\Y\Z\A\B）

Type：	Float
Unit：	mm（millimeter）
Range：	-1000.0~1000.0
Default：	X\Y: 2.0 Z: -2.0 A: 10 B: 8
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It refers to the additive moving distance after the complement of the exact stage positioning. Its value can be positive or negative. If the value is "0", it needn't to 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.

◆ N74090 Times _of_ Exact Positioning (Back_ to_ Mechanical Origin)

Type：	Float
Unit：	None
Range：	None
Default：	1
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It can be used for counting the times of exact positioning during backing to the reference point.

7.9.Other Operations

◆ N79000 Z-axis Down Feedrate Speed Selection

Type：	INT
Unit：	None
Range：	0: not available 1: only the Z-axis downwards feeding is valid 2: available when the z-component downwards feeding is included
Default：	0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	0: not available 1: only the Z-axis downwards feeding is valid 2: available when the z-component downwards feeding is included

◆ N79001 The Down Feedrate Speed _of _Z-axis

Type：	INT
Unit：	Mm/min (millimeter/minute)
Range：	0~99999.0
Default：	0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It specifies the down feedrate speed of Z-axis which is available when parameter N79000 is set "1" or "2"

◆ N79003 Safety Height

Type：	INT
Unit：	Mm
Range：	0~999
Default：	10.0
Valid Time:	Be valid immediately. Needn't to restart 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.

◆ N79100 Back to fixed point after finishing machining_ valid

Type：	Bool
Unit：	None
Range：	0(False): Invalid 1(True): Valid
Default：	1 (True)
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	To improve efficiency, user may need the machine tool move to one fixed position after the end of processing. For example, raise Z-axis to one fixed position after the complement of the workpiece processing, which facilitates the user in replacing the workpiece. If you want the machine to automatically back to one fixed point after processing, please choose this option and set the fixed point.

◆ N79110~ N79114 Mechanical Coordinates_ of_ the _Fixed Point（X\Y\Z\A\B）

Type：	Float
Unit：	X/Y/Z：mm A/B：deg
Range：	-99999.0~99999.0
Default：	X/Y/Z：0 A/B：0
Valid Time:	Be valid immediately. Needn't to restart the system.

Instruction：	This parameter is only valid on condition that the [AutoBack_to_the_FixedPoint_Valid_after_Processing] is valid. It indicates the machine coordinates of the fixed position.

◆ N79200 Displaying the limitation of simulation

Type：	Bool
Unit：	None
Range：	0(false):not clew 1(true):clew
Default：	1(true)
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is used to specify whether to clew in the situation of out of mechanical stroke in simulation mode

8.1.Position Interface

◆ N81000 Auto Track Loading

Type：	BOOL
Unit：	None
Range：	0（False）：invalid 1（True）：valid
Default：	1（true）
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	After file loading, this parameter sets whether to analyze the processing track automatically in order to facilitate users to load the track in advance.

◆ N81001 Auto Track Loading File Limit

Type：	INT
Unit：	KB
Range：	0~100000
Default：	1000
Valid Time:	Be valid immediately. Needn't to restart the system.

Instruction：	It is available when "auto loading track" is valid. If the processing file size is smaller than the value of this parameter, the system will automatically analyze the track of processing file, while if the processing file size is larger than the value of this parameter (The processing file size is too large), the system won't analyze the track of processing file.

8.3. Door Open Function Parameter

◆ N83009   Work space of Y-axis of machine tool

Type：	INT
Unit：	None
Range：	1：positive direction 2：negative direction
Default：	1
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction:	It sets the Work space of Y-axis of machine tool. "1"：positive direction; "2"：negative direction

9.3 List of Waterjet Parameters

In【WaterJet】Parameter screen, there are parameters which is 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' parameter and the manufacturers' parameter.

See the Fig.9-3, as following.
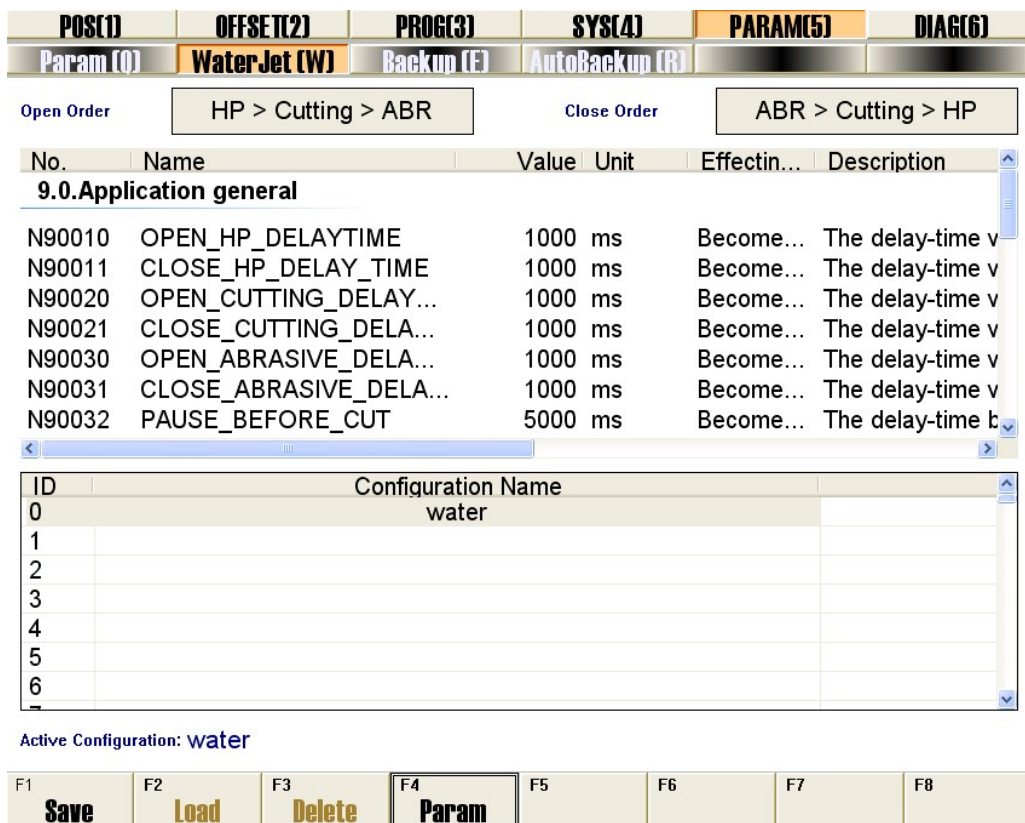
Fig. 9-3 the dialogue of water jet parameters

9.0.Concrete General Application

◆ N90010 TimeDelay_in_Opening_HighPressureValve

Type:	INT
Unit:	Ms（millisecond）
Range:	0 ~ 600000
Default:	1000
Valid time	Be valid immediately. Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time of opening the high pressure valve.

◆ N90011 TimeDelay_in_Closing_HighPressureValve

Type:	INT
Unit:	ms（millimeter）
Range:	0 ~ 600000
Default:	1000

Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time of closing the high pressure valve.

◆　N90020　TimeDelay_in_Opening_WaterValve

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	1000
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time of opening the water valve.

◆　N90021　TimeDelay_in_Closing_WaterValve

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	1000
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time of closing the water valve.

◆　N90030　TimeDelay_in_Opening_Abrasive Valve

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	1000
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time of opening the sand valve.

◆　N90031　TimeDelay_in_Closing_Abrasive Valve

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	1000
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time of closing the sand valve.

◆ N90032　TimeDelay_Before_Cutting

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	5000
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time before cutting

◆ N90034　TimeDelay_After_Cutting

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	0
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time after finishing cutting

◆ N90039　TimeDelay_in_Opening_High Pressure in advance

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

Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	This parameter can be used to specify the delay time in opening high pressure in advance

◆　N90044　TimeDelay_in_Automatically Closing_High Pressure under manual mode

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	0
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	It specifies the delay time of closing high pressure before closing water valve under manual mode

◆　N90045　Timedelay in closing high pressure & abrasive valve in machine stopping

Type:	INT
Unit:	ms（millisecond）
Range:	0 ~ 600000
Default:	0
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	It specifies the delay time of closing water valve after shutting down high pressure valve in machine stop

9.1.Operation Parameters

Following are related parameters of tool generation, and the tool path file should be saved again after modification.

◆　N91000　Tilt Angle of Cutting Plane

Type:	Float
Unit:	deg
Range:	0 ~ 0
Default:	2
Valid time:	Be valid immediately.　Needn't to restart the system.

Instruction:	It specifies the included angle degree between XY plane and cutting plane, please refer to Chapter 9.4.1 for detail

◆ N91002　Inclined Direction of Cutting Plane

Type:	Bool
Unit:	None
Range:	0 ~ 1
Default:	1
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	It specifies the inclined direction of cutting plane, "0": from the direction of machining, right slope of cutting plane, "1": left slope of cutting plane, from the direction of machining. please refer to Chapter 9.4.1 for detail

◆ N91003　Cutting Type of Planar Joint Angle

Type:	INT
Unit:	None
Range:	0 ~ 2
Default:	0
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	It specifies the cutting type of planar joint angle. "0": taper angle; "1": circular cutting; "2": mixed cutting

◆ N91005　Ellipse Fitting Segment Number

Type:	INT
Unit:	None
Range:	0 ~ 256
Default:	256
Valid time:	Be valid immediately.　Needn't to restart the system.
Instruction:	It specifies the ellipse segment number when linear fitting type is adopted in ellipse interpolation

◆ N91006 Circular Fitting Accuracy

Type:	Float
Unit:	None
Range:	0.01-0.001
Default:	0.001
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	When the linear fitting type is adopted in circular interpolation, this parameter specifies the length of each interpolation segment. The recommended range is 0.01 ~ 0.001, when low accuracy is needed, the value can be set as 0.1 ~0.01

◆ N91021 Workpiece Thickness

Type:	Float
Unit:	mm
Range:	0 ~ worktable stroke range of Z-axis
Default:	10
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	It specifies the thickness of workpiece in machining, the default value is 10mm, which is recommended to keep unchanged. Workpiece thickness refers to distance between the bottom and the surface of workpiece; For those shapes that taper angle cutting is unavailable, software will adopt automatically the mixed cutting type and the adjusting range can be fixed according to workpiece thickness.

◆ N91023 Fitting Accuracy Tolerance

Type:	Float
Unit:	None
Range:	
Default:	0.1
Valid time:	Be valid immediately. Needn't to restart the system.

Instruction:	It specifies the interpolation accuracy tolerance in adopting linear fitting type of interpolation

9.2.

◆ N92000 Machining Acceleration of Single Axis

Type:	Float
Unit:	mm/s^2 （millimeter/second^2）
Range:	0.001~100000.0
Default:	400
Valid time:	Be valid immediately. Needn't to restart the system.
Instruction:	It is used to specify the maximum acceleration which is available to each axis in machining

◆ N92001 Maximum Speed_ of_ the_ Reference Circle

Type：	Float
Unit：	Mm/min （millimeter/minute）
Range：	0.001~100000.0
Default：	1800.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It refers to maximum allowed speed corresponding to the 10 mm diameter of circle

◆ N92002 Minimum Speed of Circular Motion

Type：	Float
Unit：	Mm/min （millimeter/minute）
Range：	0.001~100000.0
Default：	180.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It refers to the minimum speed of machine tool in circular motion

◆ N92003 Linear Axis Machining Speed

Type：	Float
Unit：	mm/min（millimeter/minute）
Range：	0.001~100000.0
Default：	1200
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	It is the default machining speed (not the positioning speed) of the machine tool

◆ N92004 Accurate Stop Time

Type：	Float
Unit：	s（second）
Range：	0.001~100000.0
Default：	0.0
Valid Time:	Be valid immediately. Needn't to restart the system.
Instruction：	During the processing, because of the different inertia of each axis, the servo system may output lag phenomenon. This parameter is used to overcome the lag phenomenon produced by the servo system.

9.4 Parameters Specification and Operating Process of Five-axis Tool Path Generation

9.4.1 The Required Parameters Specification of Five-axis Tool Path Generation

1.   **Tilt Angle of cutting plane:**

The tilt angle refers to the included angle between cutting plane and Z axis, and the range is the minimum absolute value from 0 to A/B axis stroke, for instance, if the stroke range of A-axis is -8° ~ 9°, and the stroke range of B-axis is -10° ~11°, then the range of tilt angle is 0~8.

2.   **Inclined Direction of Cutting Plane:**

Left cut & Right cut:

Left cut refers to the left inclination of figure from the perspective of machining

Right cut refers to the right inclination of figure from the perspective of machining

3.   **Confirmation method of the tilt angle of cutting plane and the inclined direction**

The loaded file only has two dimension information, so if the bevel effect is required, there are two directions of bevel, which can be defined as the right inclination and left inclination (left cut and right cut). The distinguished method is as below: from the perspective of machining direction, if the bevel is towards the left hand, it is regarded as left cut; if towards right, right cut.

As the straight line AB below, the direction is as the black arrow goes. If the left cut is selected, the processed plane is "ABEF", that is to say, the line "EF" is on the left side of the line "AB"; if right cut is chosen, the processed plane is "ABCD", namely, the line "DC" is on the right side of the line "AB". Tilt angle means, as the figure shows, the included angle between plane ABCD and axis AM (AM is paralleled with Z axis), namely, the included angel ∠DAM.
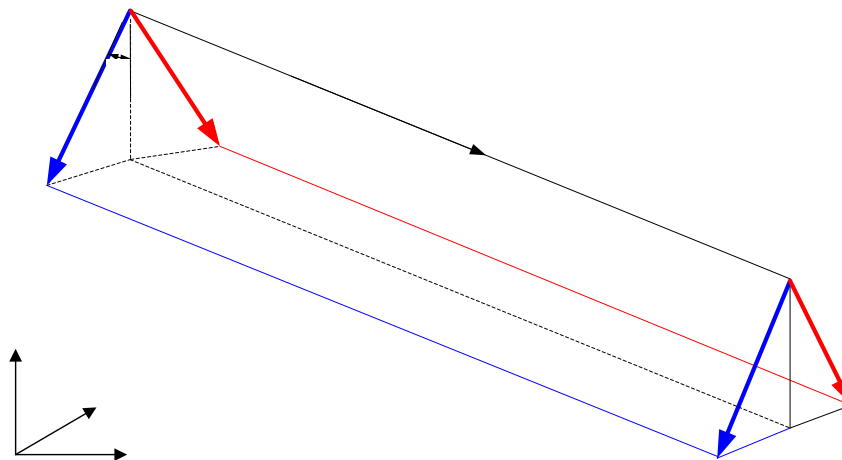
Fig. 9-4 Cutting Plane Inclined Direction Confirmation Sketch Map

9.4.2 Operating Process of Five-axis Tool Path Generation

1. Set five axes machining parameters

    ● As the previous introduced setting method of bevel inclined direction, set the parameters of inclined direction (Para ->Waterjet Para ->Waterjet Inclined Direction (left or right)), "Tilt Angle of Cutting Plane", "Circular Fitting Accuracy", "Workpiece Thickness" and "Fitting Accuracy Tolerance".

    ● Set other related machining parameters, such as dry running speed of machining, time delay parameters of high pressure, water valve, abrasive valve, and other related parameters of machining file

2. Load other machining file generated by CAD/CAM by NcEditor, take loading DXF file as an example:

    ● Open NcEditor;

    ● Menu-> File-> Load DXF file. Before loading DXF file, in Ncstudio, the related "6.5 file para" should be set as following, Ncstudio-> Parameters ->Program Para-> 6.5 File Translation Para

3. Preprocess the figure in NcEditor. E.g. Combine the available figures and set the unified inclined direction of all CAD figures; Set lead-in line, tool compensation data. Be noted that whether the inclined direction of machining figure is consisted with the previous setting direction in combination.

4. Save/ save as CAD figure to ".nce" format file. At the end of manipulation, pls save the machining file again.

5. pls return to mechanical origin first. And user must guarantee that the workpiece coordinate origin is on the workpiece surface and the coincidence of workpiece origin and mechanical origin; Be noted that workpiece origin can not be directly set by clearing method of A/ B axis to prevent the misalignment of workpiece origin of rotary axis and mechanical origin of rotary axis.

6. Begin machining

10 Function and Features of NcEditor

The features of NcEditor are as following:

■ It is designed for planar cutting, editing planar picture and outputting data of five axes machining objects.

■ Mate with Ncstudio and support directly simultaneous processing with Ncstudio.

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

■ Support setting processing condition of single object, distinguished by color, and parameters setting function;

■ Support object change function, including translation, rotation, mirror image, array, chamfer, zoom, slant, combination and breakup of the objective;

■ Support view change function, including translation, zoom in, zoom out, fit to window;

■ Support spot catching function, such as catch node and midpoint of line, catch circle center and quadrant, intersecting point of multi-object, object edge, auxiliary line and characteristic line;

■ Support view gridding and ruler displaying function;

■ Support gridding point catching function, when the gridding catching function is selected, only the gridding point can be caught;

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

■ Support displaying and modification function of attribute of object;

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

■ Support object selection function, including point picking and frame picking;

■ Support auto and manual bridge addition function, currently, this function is unavailable to ellipse, combination and other stretching figures;

■ Support delete of duplicate object;

■ Support inserting of other type of programmable files into existed processing file and display of the existed file and inserted figures in the plot area;

■ Support saving parts of the processing file as another processing file;

■ Support basic editing function: copy, cut and paste, cancel, undo and select all;

■ Support analog processing function;

■ Support loading file format of ENG files, G code files, DXF files and PLT files, and inserting file format of ENG files, G code files, DXF files, PLT files and NCE files. In file saving, all files will saved as NCE format;

- Support Chinese and English bilingual interface, and the internal exchange of the two languages, synchronous language switching with NcStudio;

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

- Support viewing and editing of the 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 [machining parameters attribute column] includes: [machining parameters color window], [machining parameters modification window]. See Fig.10-1.
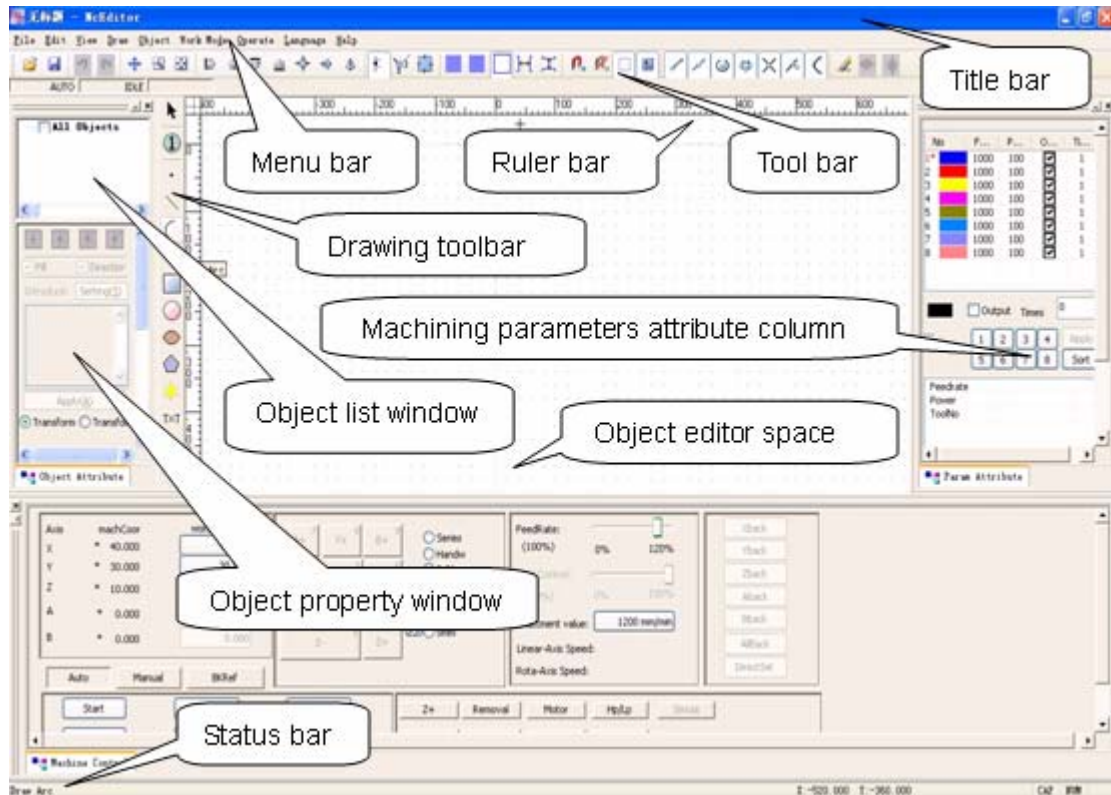
Fig. 10-1 Integral Interface Dialogue

10.2 Object Attribute Window

[Object Attribute Window] is mainly used to set the property of selected object, involving setting Lead-in line, fill or not, the precise position of object in coordinate system, rotation & width setting and setting machining direction, etc.

10.2.1 Classification of Lead-in line

[Lead-in line] is exclusively for closed figures, they can be classified into 4 types according to shape: 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.

a) [None Lead-in line]:

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

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

b)   [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.
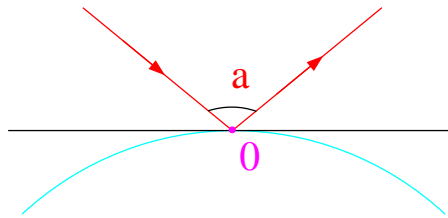
Fig. 10-3 [Tine Lead-in line]

c)   [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, etc.
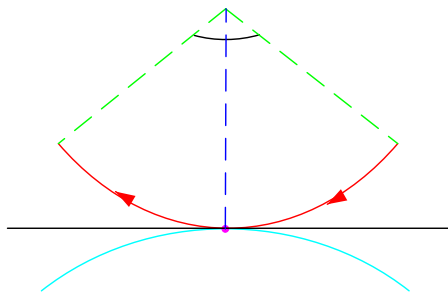
Fig. 10-4 [Arc Lead-in line]

d)   [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 stops after travelling a distance in terms of different sealing length.
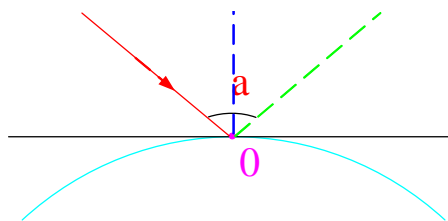
Fig. 10-5 kerf [Lead-in line]
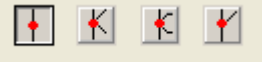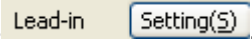
Note: there is no definite limitation on the usage of tine Lead-in line, arc Lead-in line and kerf Lead-in line, which can be interchangeable, for instance, setting the arc Lead-in line of rectangle vertex, the system will figure out automatically the tine Lead-in line; The type of Lead-in line is chosen in terms of the cutting; Besides, the addition of Lead-in line is for precisely machining, user 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 to position, which mainly depends on the reserved parts.

If you select filling the object, [Lead-in line] will be set as the internal one by system, if you select not filling the object, [lead-in line] will be set as the 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.

10.2.2 Set Lead-in Line

Method One: Selecting 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, there are [none lead-in line], [tine lead-in line], [arc lead-in line], [kerf lead-in line]. Users can press the button to choose one of lead-in lines according to cutting process; On the lower parts of the [object attribute window], shown as Lead-in Setting(S). Pressing the button [Setting], if the selected figure is a closed one, system will pop up a dialogue of [lead-in line] setting as following, or there will be no response in system:

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 [setting lead-in line]→ the following 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 [setting lead-in line] option → the following dialogue will pop up
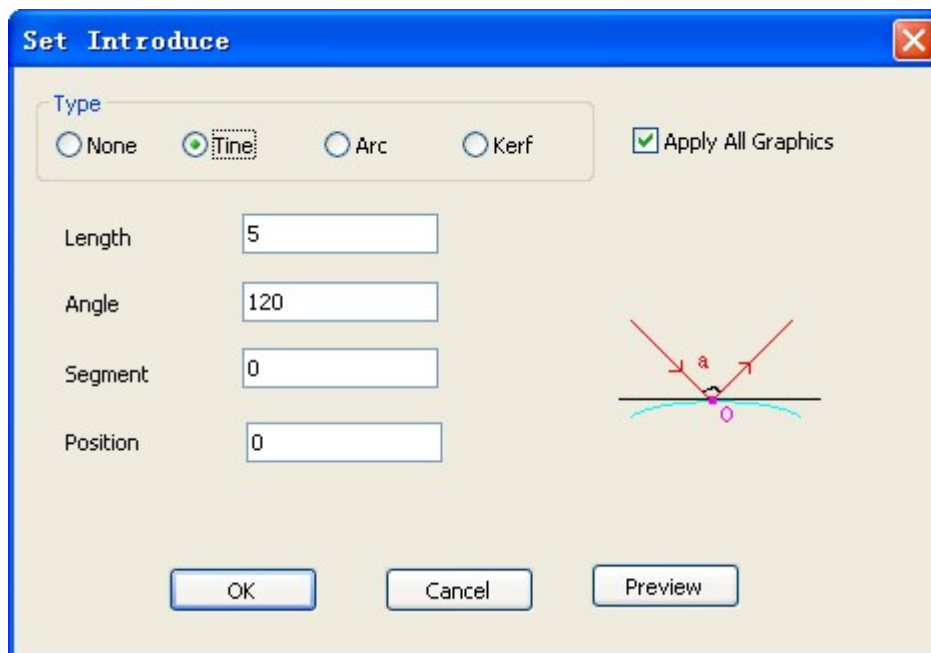
Fig. 10-6 Setting lead-in line dialogue

In the dialogue of [Setting lead-in line], user can set length, angle, segment No. 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, the size of angle is as angle "a" in Fig.10-3, Fig.10-4, Fig.10-5.

➢ [Segment]: there are probable N straight lines (N is natural number) on selected object, all the segment orders on lead-in line segment are called ''segment''. (It is used to fix the location of lead-in line on which segment).

Sort segment: Choosing a closed object, or the combination of multiple objects, there will be a remark on the object. Choosing [view direction] under [view] menu, the direction can be modified in [object attribute window]. Sort begins from the remark, in line with the machining direction. The first order number is "0", and user can set the segment number according to their own desire.

➢ [Position Ratio]: It refers to the ratio of the distance from the position of lead-in line on segment to the beginning of this closed segment to the whole length of the segment, and the 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 fitted for all the graphics (Different range will be displayed for various figures, which depends on concrete situation).
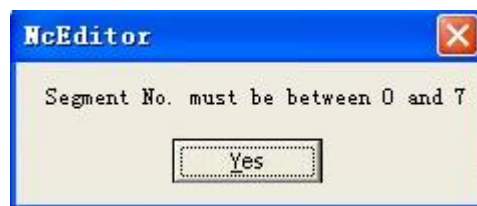
Fig. 10-7 Inquiry Dialogue

After pressing "yes", the segment number that not fits for the graphics will be set as default "0". After pressing "cancel", the unfitted graphics will not be drawn.

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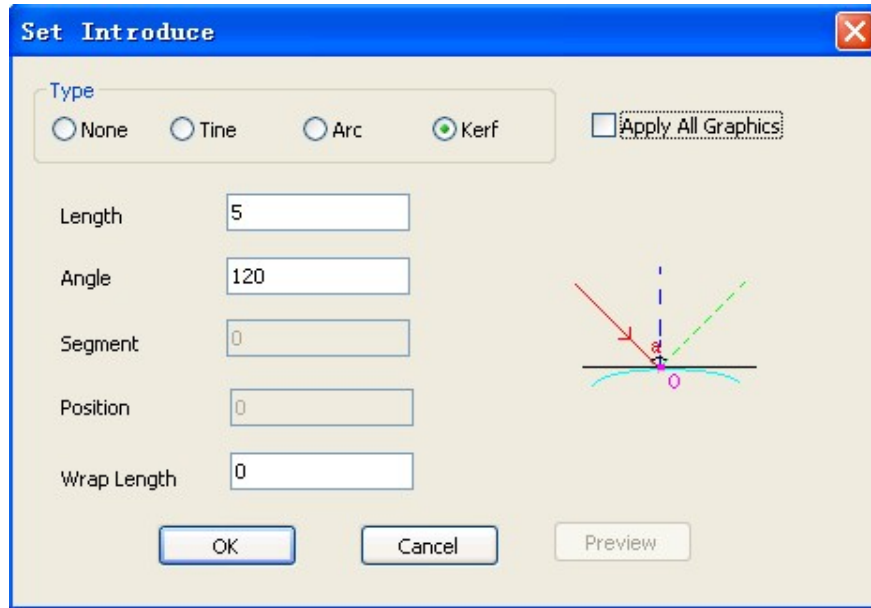
Fig. 10-8 Setting [kerf lead-in line] Dialogue

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

The definition of [wrap length]: when reaching the cut-in point, tool will go along a segment of the same path ahead till stopping, the distance between cut-in point and the repeated path length is called [wrap length].

An example for setting the lead-in line:

Note:
[Lead-in line] is only available in the closed graphics, if user draws the polylines, the graphic can be set lead-in line by finishing with shortcut key "shift+c".

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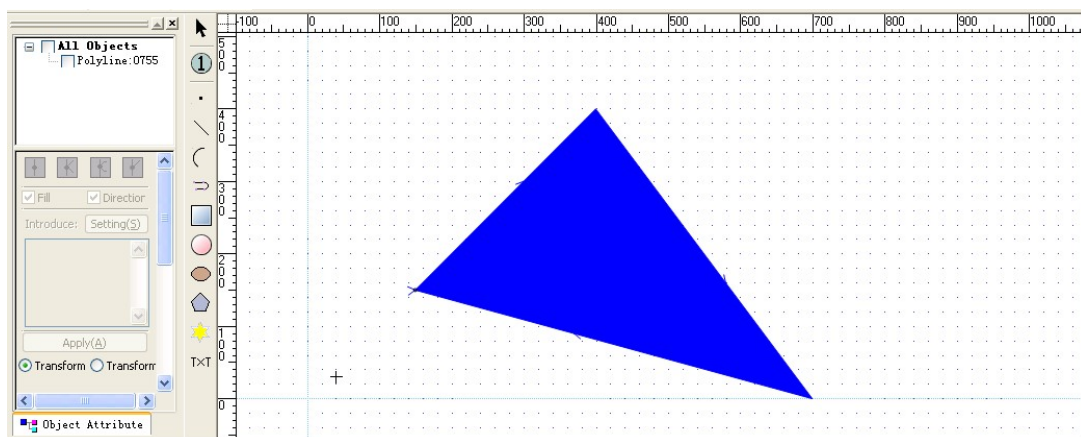
Fig. 10-9 draw closed triangle with polylines dialogue

Select "view" → choose "wireframe model" and "view direction", see Fig.10-10.
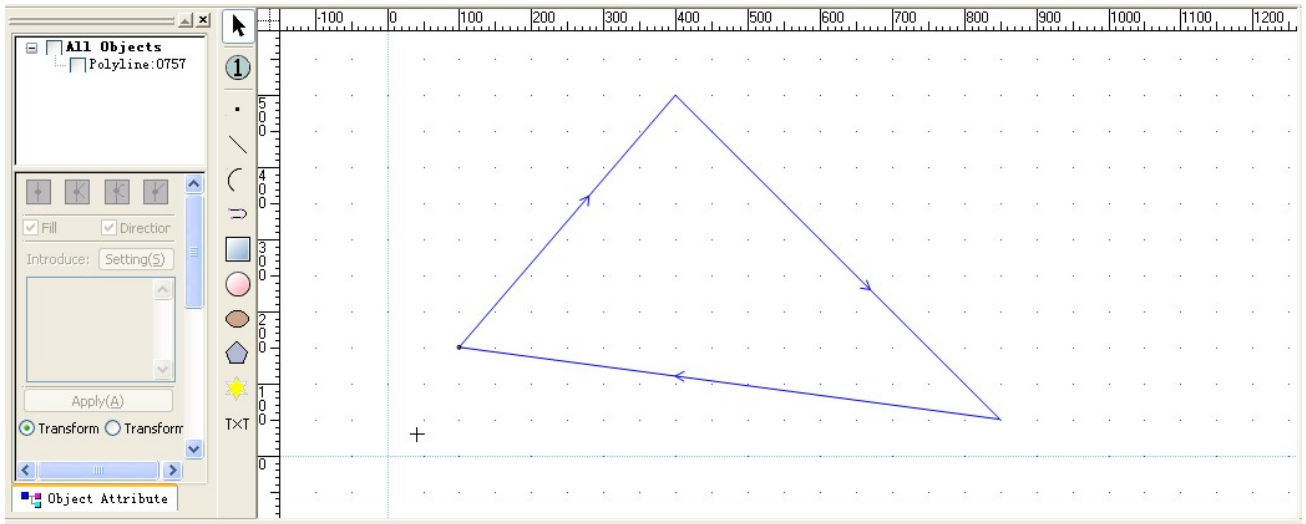
Fig. 10-10 [wireframe model] and [view direction]

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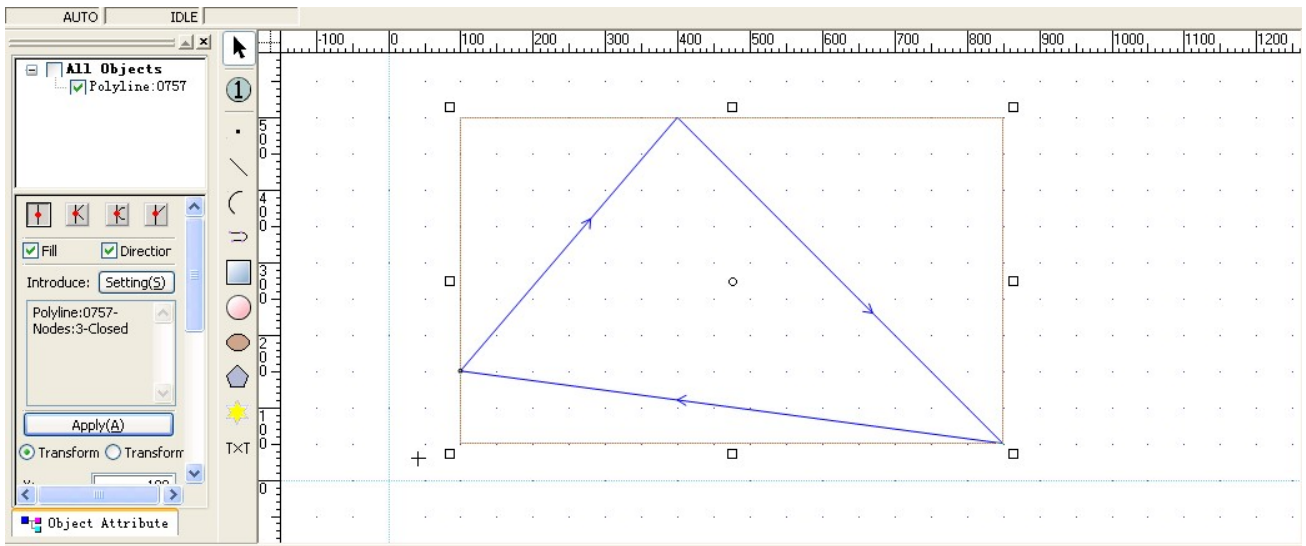
Fig. 10-11Selection of Object Dialogue

Select this closed object → right-click mouse button→ select "setting lead-in line", or [setting lead-in line] under [object] menu or on [object attribute window], input the data as following:
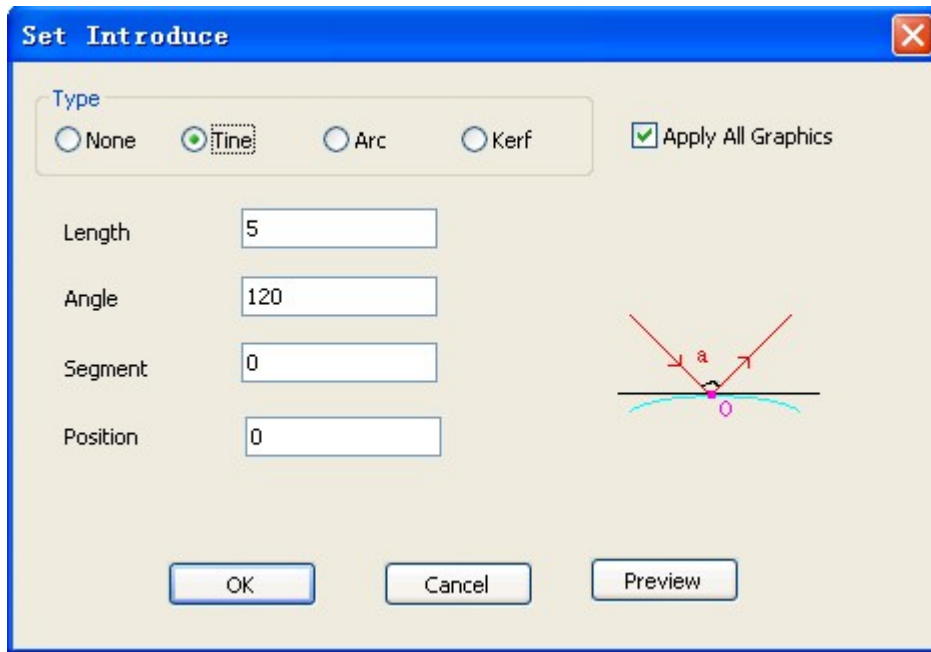
Fig. 10-12　setting lead-in line data

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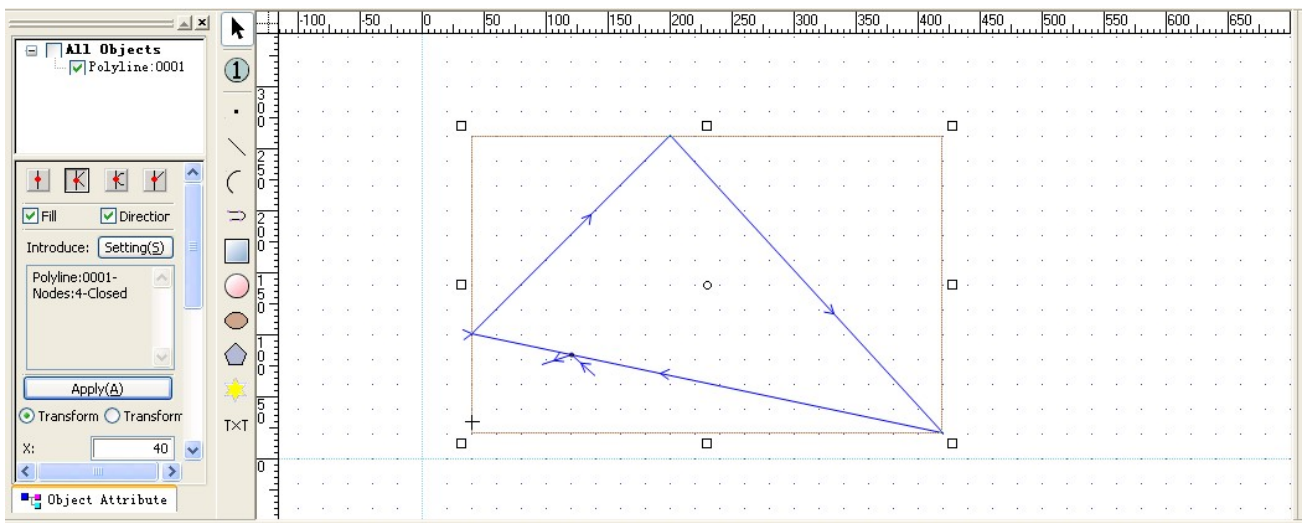
Fig. 10-13 after setting lead-in line

Besides, Nceditor can support manual setting the position of lead-in line. In [setting lead-in line], don't put in any data in [position ratio] of Fig.10-12, and then click the frame of the processing object to set the position of lead-in line.

10.2.3　Color Selection of Machining Objects

Under the [parameter 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], we will introduce the delegation of color corresponding to related parameters in detail.

◆　Operation procedures:

- User can select an object by clicking, or, using the button on [drawing toolbar] (only the object totally in the frame is selected);
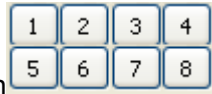
- Press one button in , the selected object will change into the color you chosen.

The objects with the same color are regarded as the default same parameters, including velocity, power and cutter No., etc. so, users have to set all the objects of the same parameters in the same color.

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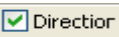
[Fill or not] of the objects are only for the closed objects.

◆ If user wants to keep the inner parts of objects, please choose [fill], shown as ;

◆ If user only wants the shape of the objects, please select not fill, shown as , the inner parts of the objects will be milled.

10.2.5　Selection of Machining Direction

[Selection of machining direction] is used to change the machining direction of object, and user can adjust it according to cutting method, texture of workpiece, 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 of the object;

➢ User can select an object by clicking, or, using the button on [drawing toolbar] (only the object totally in the frame is selected);

➢ In [object attribute window], choose , the selected object will reverse the direction.

➢ If user wants to cancel the alteration of direction, you just need to cancel the selection of the choice box, see .

10.2.6　Precise Positioning of Machining Objects

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

The concrete example: Drawing a circle in [object editor space], then select this circle, as following:
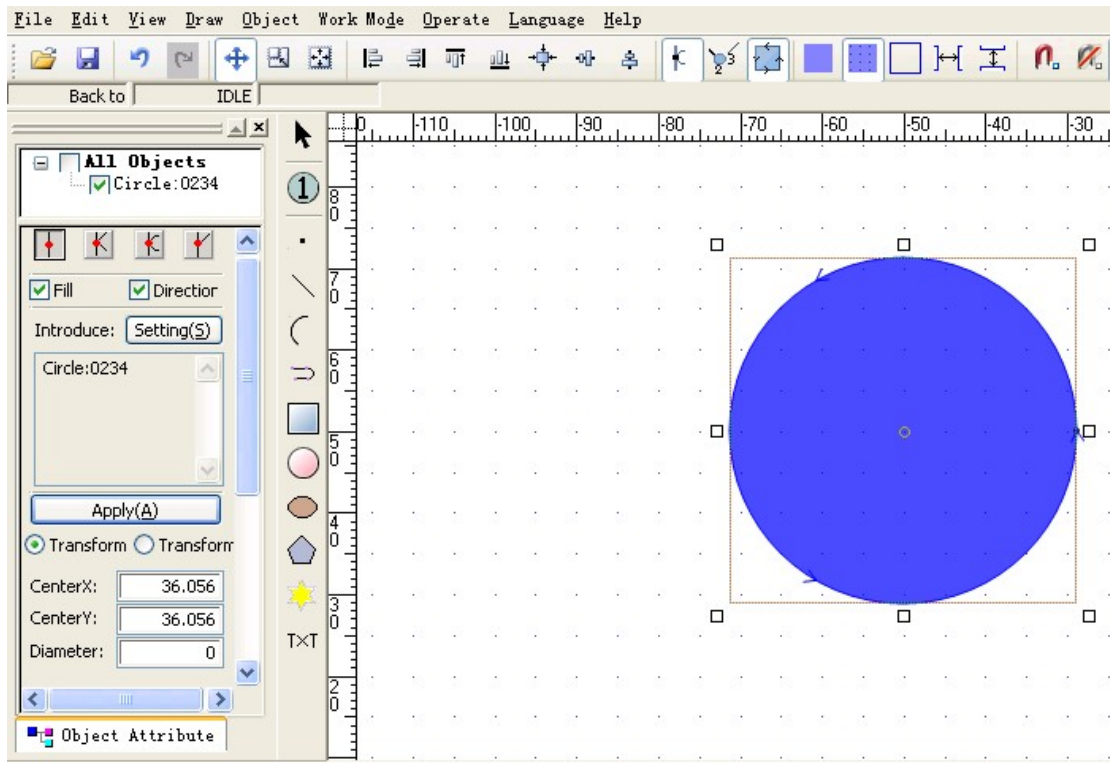
Fig. 10-14 positioning a circle

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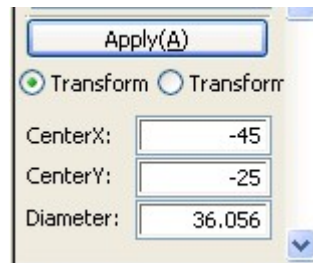
Fig. 10-15 precise positioning window

After inputting the data, press "Apply", and then this circle can be precisely positioning the position centered XY (-45,-45), radius 15.028.

If the selected objects are straight-lines, rectangle, arc and point, the precise positioning dialogue will alter accordingly;

10.2.7   Object Information Diagram

When users select one or multiple objects in [object editor space], the [object information dialogue] window will display the detailed information of objects, such as mane, node and fill or not. But the [Fill] option is unavailable (grey) for not closed figure, and user can check the detail information of object to avoid misunderstanding of object attribute under different view.

For instance, draw a straight line and a closed polyline, and select them, as Fig.10-16.
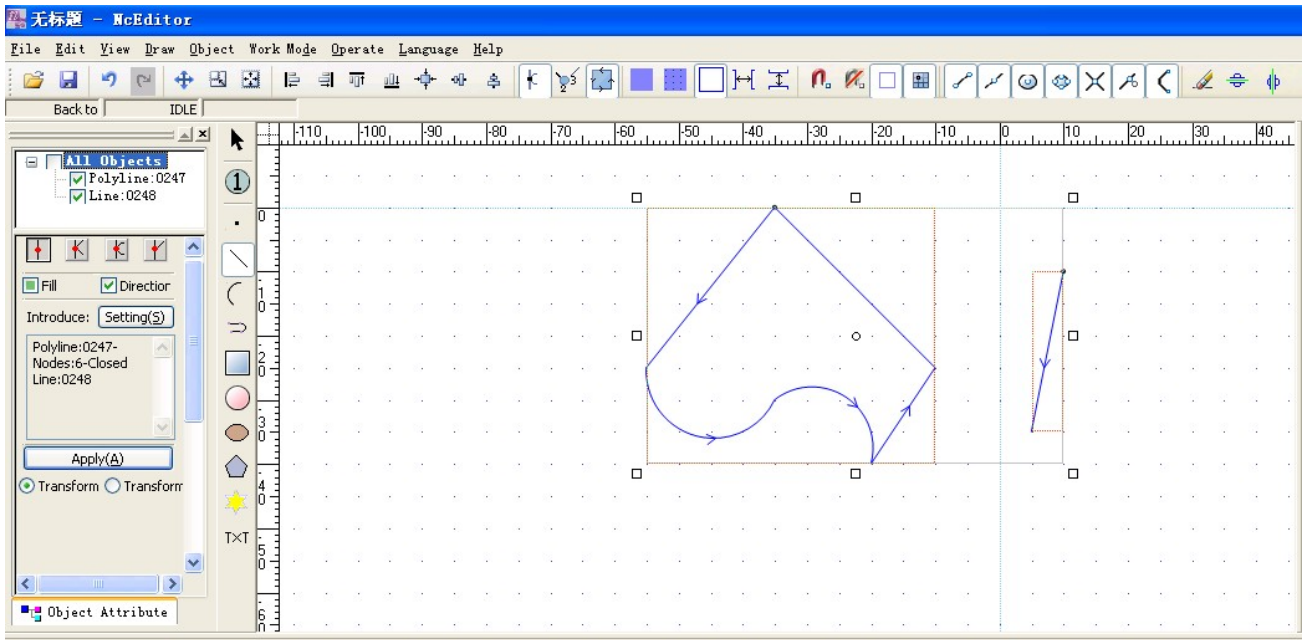
Fig. 10-16 the object drawn by polylines

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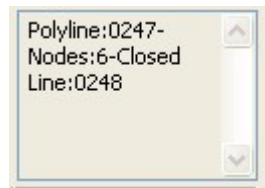
Fig. 10-17 object information dialogue

10.3 Origin setting

The criteria 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 remark , click the cross center and drag it in [object editor space] to release it in the required origin position, which is also the origin of workpiece coordinate.

The advantage of setting an origin:

➢ The position of object in workpiece coordinate can be more precise positioning;

➢ When loading a file, the workpiece position in workpiece coordinate can be more precise positioning;

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

10.4 Machining Parameters Attribute Bar

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

10.4.1　Color Window of Machining Parameters

There are currently 8 color in color window with corresponding number, each color is corresponding to different machining parameters, including velocity, cutter No., etc.

The color shown in highlight is the default color of next drawn objects.

10.4.2　Modification Window of Machining Parameters

Machining parameters of same color are regarded as the same machining parameters under two-dimension mode in NcEditor, so if user changes the parameters related to certain color, the machining parameters of all the objects with this color will be altered.

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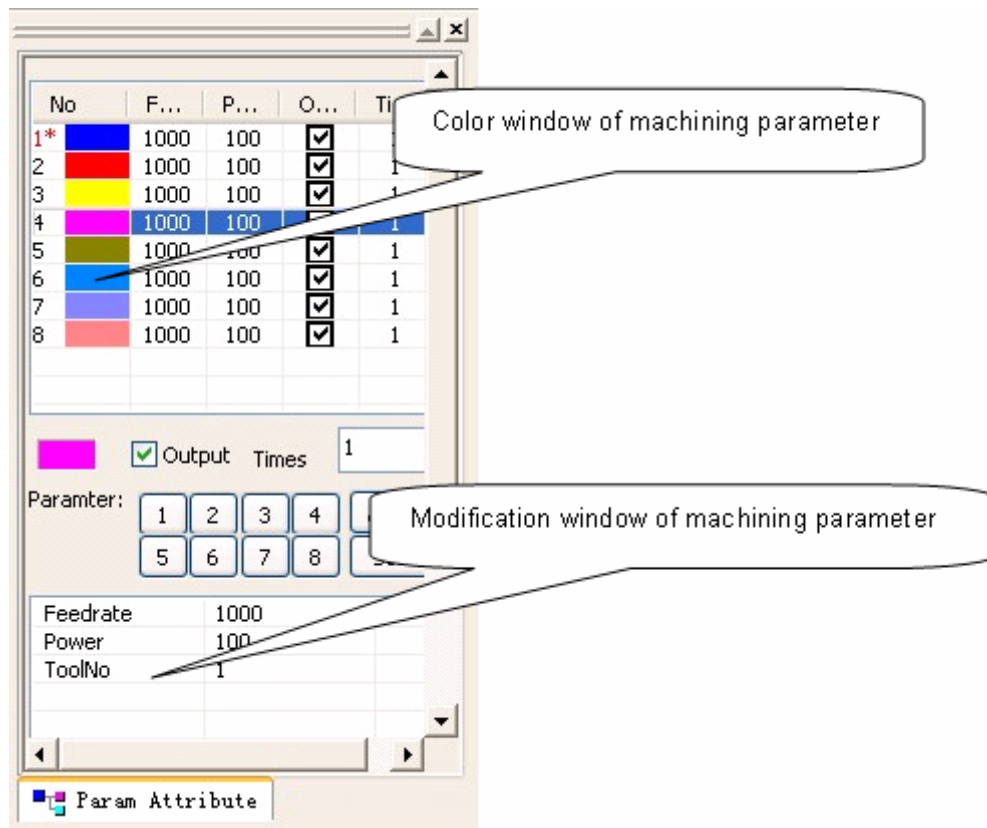
Fig. 10-18 machining parameters attribute bar

In [modification window of machining parameters] , the related parameters of color 4 can be modified directly, such as velocity, power, input and output, press" apply" to confirm the modification.

10.5 Figure Selection

Selecting a figure, users must ensure that there isn't other running command, holding down the button, shown as ⬚, the objects only total in the box are selected.

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

➢ Click the frame of an object to select this object;

➢ Clicking the left mouse button and moving the mouse, there will be a rectangle box in the space to

ensure the selection of object frame.

If users want to select multiple objects or object group, following method can be used:

➢ Click the object, and press the button shift/ctrl at the same time, new object can be added into the selections.

➢ Holding down the left mouse button and move the mouse, there will be a dashed rectangle box in the space. The object total in the box is selected, and there will be red dashed rectangle outside the selected object. Besides, after selecting the object, user can holding down the left mouse button to drag the object to the required place.

10.6 Menu Function Section

Menu function that is same with ncstudio will not be introduced any more.

10.6.1　File Menu

[File menu] includes every operation command related to machining file, as Fig.10-19:
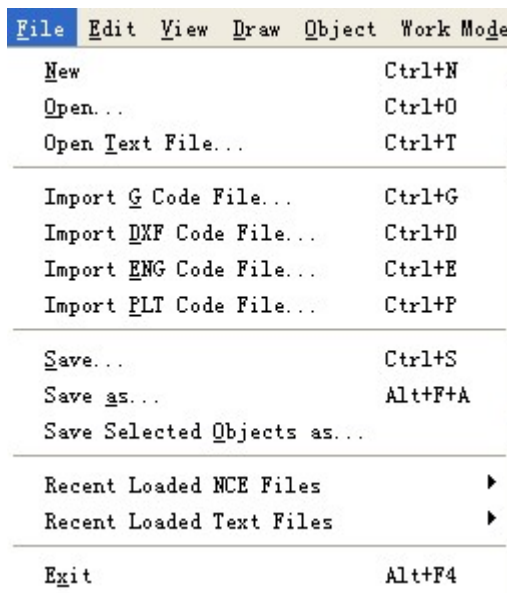
Fig. 10-19 File menu of NcEditor

◆ [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 parse engine will parse the file into G instructions, and transfer these instructions to CNC card.

[Open dialogue] will pop up, as Fig.10-20. Press "open" after selecting the machining file, which will be loading automatically, and file name will display in the operation panel. Users can view it clearly, since the file name will appear on the operation panel.

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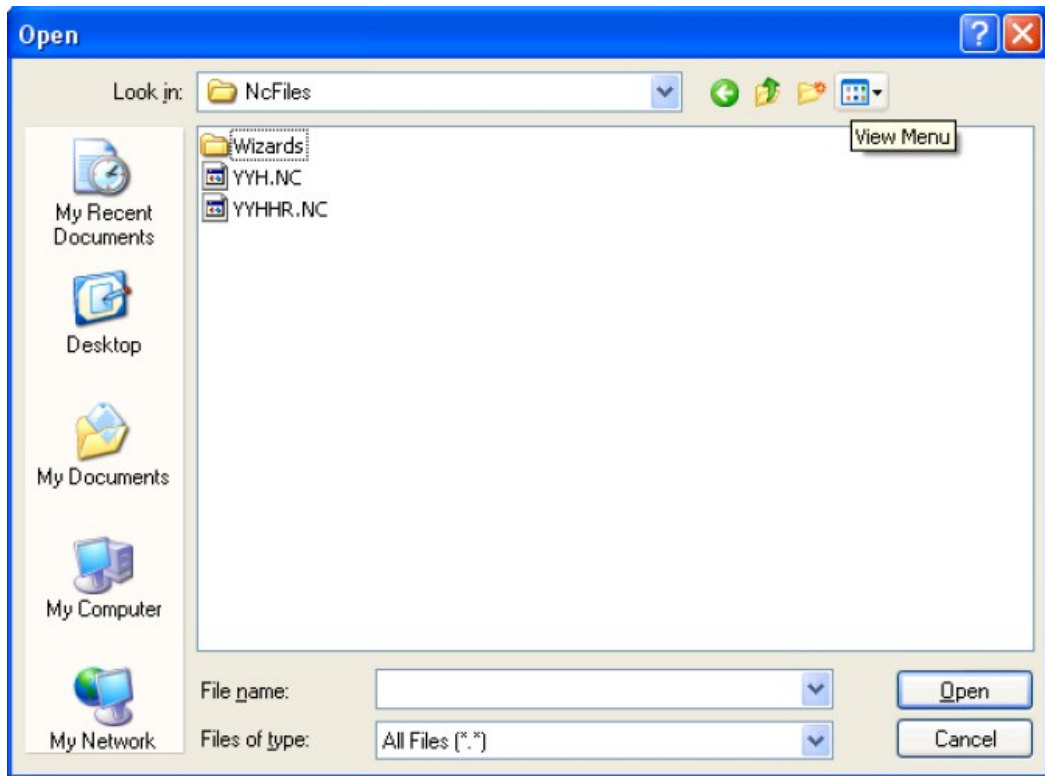
Fig. 10-20 [open] dialogue

◆　[Open text file]

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

◆　[Load G code format files]

Shortcut key: Ctrl+G, the function of this menu is to add machining object of G code file on the basis of opening a 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 code file, ENG code file and PLT code file in NcEditor is the same as above.
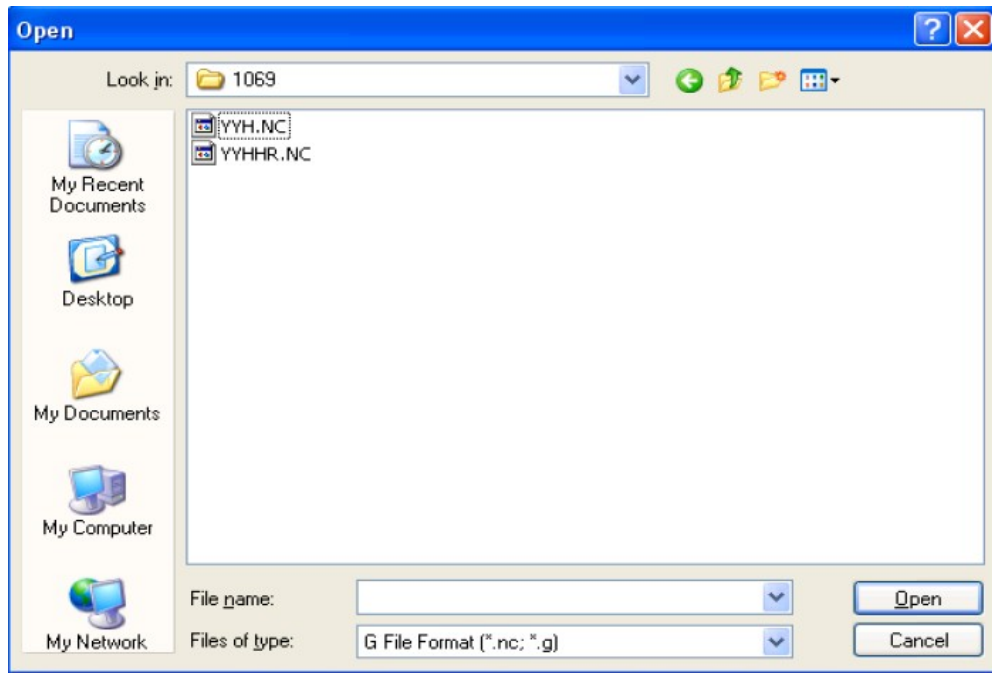
Fig. 10-21 [Open G code format files] dialogue

◆ [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 the selected figure]

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

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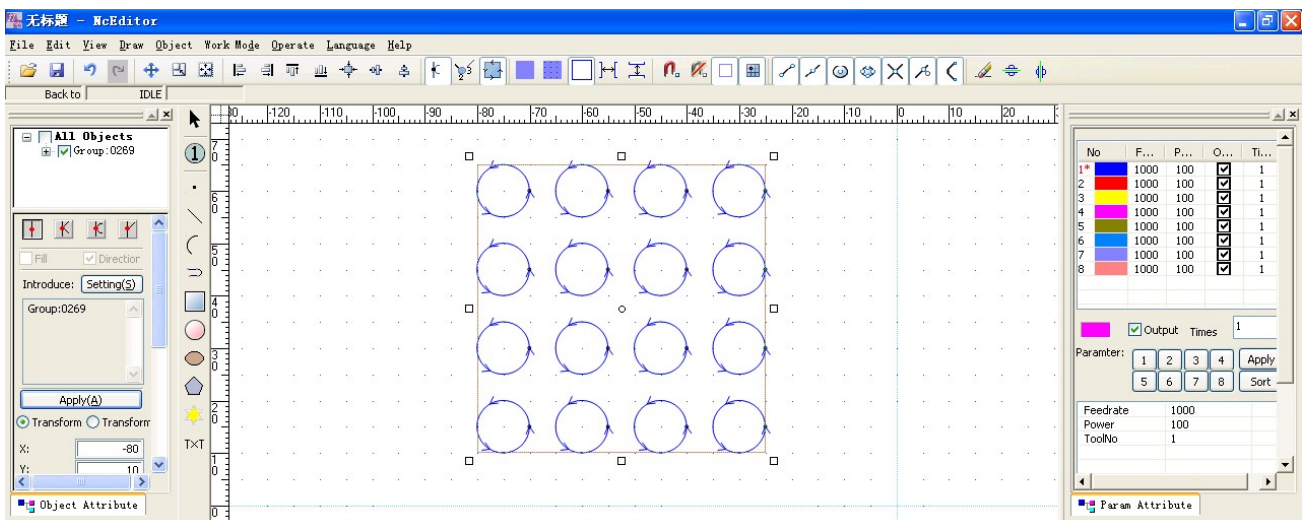
Fig. 10-22   Selection of figure Dialogue

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

◆ [Recent loaded NCE files]

This function enables user to view recent loaded NCE files and open these files.

◆ [Recent loaded Text files]

This function enables user to view recent loaded Text files and open these files.

◆ [Exit]

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

10.6.2 Edit Menu

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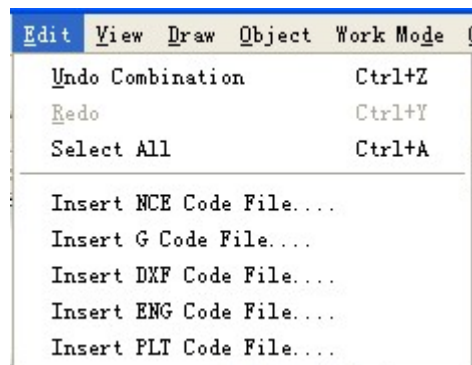
Fig. 10-23 Edit Menu Picture

◆ [Undo/ Redo]

The software is in possession of strong function of undoing and redoing, in order to save all the operation histories. Users can undo and redo continuously former several manipulations until satisfied, in order to avoid misoperation.

Users 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], to avoid continuously selecting objects.

◆ [Insert files]

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

Methods are as following:

● Open or draw a machining file;

● Select [Import DXF code file] under [File] menu, [opening file dialogue] will pop up as Fig.10-24.
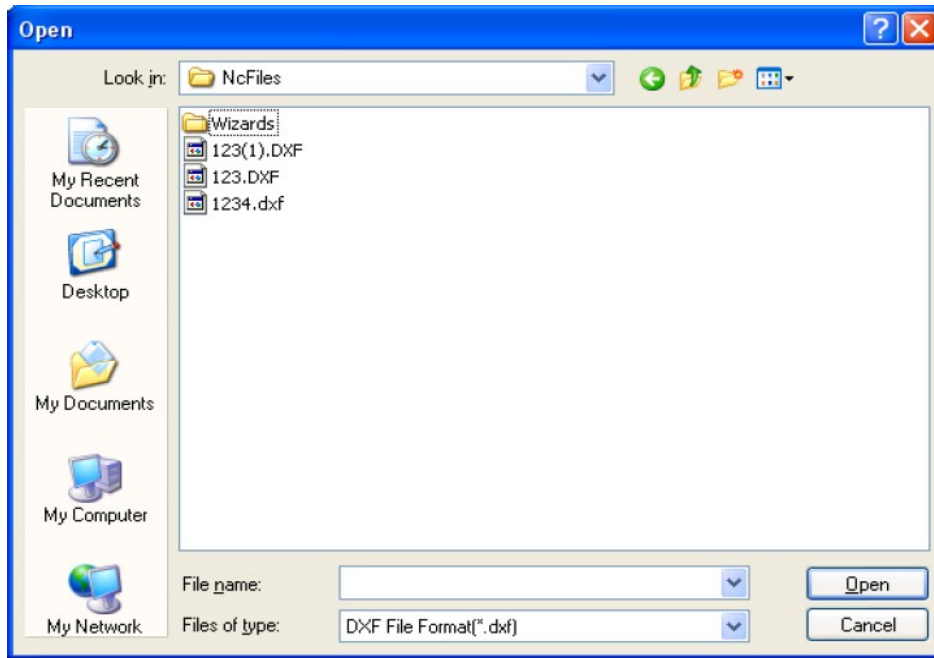
Fig. 10-24 open file dialogue

The method of inserting G code file, NCE file, ENG file is the same in NCEDITOR

10.6.3 View Menu

The main option of view menu is: toolbar, status bar, object list, draw toolbar, parameters bar, view translate, view zoom, catching option, etc, as Fig.10-25.
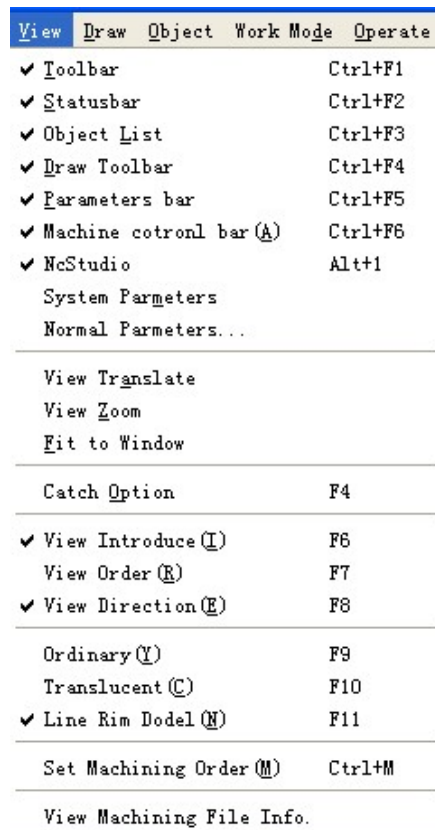
Fig. 10-25 [view] menu picture

◆　[Toolbar]

This function is used to display/hide the toolbar.

◆　[Status Bar]

This function is used to display/hide the status bar.

◆　[Object List Bar]

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

◆　[Draw Toolbar]

This function is used to display/hide the draw toolbar.

◆　[Parameters Bar]

This function is used to display/hide the parameters bar.

◆　[Ncstudio]

This function is used to display/hide the interface of Ncstudio

◆　[Machine Control Bar]

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

◆　[System Parameters]

This function is used to display/hide the system parameters.

◆　[Normal Parameters]

This function is used to display/hide the normal parameters.

◆　[View Translate]

User can reposition the figure to view clearly the other parts of the figure, this command will not change the position in the coordinate or ratio, only the view. User can also translate the figure up, down, left and right.

● Method one: open [view] menu, select the option [view translate (A)];

● Method two: use the translate button ✛ on toolbar.

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

◆　[View Zoom]

User can zoom the track by menu + mouse or toolbar + mouse, the actual size of the figure won't change, only the view zoom in or out.

● Method one: open [view] menu, select the option [view zoom];

● Method two: use the [view zoom] button 🔍 on toolbar.

After selecting this option, user should slide the mouse wheel in the [object editor space]. Sliding forward means zoom in, while sliding backward means zoom out.

◆ [Fit to window]

After zooming and translating, if user wants to fit the view to window, this command can be worked.This function can make user 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: use the button [▣] on toolbar.

◆ [Catching option]

[Catching option] is used to better position the feature point of some figures during drawing objects, so user can easily catch the objects.

Select [View] menu → [Catching option] or pressing the icon [🔒] ([🔒] catching off) on toolbar, the [catch option] dialogue will pop up.
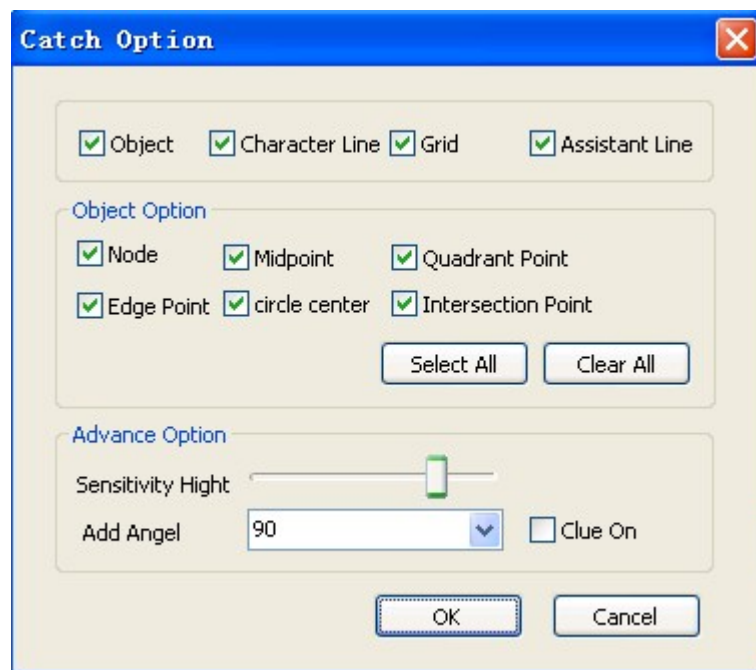
Fig. 10-26 [Catch option] Dialogue

If the object option is selected, the corresponding object will appear in the [object editor space], or it will be concealed.

1) The feature 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;

2) The advantage of [Catching option]

User can directly use cursor to position the aimed point precisely (such as the above character point in figure) without knowing the concrete coordinates, which can connect the figures more accurately. This function will be used in following [Combine object].

Note: In order to better use [catch option] function, please closed the function [catching grid].

For instance: how to introduce another straight line from the midpoint of a straightline.

➢ Select the option of [Catch object] under [view] menu, or choose the icon 🔲 on toolbar.

➢ Drawing a straightline in [object editor space], a little pane will appear when the cursor is on the straightline near the midpoint, this is the caught character point (midpoint of straightline), so user can find the midpoint accurately, clicking the pane to introduce another straightline from the midpoint.

3) [Catch grid]

The advantage of [Catch grid]

[Catch grid] can execute integral positioning on the objects, which can make the point in the grid of [object editor space] automatically. The default unit of grid in NcEditor is "1", and there will be catching grid if encountering integral crossing in the direction of X, Y, which is more precise than other grid positioning software. The coordinate of grid point will be display in the status bar precisely.

The manipulation method:

Select the option of [Catch grid] under [view] menu, or choose the icon 🔲 (🔲 catching off) on the toolbar.

Note: [Catching grid] is available only under the condition that drawing function is activated (One option is selected under drawing menu)

4) [Sensitivity], [add angel], [character line] and [assistant line]

[Sensitivity] 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 user.

[Add angle] is to catch object with a little angle, for instance, choose "45", when drawing the straightline, this line will be caught in moving at position that horizontal contained angle equals 45 after drawing the first point.

Referring to the straightline from ruler to [object editor space], [assistant line] is used to assist the user in limiting the position of drawn object.

◆　[View lead-in line]

This function is used to display/ hide the lead-in line.

User can decide whether to add lead-in line or not as well as the type of lead-in line according to the cutting method. User can refer to [setting lead-in line] function for detail.

◆　[View order]

This function is used to display / hide the machining order of current object.

◆　[View direction]

This function is used to display / hide machining direction.

◆　[Drawing mode]

In order to make user more convenient in drawing, this software set three views, including ordinary, translucent and wireframe mode.

5)　[Ordinary view mode]

It is used frequently, when there are many layers, the overlapping parts are concealed, shown as Fig.10-27.
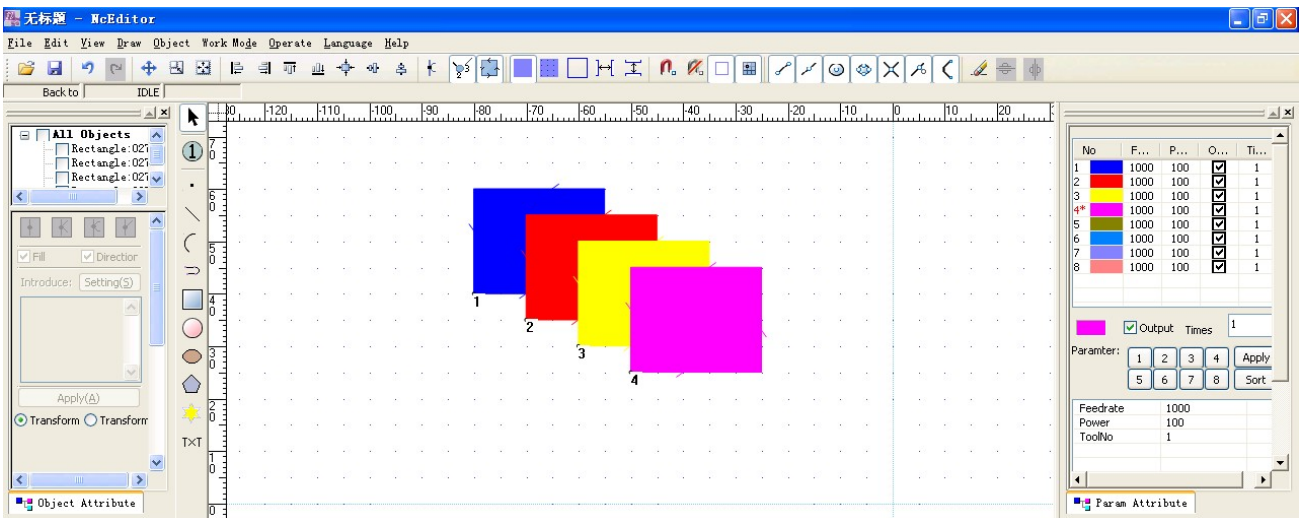
Fig. 10-27 Ordinary view mode

◆　[Translucent view mode]

In order to make drawing more convenient, NcEditor can make layer translucent, the backward grid background or overlapping parts will be concealed, shown as Fig.10-28.
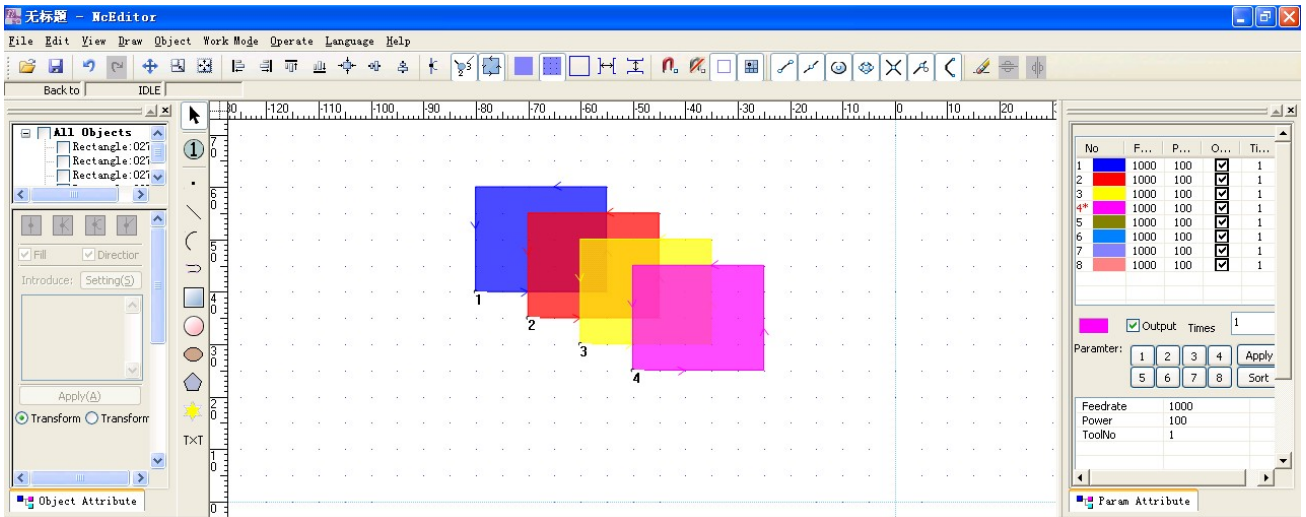
Fig. 10-28 Translucent view mode

◆ [Wireframe mode]

This is the default view mode, all the figures are shown in wireframe mode, the overlapping parts are revealed, as Fig.10-29.
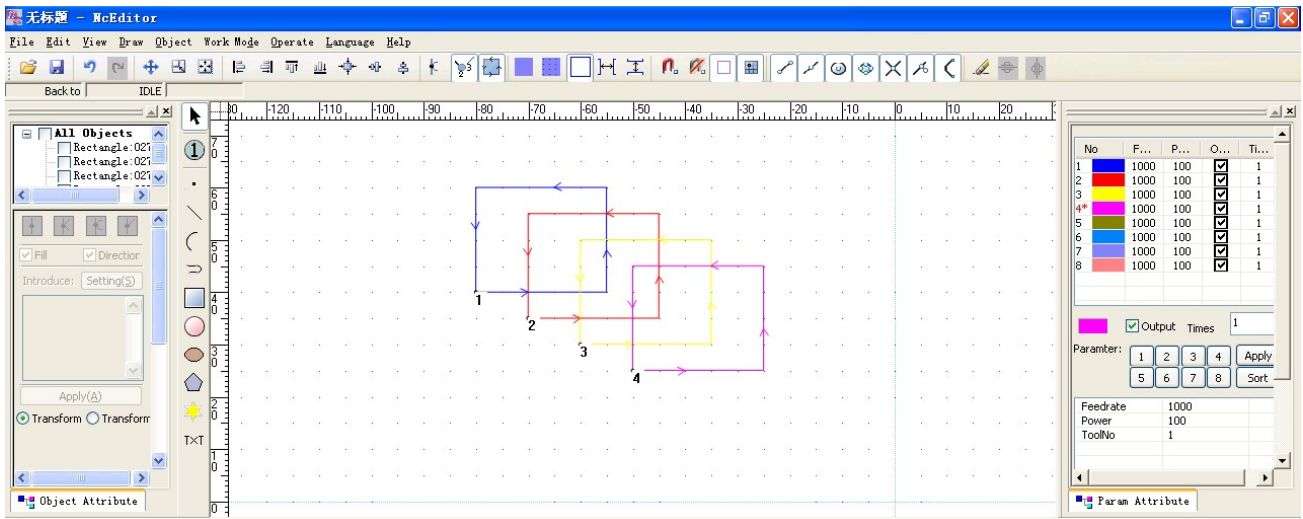
Fig. 10-29 wireframe view mode

◆ [Machining File Information]

The file information can be checked after machining. After clicking the option [Machining File information], users can know the information of file name, cutting time and dry running time, cutting length.

After finishing machining, click the option [Machining File Information], the dialogue will pop up as following, Fig.10-30.
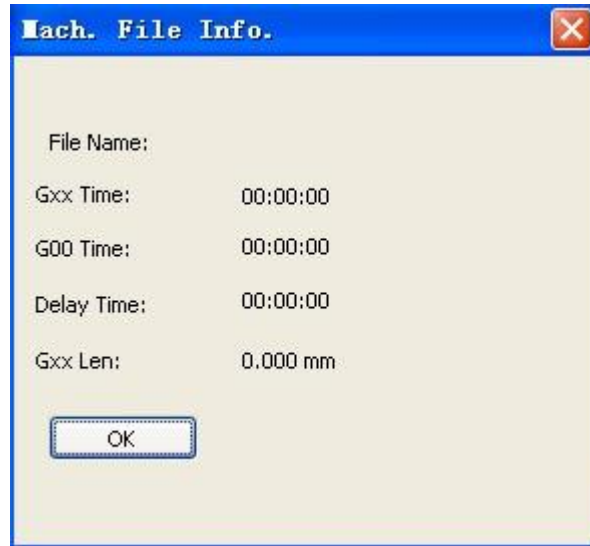
Fig. 10-30   Machining file information

There will be information shown after machining.

10.6.4   Drawing Menu

Drawing menu is used to draw the ordinary figures, including point, straightline, polyline, rectangle, circle, ellipse, arc, and photo. All the functions are corresponding to the buttons on the drawing toolbar, and users can execute manipulation by buttons on toolbar. See Fig.10-31.

Fig. 10-31 Drawing toolbar

Note: The following drawn figures can be accurate positioning through object properties window until users click the right mouse button for completion of drawing.

◆ [Point(D)]

The simplest drawing in [object editor space] is drawing a point.

Select the option [Point] under [Draw] menu or press the button on toolbar→ click the left mouse button in the selected place of [editor space] to draw a point

If users need to draw the point more precisely, they can input the coordinate of the point in [object attribute window], and then pressing [Enter] or [Apply].

◆ [Straightline]

If user wants to draw a straightline, they can select the option [Straightline] under [Draw] or single-clicking icon ＼.

Two points can determine one straightline, drawing two points in [object editor space] as the beginning and end points of straightline.

After selecting the object, user can input the beginning and end points in [object attribute window] to draw the straightline accurately.

◆ [Arc]

If users want to draw a arc, they can select [arc] under [Draw] or single-click icon ⌒.

NcEditor ensures the position and size of arc by radius, arc center, beginning angle and end angle.

After selecting the figure, user can accurately draw an arc by inputting circle center coordinate and radius, beginning angle, end angle in [Object attribute window].

> Note: 1 rad= 180/π degree

◆ [Polyline]

Polyline refers to different straightline and curves combined by connection of arc from beginning to end. The manipulation of multiple lines and arcs can be finished at one time, which shortens the operating times and raises speed of modeling.

If users want to draw a polyline, they can select [Polyline] under [Draw] or single-click icon ⇁.

Shortcut key: arc→ Ctrl+ A; line→ Ctrl+L; Closed the figure→ Ctrl+ C; Finishing Drawing→ Ctrl+F; Esc→ cancel drawing.

> Note:
> In drawing the polyline, the input state should be English.

◆ [Rectangle]

If users want to draw a polyline, they can select [Rectangle] under [Draw] or single-click icon ▢.

NcEditor determines the position and shape of rectangle by the vertex coordinate, width and height on the left corner.

After selecting the figure, user can accurately draw a rectangle by inputting vertex coordinate and height on the left corner of [Object attribute window].

◆　[Circle]

If users want to draw a circle, they can select [circle] under [Draw] or single-click icon 　.

NcEditor ensures the position and size of circle by radius and circle center.

After selecting the figure, user can accurately draw a circle by inputting circle center coordinate and radius in [Object attribute window].

◆　[Ellipse]

If users want to draw a ellipse, they can select [ellipse] under [Draw] or single-click icon 　.

NcEditor ensures the position and size of ellipse by circle center and radius of X direction and Y direction.

After selecting the figure, user can accurately draw an ellipse by inputting circle center coordinate and radius in [Object attribute window].

◆　[Polygon]

If users want to draw a polygon, they can select [Polygon] under [Draw] or single-click icon 　.

NcEditor ensures the position and size of polygon by radius, center X and center Y, edges.

After selecting the figure, user can accurately draw a polygon by inputting the related value in [Object attribute window].

◆　[Star]

If users want to draw a star, they can select [star] under [Draw] or single-click icon 　.

NcEditor ensures the position and size of star by radius, center X and center Y, angles.

After selecting the figure, user can accurately draw a star by inputting the related value in [Object attribute window].

10.6.5　Object Menu

◆　[Delete Object]

The object should be selected before deleting.

Following three methods are used for deleting objects:

➢　After selection of object, user should right-click for menu, choosing [delete object];

➢　Select [Delete object] under [Object];

➢ Use the button [Delete] on keyboard.

User can select all the objects to delete them by shortcut key: Ctrl+A, then pressing [delete] button.

◆ [Combine objects]

> Note: the points of unclosed objects don't connect together because of misoperation or other reason, when the figures should be unified, user can use [Object combination] to combine several unclosed objects (straightline, arc and unclosed polyline) with nearby endpoint (<0.1) to single path object (polyline or path). It is suggested to use [catching object] before using [object combination].

For instance, combine two straight lines.

First, draw the first line 0001. And then, user can open [catching object] to precisely position the endpoint of first line when drawing the second line 0002. Then user can select option [combine object] under [Object] or right-click the menu to choose [combine object]. See Fig.10-32.
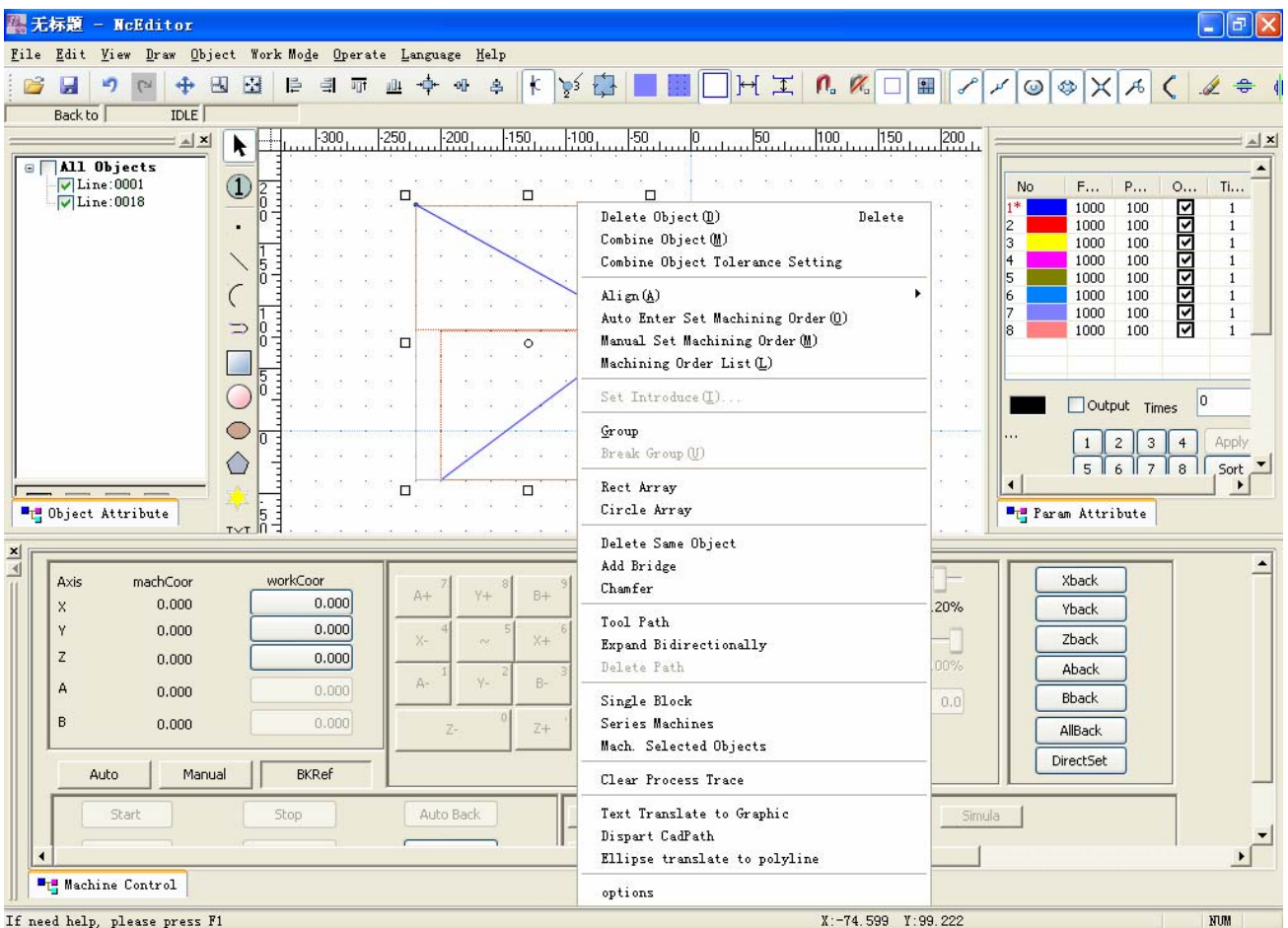
Fig. 10-32 Manipulation of combination of two straight lines

The output of combination of two straight lines is as following, Fig.10-33, the two objects are combined to one single object (polyline).
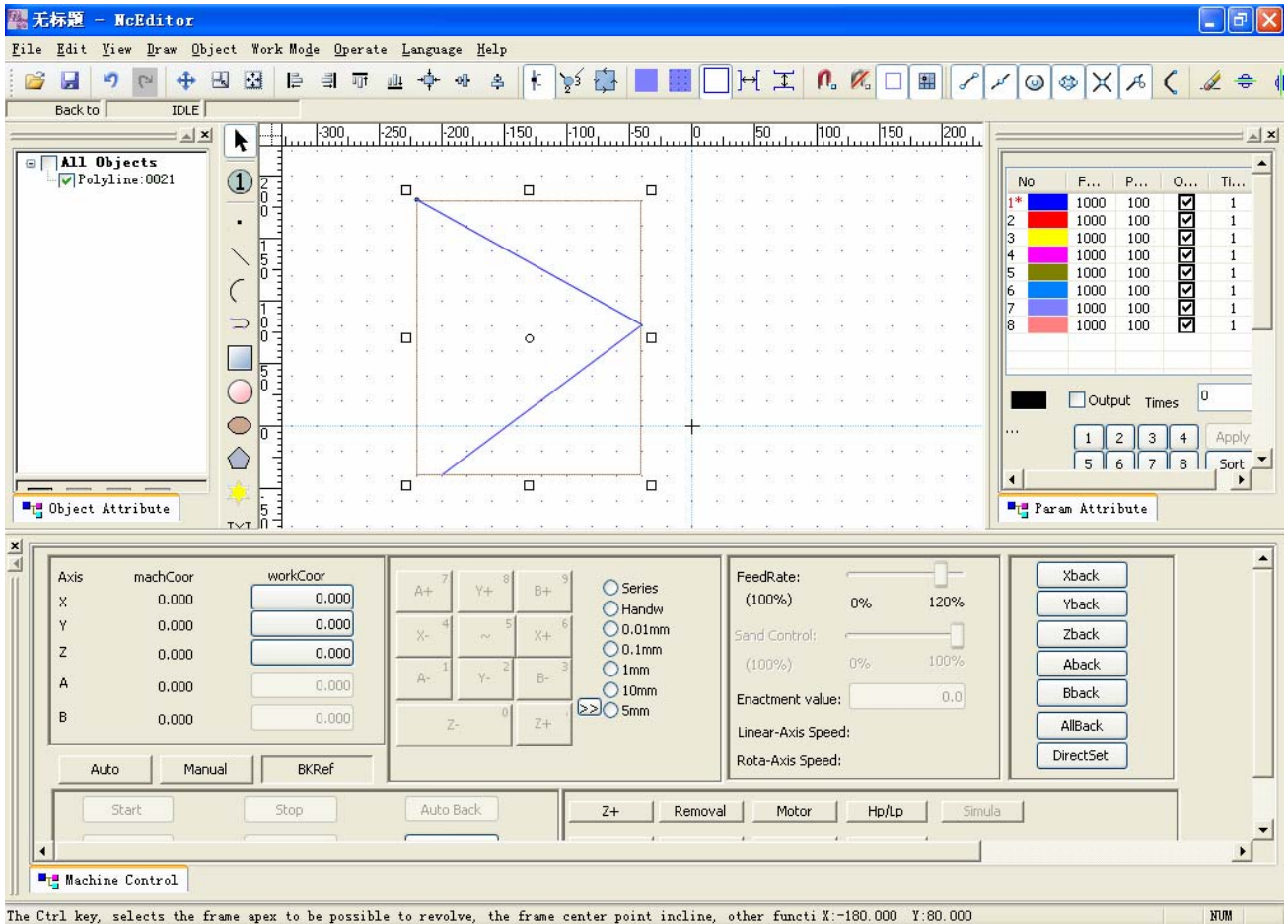
Fig. 10-33 output of the combination of two straight lines

In combination of straight lines, arc and unclosed polylines, user can select [catching object] function to catch the endpoint of figures in drawing to improve the effective of [combine object].

Note: in combination of two or multiple arcs, the combination of arc will begin from the origin object counter-clockwise.

◆ [Setting combine tolerance]

In combination, users need to combine objects following certain criteria, requiring setting combine tolerance (the objects combine at such distance), so the software offers the function of setting combine tolerance.

The following dialogue will pop up in selecting [combine object tolerance setting]. User only needs to input a number between 0.1-5 into the input dialogue.
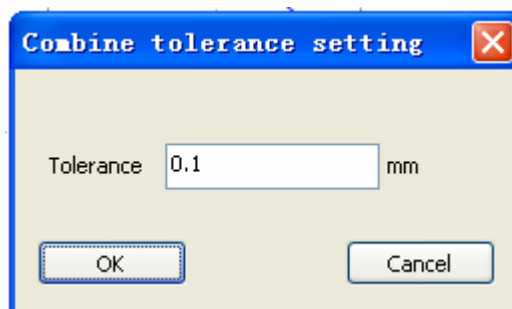
Fig. 10-34 setting combine tolerance

◆ [Align]

When users choose above two objects in [Object editor space], the [Align] is enabled, this function can align all the selected objects in two-dimension plane.

Note: the following align function is on the basis of top, bottom, center point, left and right of the rectangle box for selection of figures.

See Fig.10-35, all the following align functions are moved on the basis of the two objects.
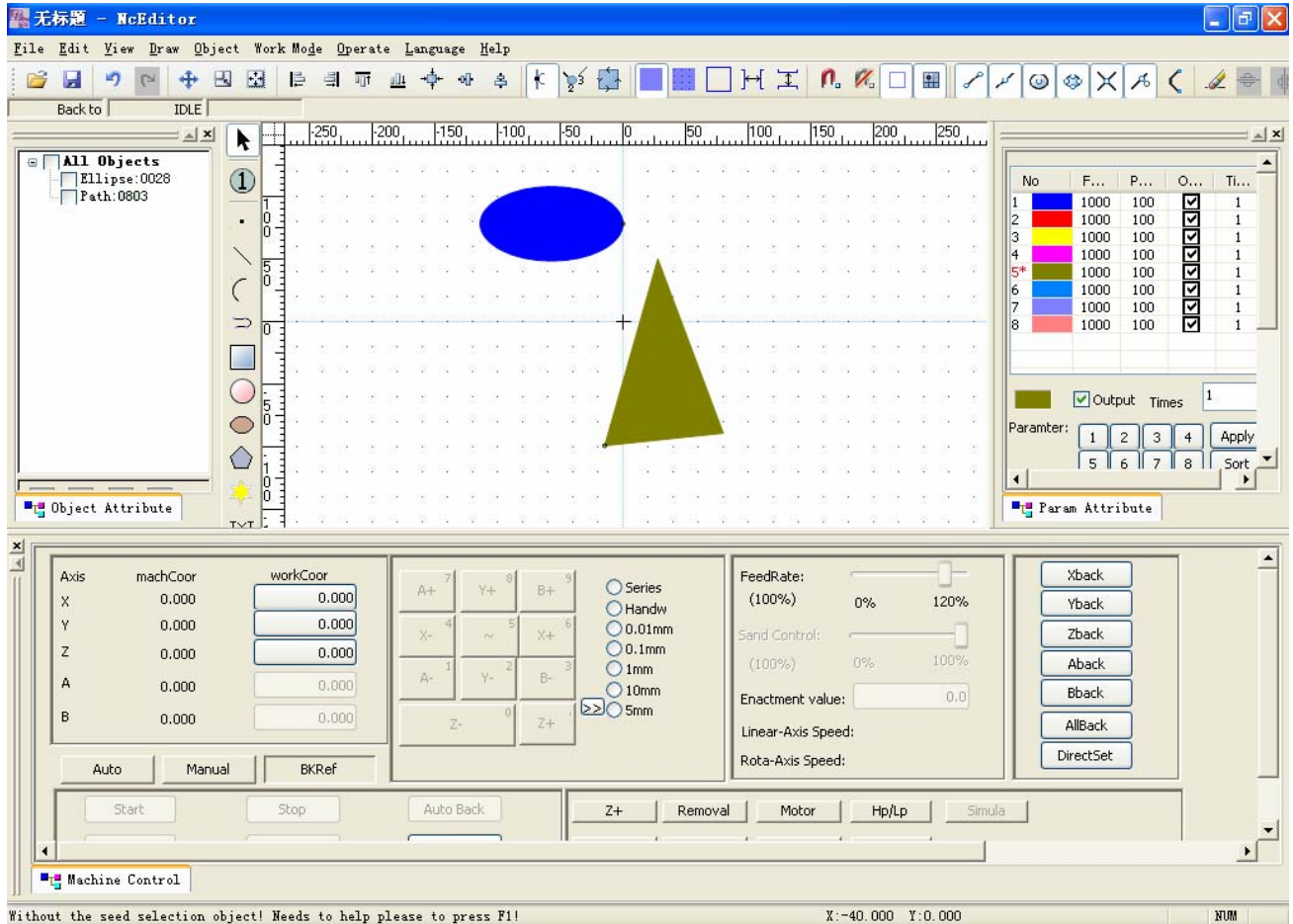
Fig. 10-35 two objects used for alignment

◆ [Align left]

The left edge of rectangle box in selection of figure is regarded as the remark for alignment.

After selecting the object, 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;

➢ Pressing the [align left] button on the toolbar

After executing [Align left], the view is as following:
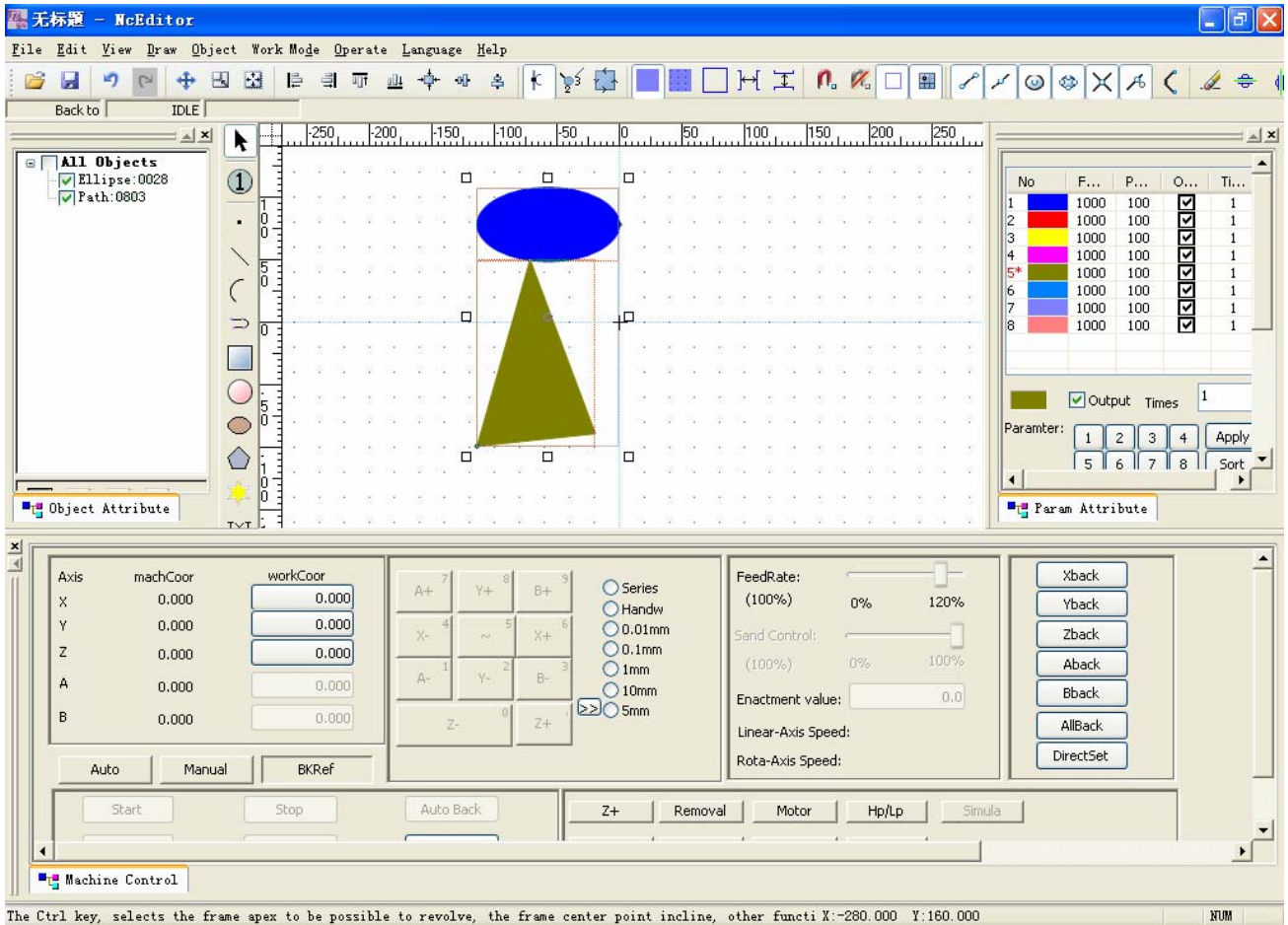
Fig. 10-36 [Align left] operation

◆   [Align right]

The right edge of rectangle box in selection of figure is regarded as the remark for alignment.

After selecting the object, 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;

➢   Pressing the [Align right] button 　　 on the toolbar

After executing [Align right], the view is as following:
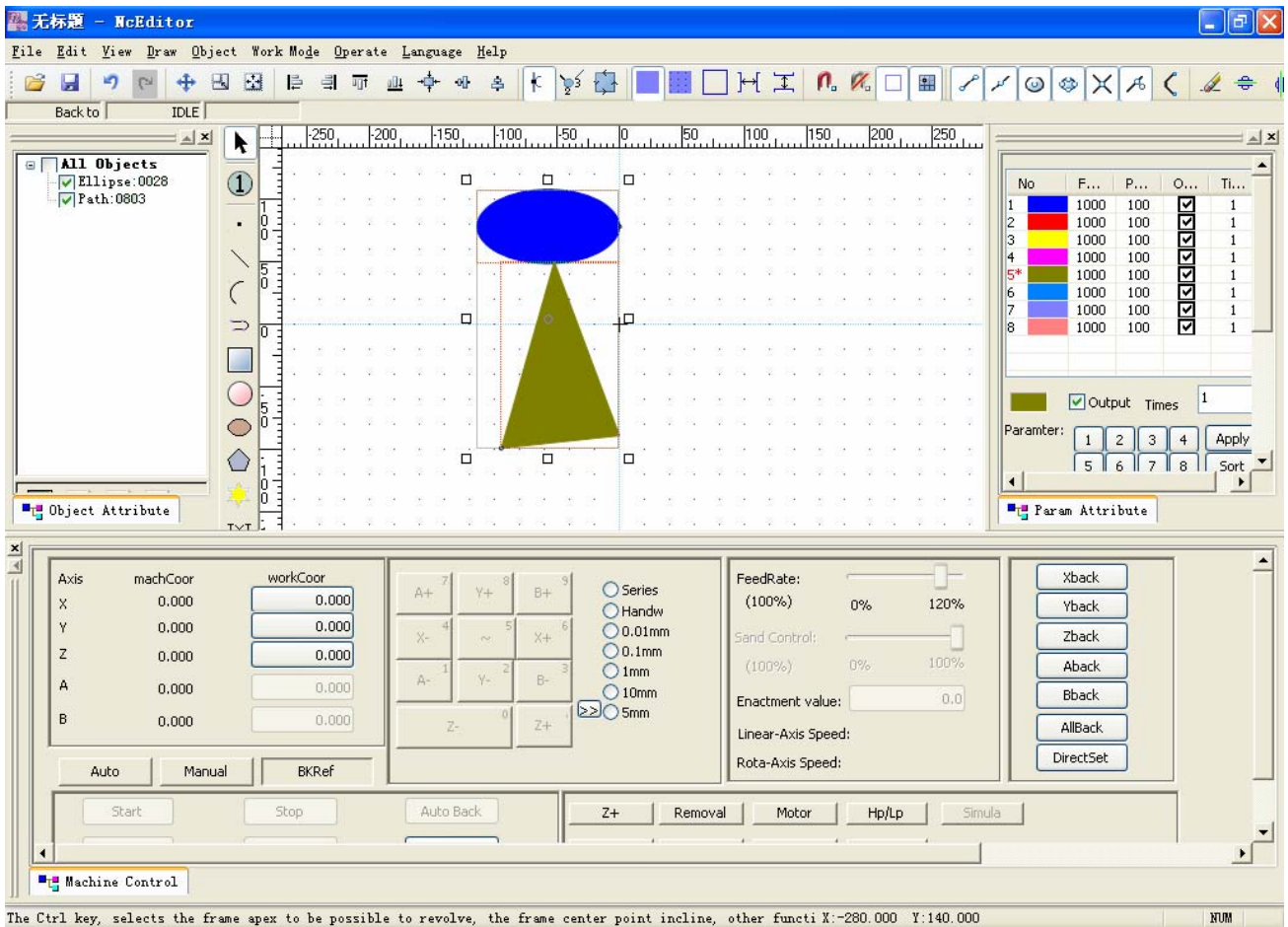
Fig. 10-37 [Align right] operation

◆ [Align vertical line]

The X-axis coordinate of center point of rectangle box is regarded as the remark of the X-axis coordinates of center points of all the selected objects.

After selecting the object, there are three methods to fulfill [Align vertical line] function:

➢ Right-click the object and select the option of [Align vertical line] under the menu;

➢ Select option [Align vertical line] under [Object] menu;

➢ Pressing the [Align vertical line] button 🔲 on the toolbar

After executing [Align vertical line], the view is as following:
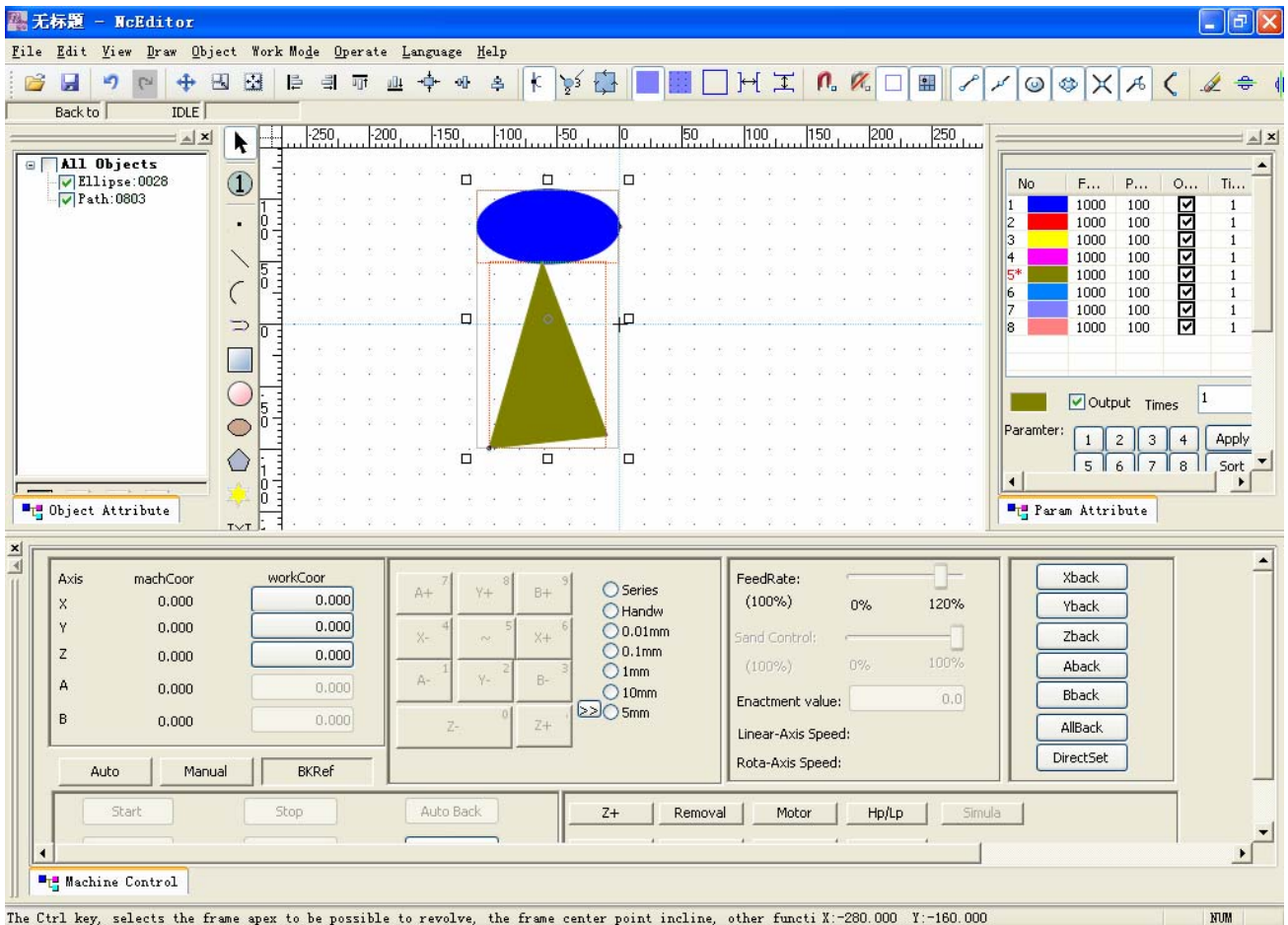
Fig. 10-38　[Align vertical line] operation

Note: the objects move in horizontal direction in [Align left], [Align right], [Align vertical line] modes.

◆　[Align Top]

The top edge of rectangle box in selection of figure is regarded as the remark for alignment.

After selecting the object, there are three methods to fulfill [Align Top] function:

➢　Right-click the object and select the option of [Align Top] under the menu;

➢　Select option [Align Top] under [Object] menu;

➢　Pressing the [Align Top] button 🔲 on the toolbar

After executing [Align Top], the view is as following, Fig.10-39.
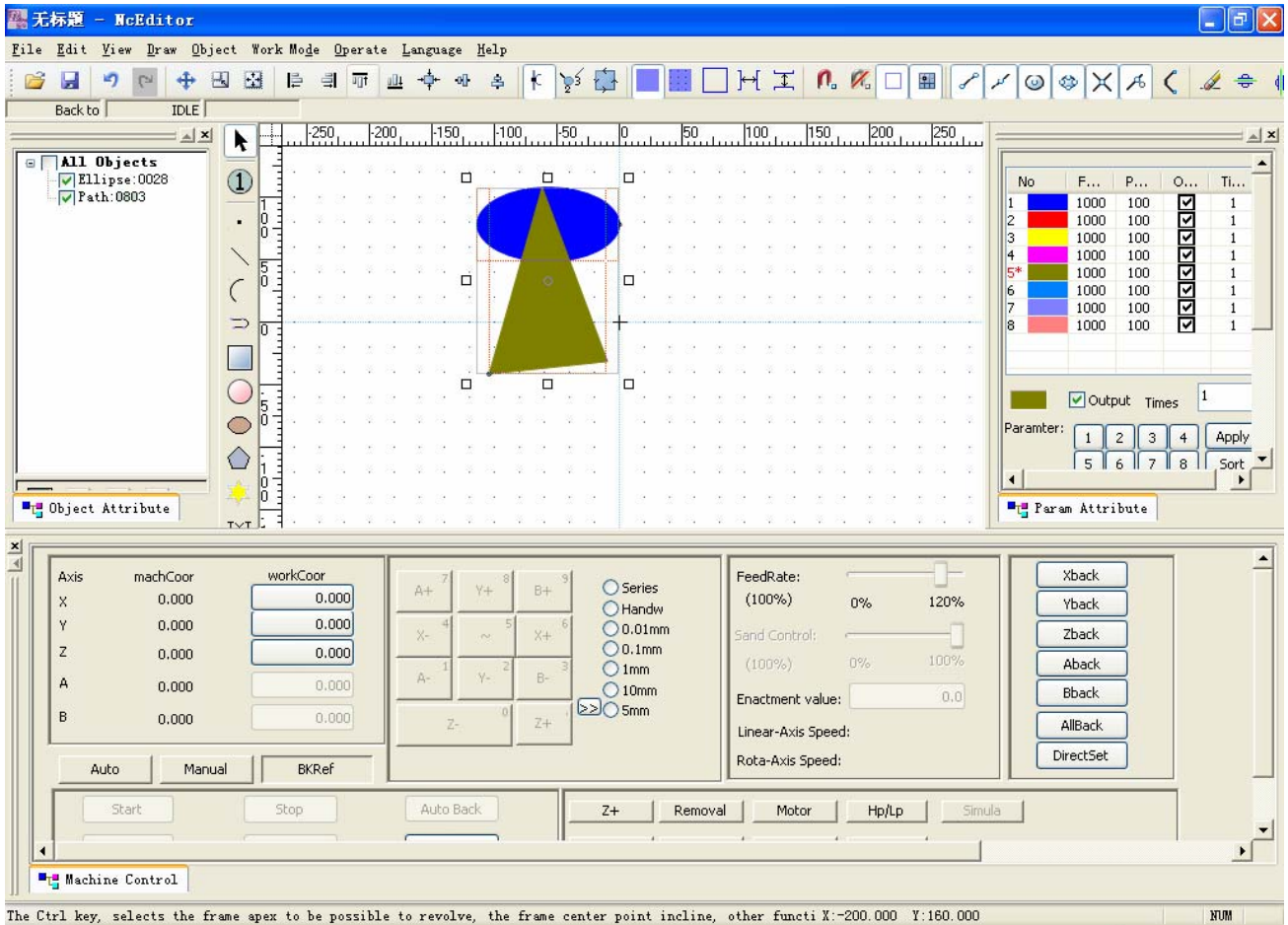
Fig. 10-39 [Align Top] operation

◆ [Align Bottom]

The bottom edge of rectangle box in selection of figure is regarded as the remark for alignment.

After selecting the object, there are three methods to fulfill [Align Bottom] function:

➢ Right-click the object and select the option of [Align Bottom] under the menu;

➢ Select option [Align Bottom] under [Object] menu;

➢ Pressing the [Align Bottom] button 🔲 on the toolbar.

After executing [Align Bottom], the view is as following, Fig.10-40.
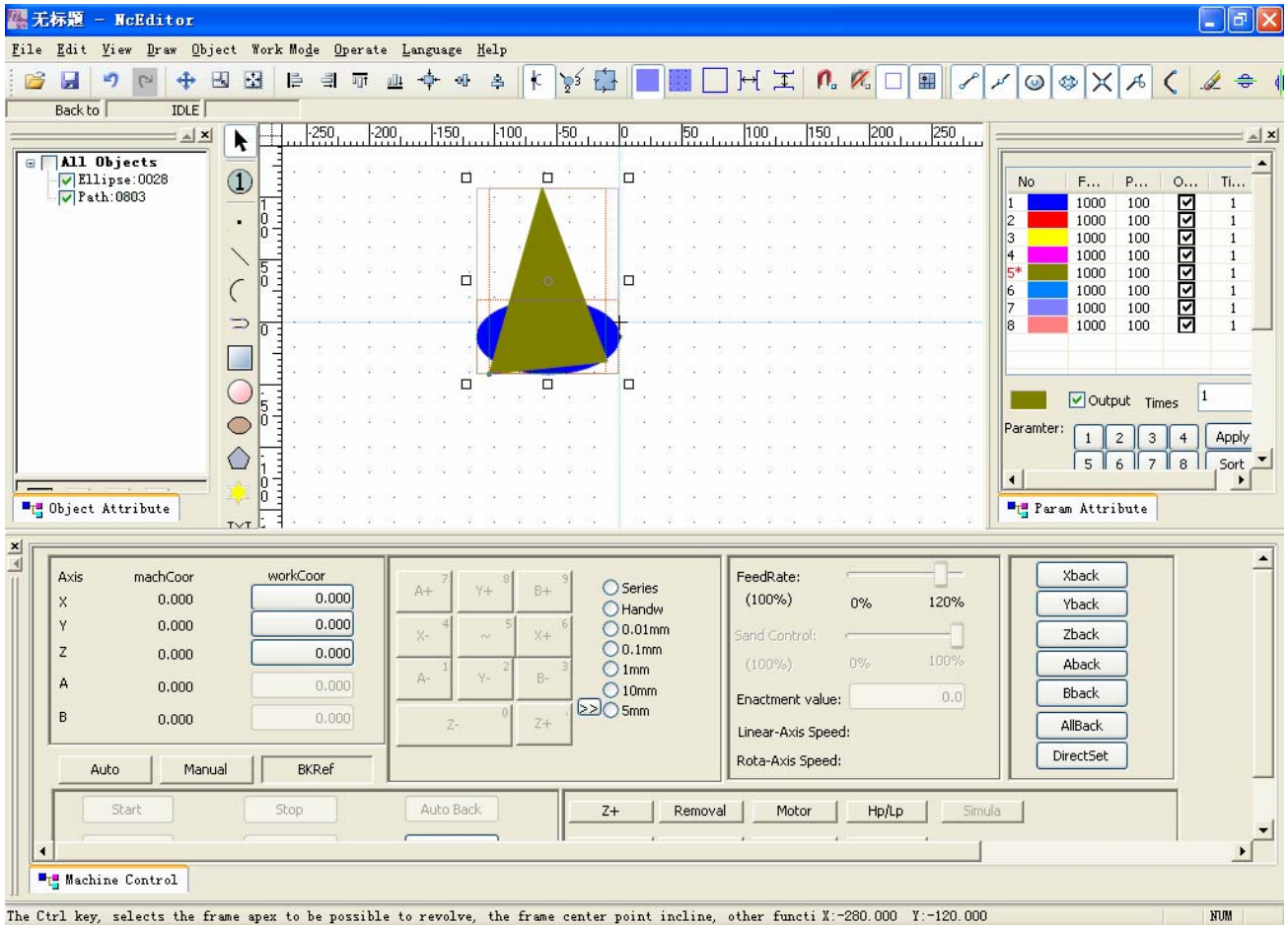
Fig. 10-40 [Align Bottom] operation

◆ [Align level line]

The Y-axis coordinate of center point of rectangle box is regarded as the remark of the Y-axis coordinates of center points of all the selected objects.

After selecting the object, there are three methods to fulfill [Align level line] function:

➢ Right-click the object and select the option of [Align level line] under the menu;

➢ Select option [Align level line] under [Object] menu;

➢ Pressing the [Align level line] button 🔲 on the toolbar

After executing [Align level line], the view is as following, Fig.10-41.
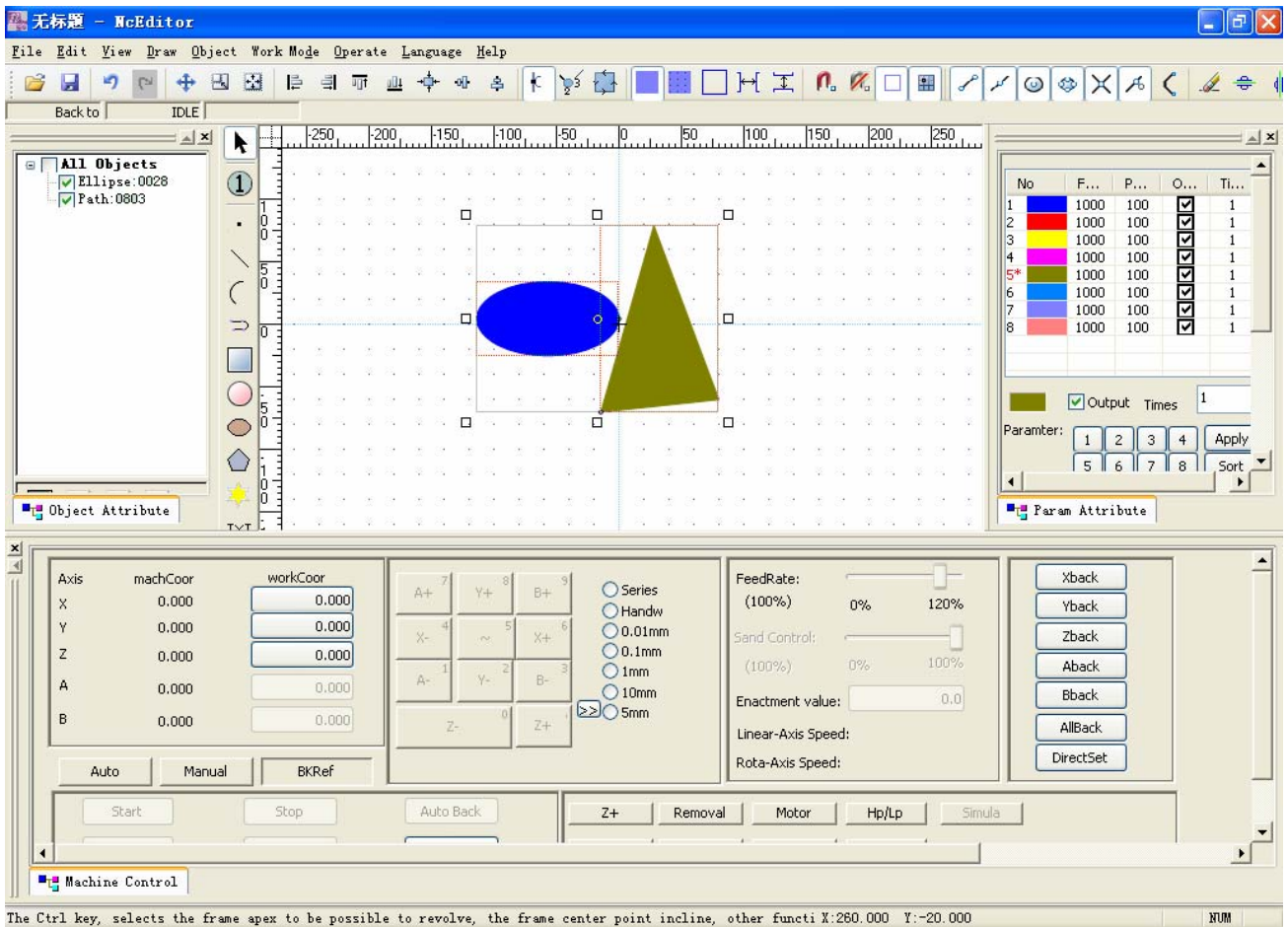
Fig. 10-41 [Align level line] operation

Note: the objects move in vertical direction in [Align Top], [Align Bottom], [Align level line].

◆ [Align center point]

The center point of rectangle box in selection of figure is regarded as the remark for alignment.

After selecting the object, there are three methods to fulfill [Align center point] function:

➢ Right-click the object and select the option of [Align center point] under the menu;

➢ Select option [Align center point] under [Object] menu;

➢ Pressing the [Align center point] button on the toolbar

After executing [Align center point], the view is as following, Fig.10-42.
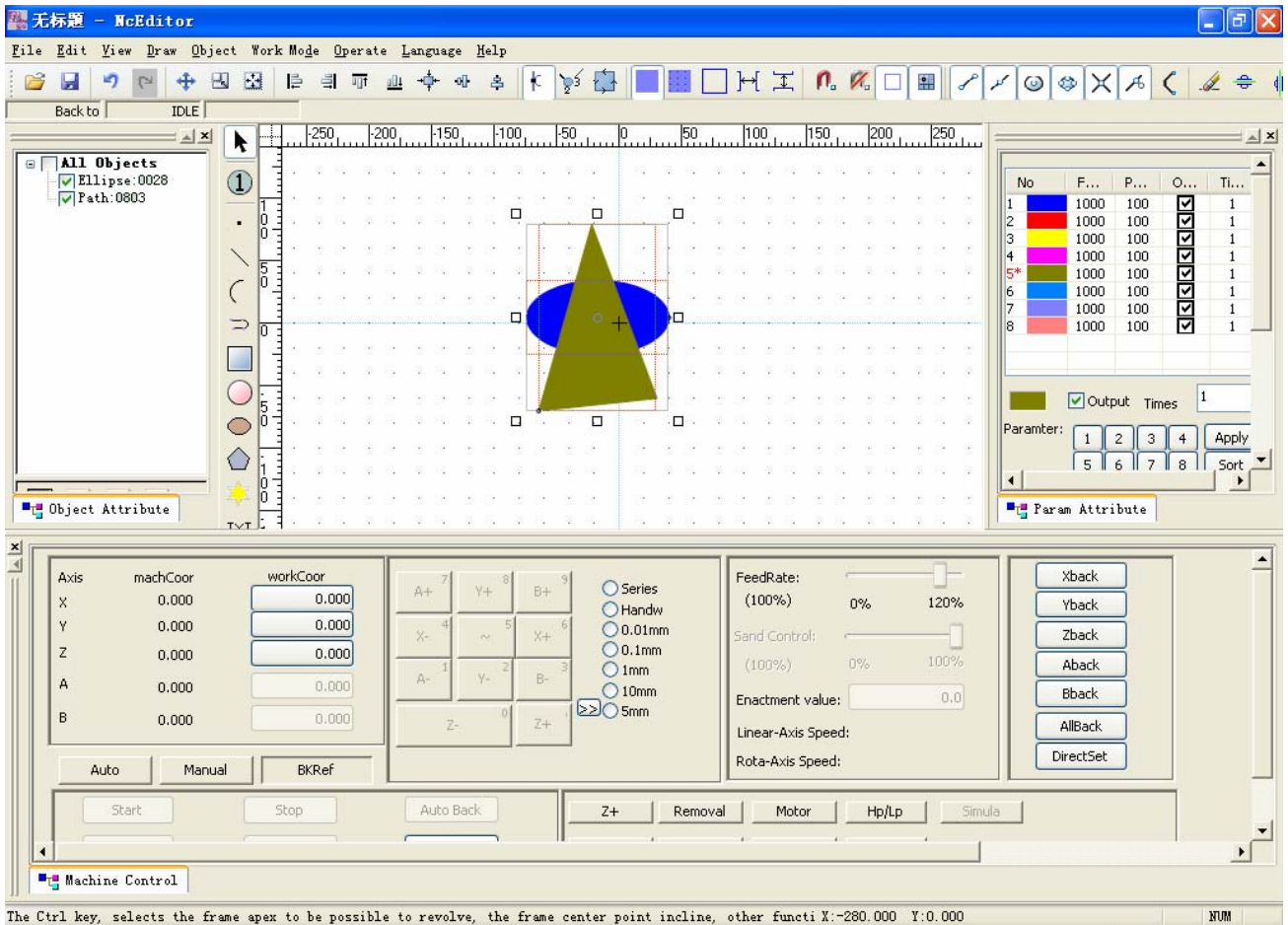
Fig. 10-42 [Align center point] operation

Note: the objects move in both vertical direction and horizontal direction by [Align center point].

◆ [Align level dispersion]

There are two align 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 object 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;

➢ Pressing the icon]↔[on toolbar;

➢ Right-click the object, and select the menu option [Align level dispersion]

After pressing the button [Align level dispersion], the following dialogue will pop up, as Fig.10-43.
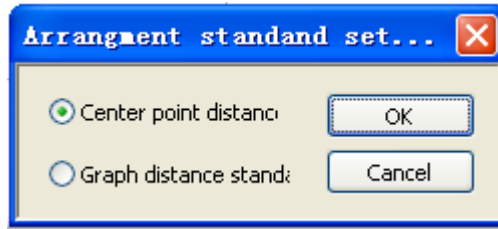
Fig. 10-43 [Arrangement standard setting]

Select [center point distance] after [Align level dispersion], see Fig.10-44, Fig.10-45.
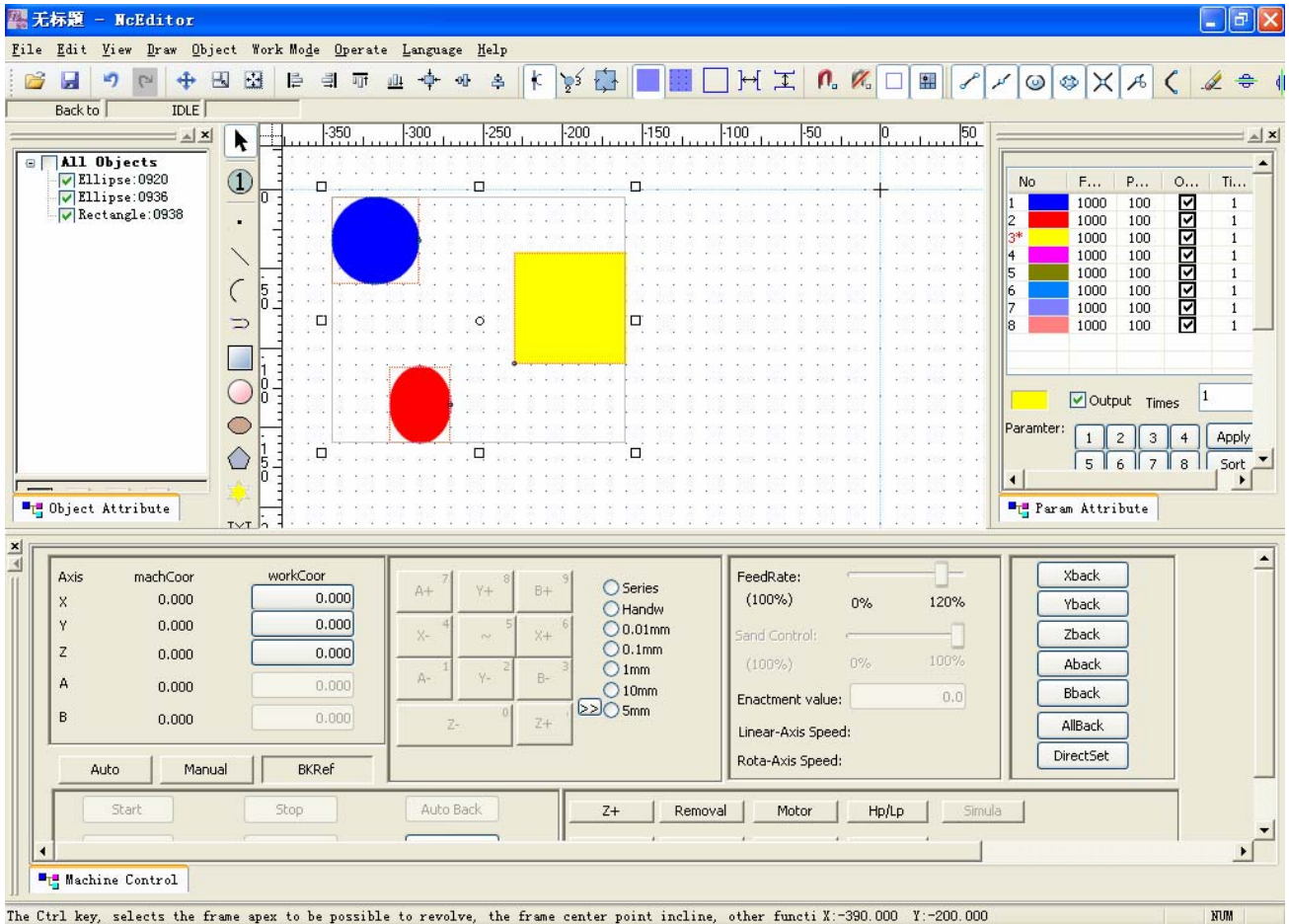
Fig. 10-44 before [Align level dispersion]
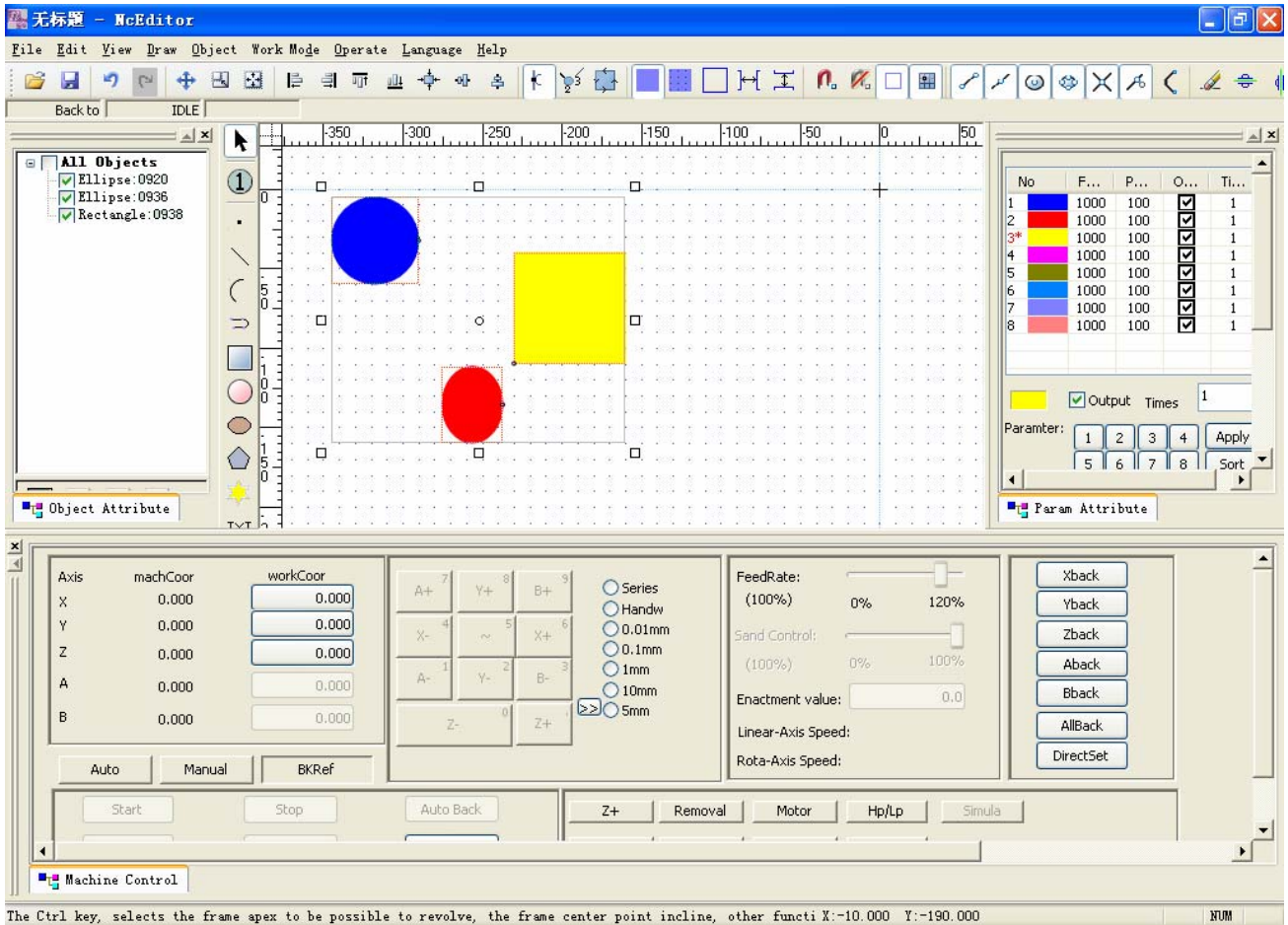
Fig. 10-45 after [Align level dispersion]

Note:

The selected objects must be three or above three, or, a dialogue will pop up, "objects selected are less than 3, operation cancels."

◆ [Align vertical dispersion]

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 object 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;

➢ Pressing the icon ⊥ on toolbar;

➢ Right-click the object, and select the menu option [Align vertical dispersion]

After pressing the button [Align vertical dispersion], a dialogue will pop up, which is the same as [Align level dispersion].

Select [center point distance] after [Align vertical dispersion], see Fig.10-46, Fig.10-47.
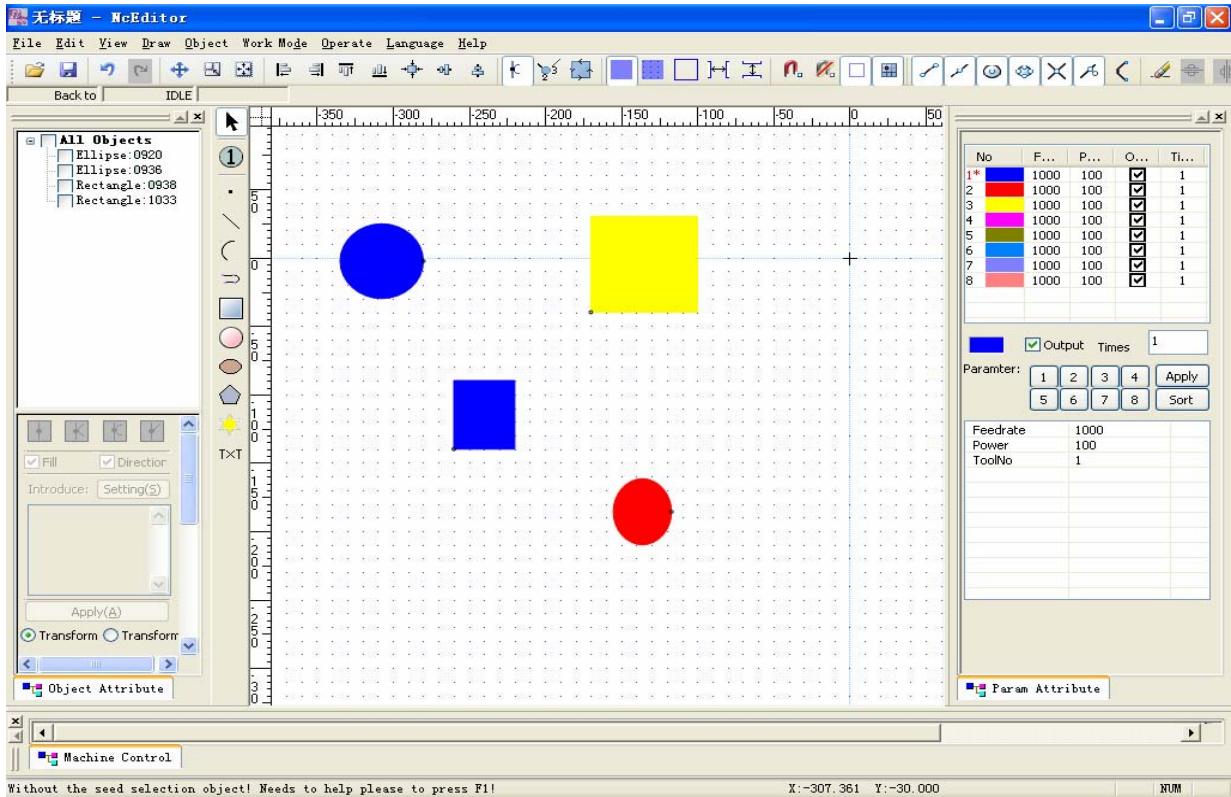
Fig. 10-46 before [Align vertical dispersion]
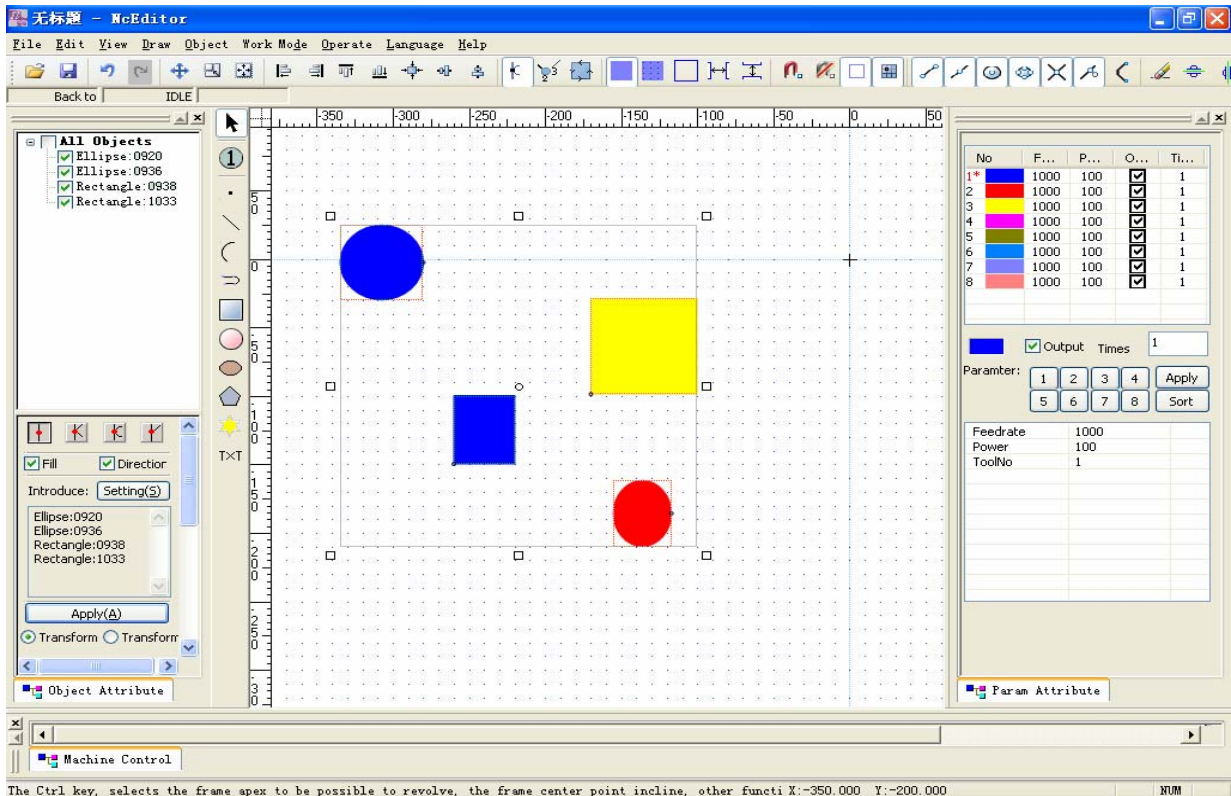
Fig. 10-47 after [Align vertical dispersion]

◆ [Auto setting machining order]

In order to save time and improve efficiency in machining, NcEditor holds the function of [Auto setting machining order], which denotes system will automatically set the machining order according to best path algorithm.

Manipulation method:

➢ Draw several different types of figures.

➢ Select [View order] option under [View] or pressing the icon ⬚ on toolbar to display the machining order of the figures, see Fig.10-48.
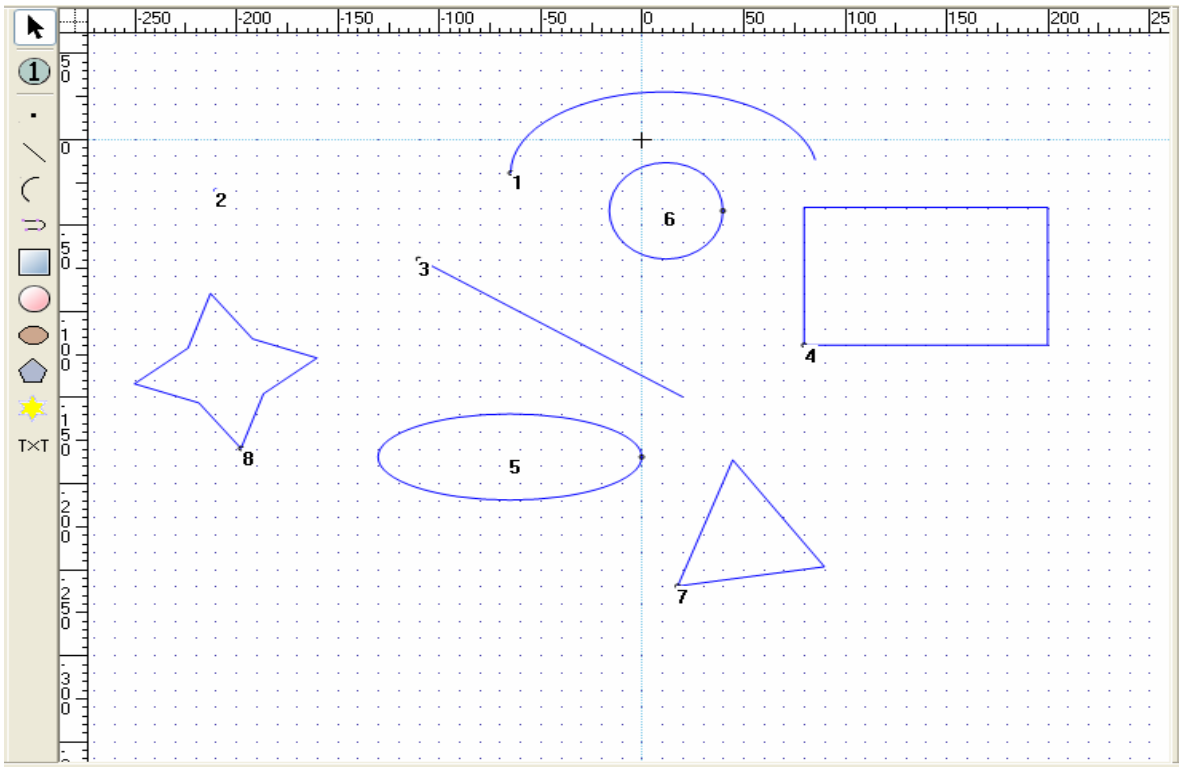
Fig. 10-48 displaying the machining order of the objects

➢ Click the option [Auto setting machining order] under [Object] or right-click the object to select [Auto setting machining order] option, and system will automatically set the machining order adhering to best path algorithm, see Fig.10-49.
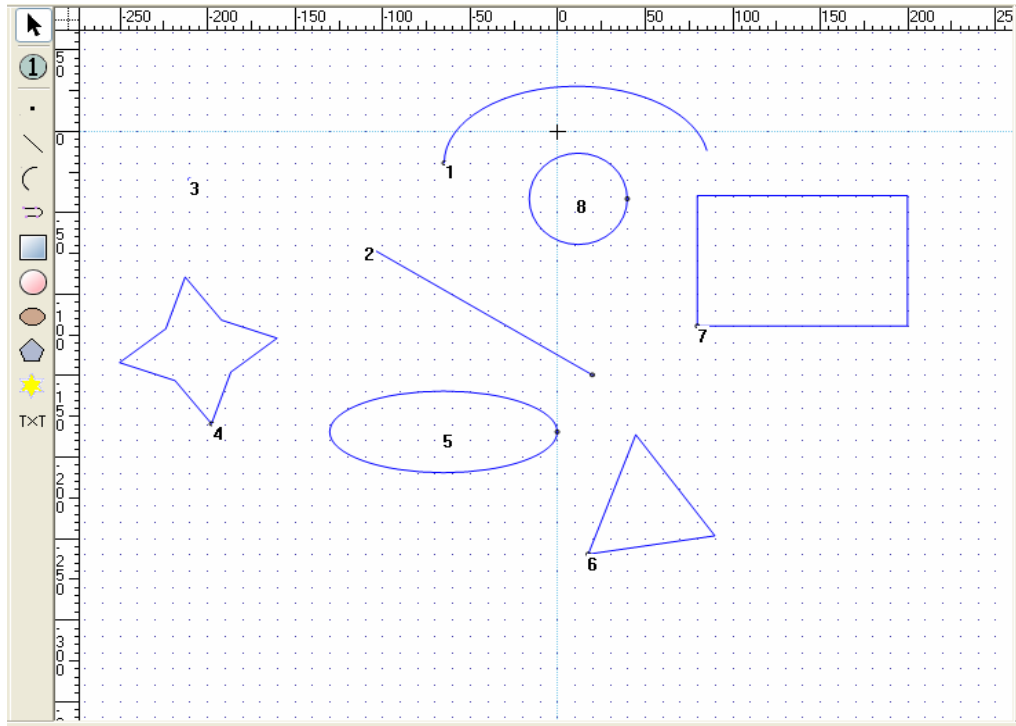
Fig. 10-49 auto setting machining order

Users can modify the machining orders of objects according to their 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 sequence from small number to large number.

Save the sequence of several ahead objects in [Auto setting machining order], and change the sequence of following machining objects. Pressing the button "Ctrl" and then click the frame of object No.4 to keep the machining order of No.1 -4 unchanging; If users want to change the machining order of No.5-8, they can click the object 5, then click the rest objects sequentially.

◆ [Manual setting machining order]

This function is to make it more convenient for users to set the special machining order.

After executing [Auto setting machining order], if users want to change the machining order of certain object, the method is as following:

Select the objects → choose [Manual setting machining order] under [object] menu or under right-click menu → input the number in the input frame → press [ok]
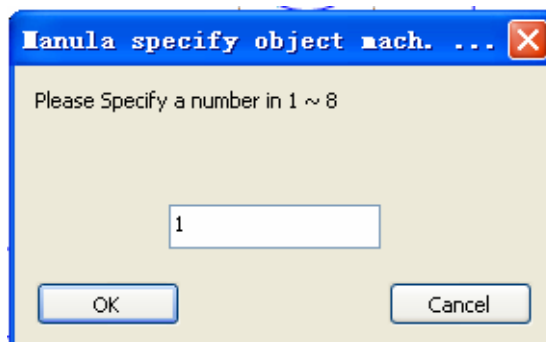
Fig. 10-50 [manual setting machining order] Dialogue

Please specify a number between 1 and 10, "10" is the total number of objects in [Object editor space], which is variable.

[Setting lead-in line] has been introduced in [Object attribute], which will be omitted here.

◆　[Group]

If selecting multiple objects and multiple groups or objects with groups simultaneously, user should set them as a group.

See Fig.10-51, please select the required objects or object groups in [Object list window], and then choose the option [Group] under [Object] menu or under right-click menu.
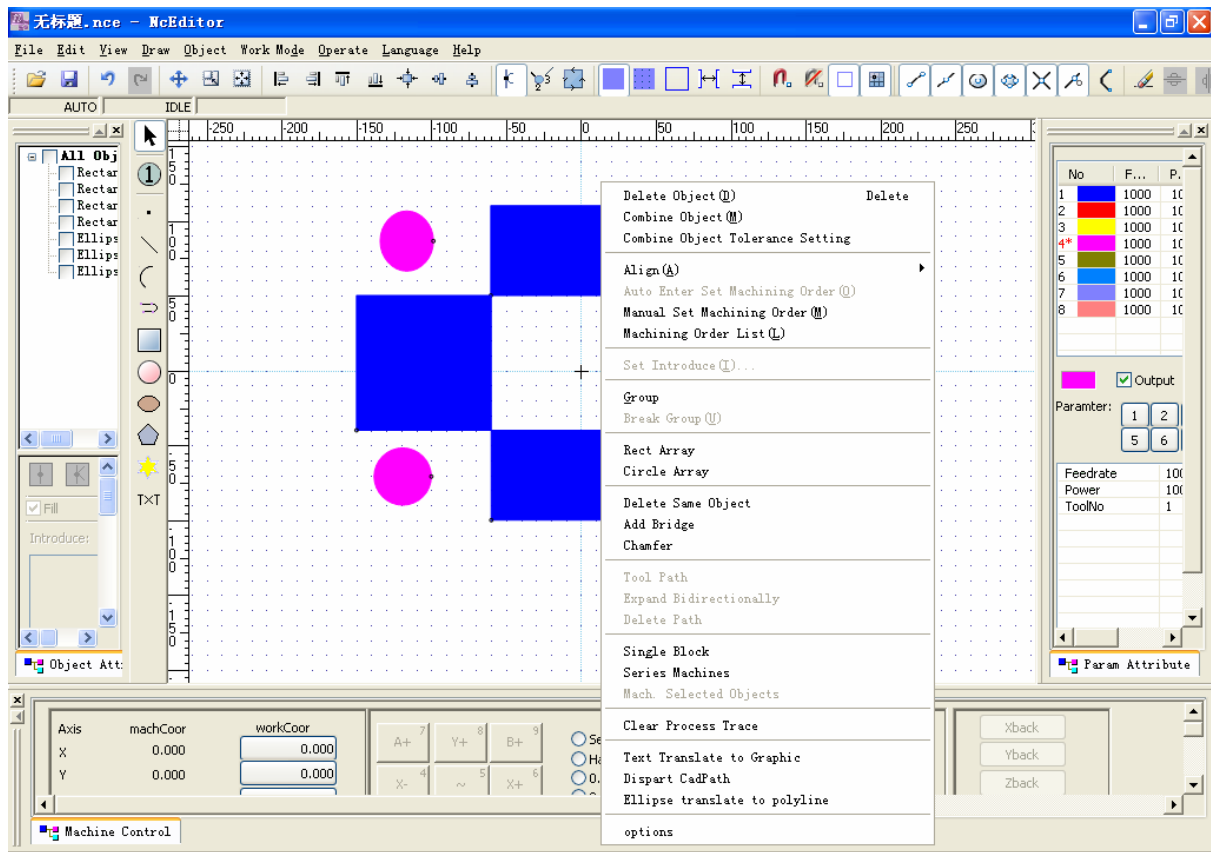
Fig. 10-51 setting group objects

See Fig.10-52, the rectangles "0264", "0266", "0273", "0274" are set as a new group "0284".
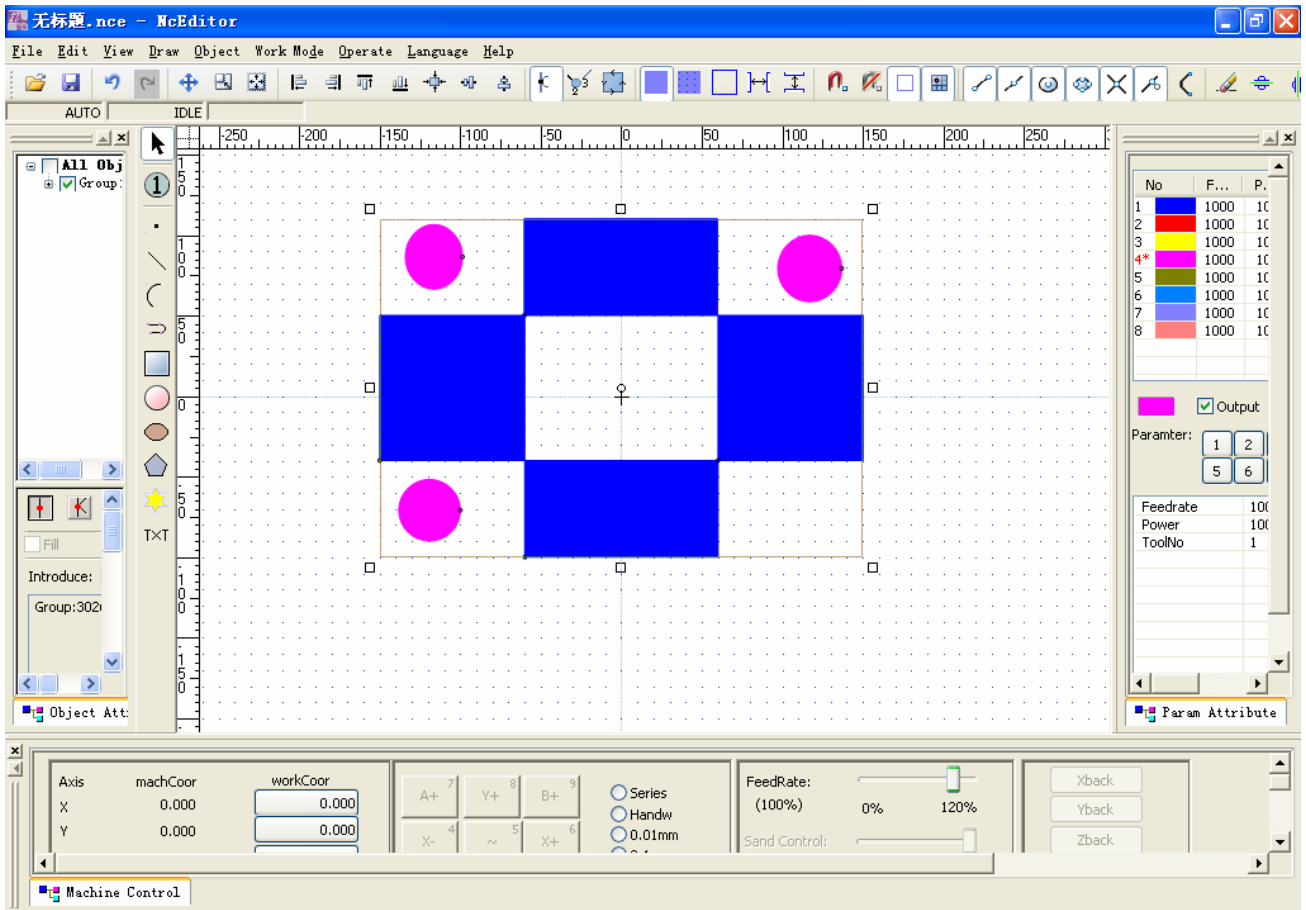
Fig. 10-52 setting rectangle as a group

At this time, the new group can be edited entirely.

◆ [Break Group]

If users only want to edit parts of the objects, the group should be broken first.

Please select the group in the [Object list window] (click the group name to highlight it) →choose the option [Break group] under [Object] or under right-click menu→ the objects in group will back to the upper group.

◆ [Circle array]

Select one or multiple objects or object groups in [Object list] (objects in different group 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 and units on circle into the input frame

For instance, select the group→ choose [Circle array] → input the data as following and press "OK"
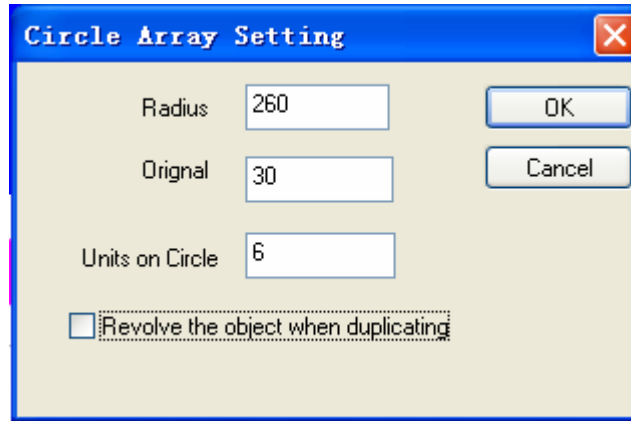
Fig. 10-53   [setting circle array] dialogue

The output after setting circle array is as following, Fig.10-54.
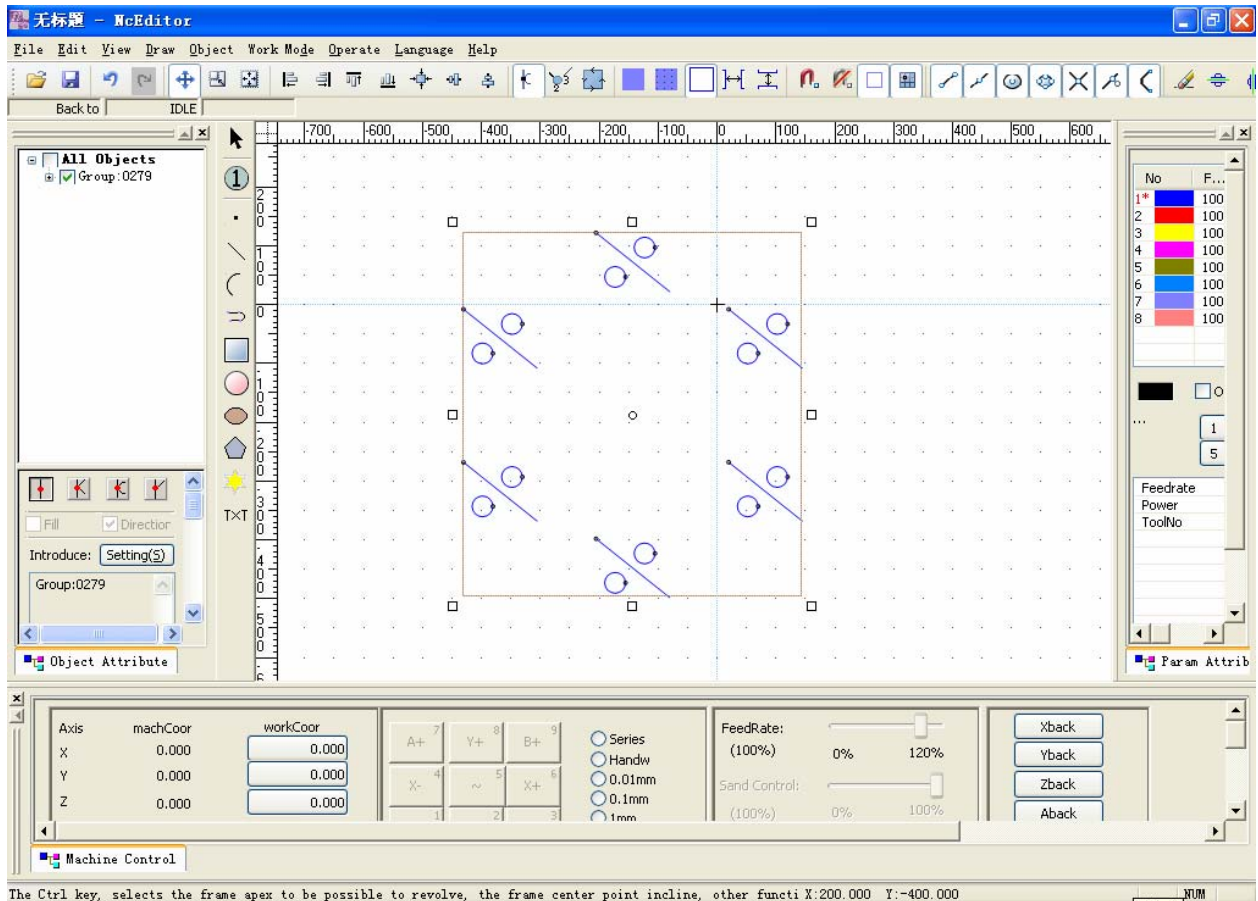
Fig. 10-54 output after setting circle array

Note: objects should be set as a group before executing array function, when multiple machining objects are selected.

◆   [Delete duplicate object]

This function is used to delete the object of same size, position and shape, which are exact overlapping objects. The manipulation is as following:

➢   Select the object

➢ Right-click the mouse → choose [Delete duplicate object] under menu or choose [Delete duplicate object] under [Object] menu

NcEditor will save only one machining object and delete rest duplicate objects.

◆ [Add Bridge]

The function of bridge is to make management of processed sections more convenient, after processing, the section will remain in the origin position.

[Add bridge] is divided into [auto add bridge] and [manual add bridge], [Add bridge] is available for single object, 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]

Draw a selected rectangle as following:
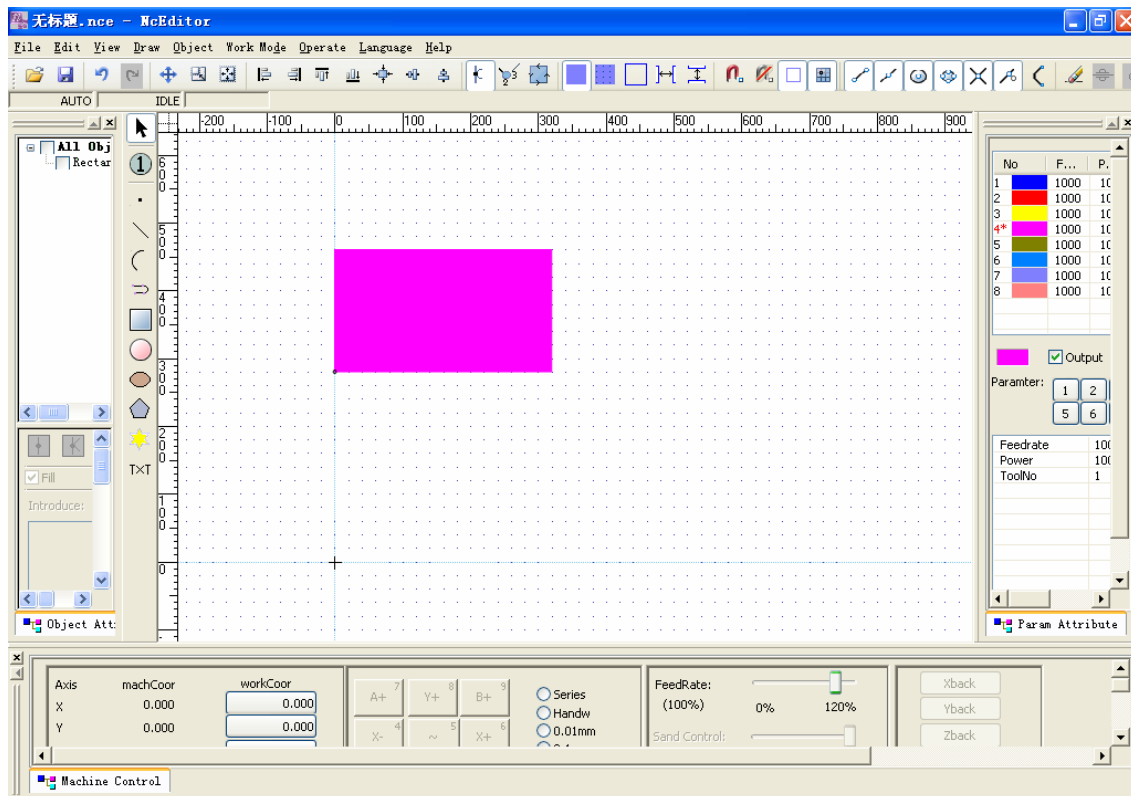
Fig. 10-55 draw a selected rectangle

Right-click this object →choose [Add bridge] under right-click menu or under [Object] menu → input data as following:
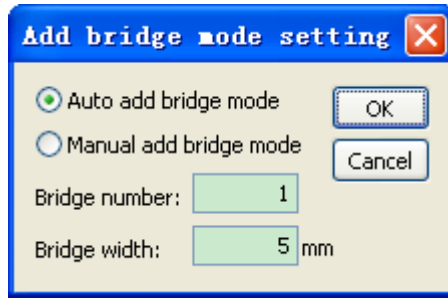
Fig. 10-56 [Auto add bridge mode] dialogue

Click "ok", the output of rectangle is as following, Fig.10-57.
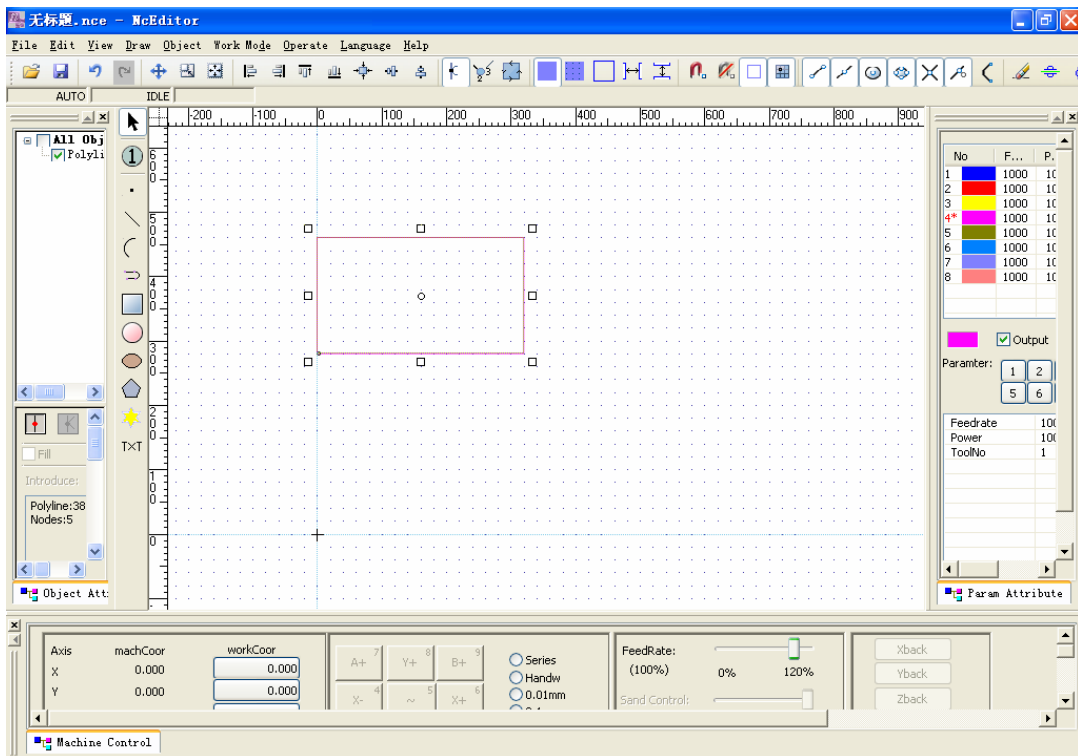
Fig. 10-57 after auto adding bridge

The bridge is located in the gap on the lower left corner, and during machining, this position will be skipped to keep rectangle on the workpiece.

◆ [Manual add bridge]

Draw an unselected circle and a rectangle as following:
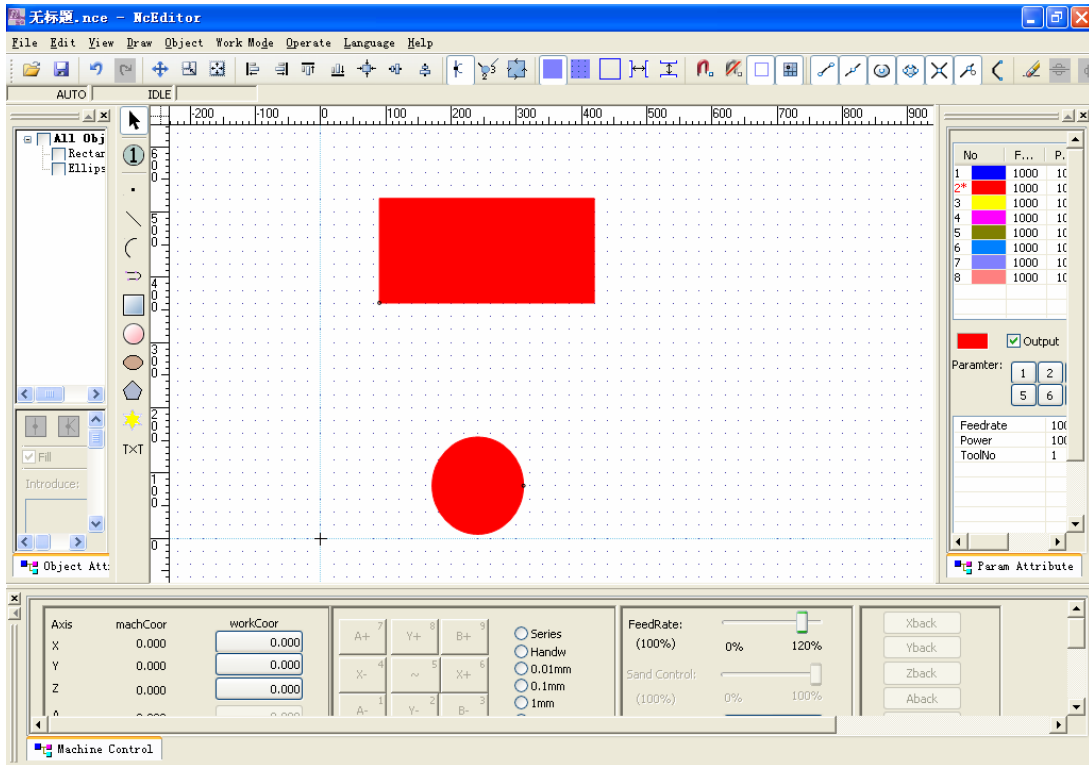
Fig. 10-58 example of manual add bridge mode

Select [Add bridge] under right-click menu or under [Object] menu → choose manual add bridge mode → input the following data and press "ok"
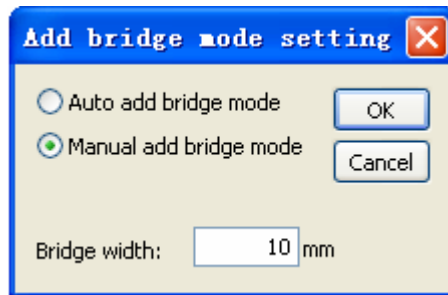
Fig. 10-59 manual add bridge mode dialogue

Single click the frames of the circle and rectangle to add bridge, as following, Fig.10-60.
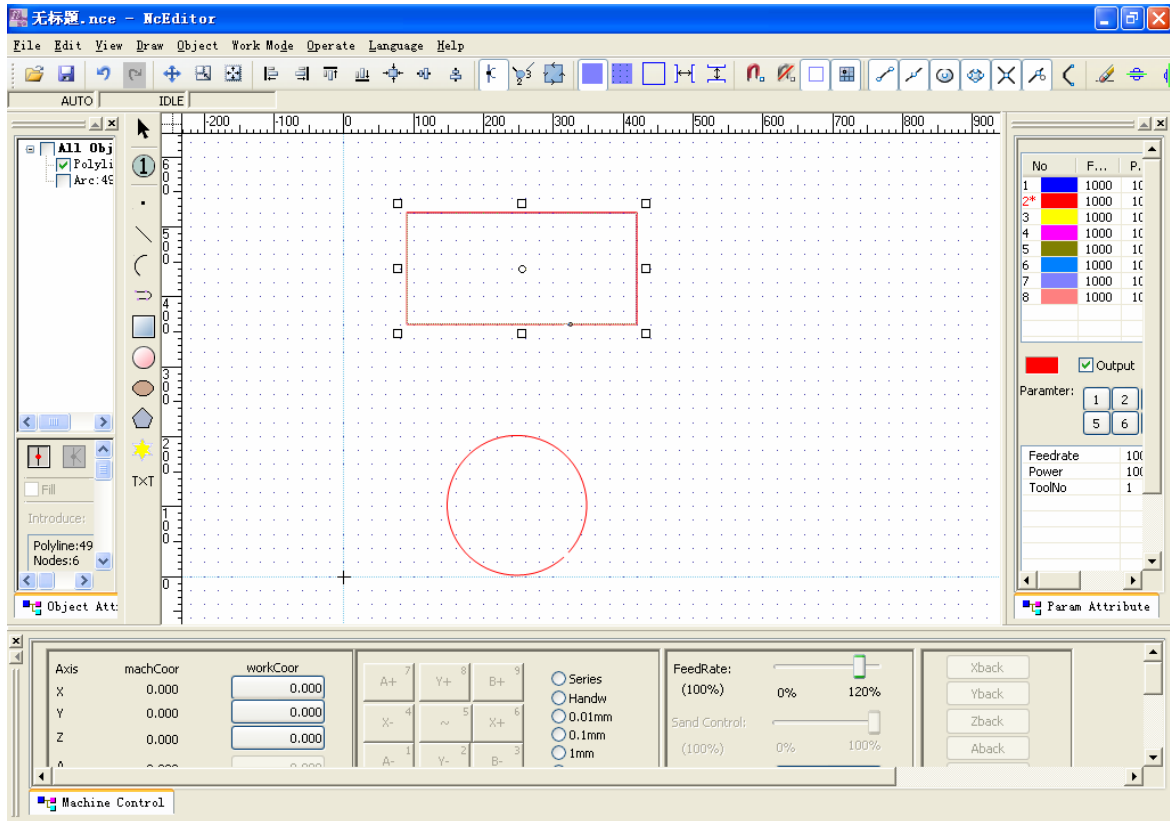
Fig. 10-60 after manual add bridge

◆ [Chamfer]

The software supports fillet and chamfer. As following, Fig.10-61, draw a selected rectangle.
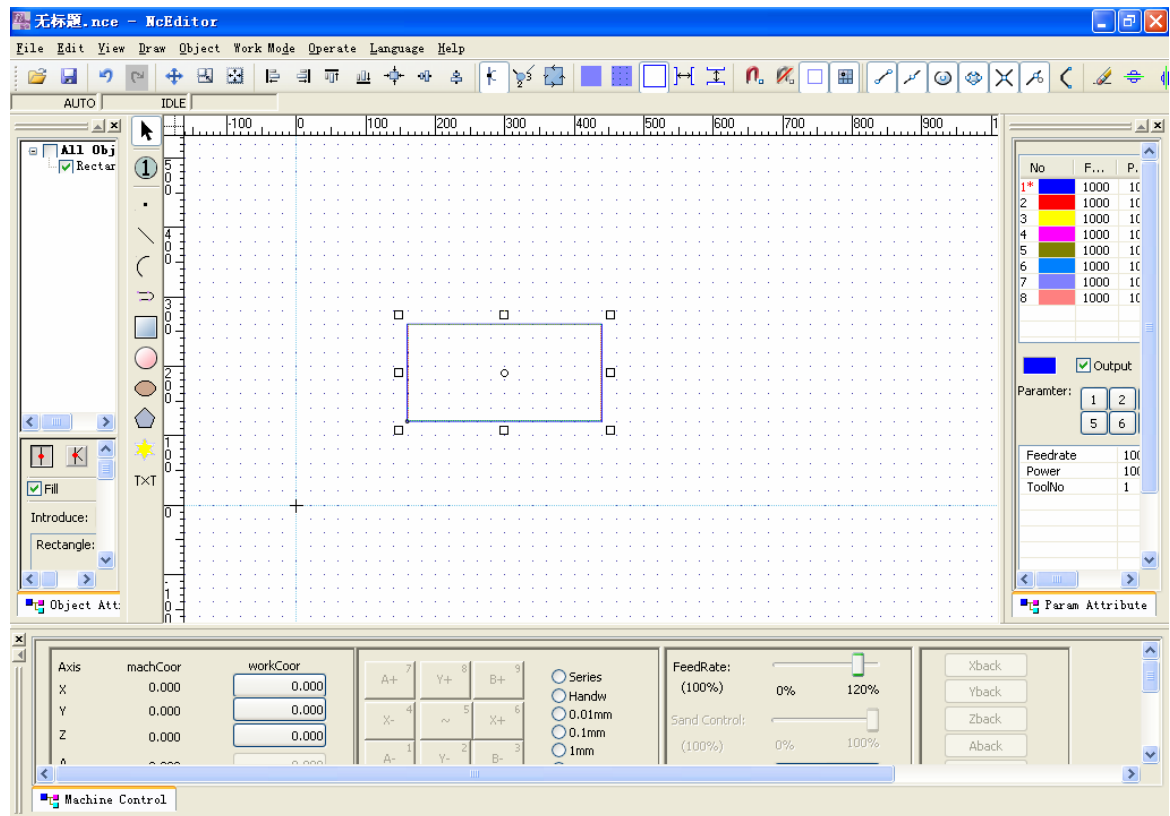
Fig. 10-61 draw a selected rectangle

Choose [Chamfer] under [Object] menu, the following dialogue will pop up.
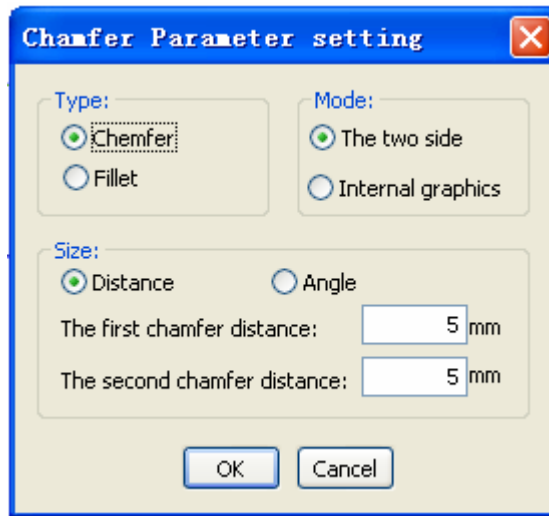
Fig. 10-62 setting chamfer parameters

There are two modes for fillet and chamfer, [Two sides] and [Internal graphics]

◆ [Two sides] mode

[Two sides] mode is ensuring that the chamfer is the angle contained by two sides. See Fig.10-62, under the chamfer mode, there are two modes for setting size, [Distance] and [Angle]. Under [Distance] mode, users can input the distance of two sides, as Fig.10-62, for instance, input 5mm into the input frame → press "ok", cursor reshaping. The tip will prompt in the bottom of Nceditor. See Fig.10-63.

Please select the first side.The two side must be two lines not parallel with each other or two lines in a M X:100.000 Y:-200.000

Fig. 10-63 first edge selecting tip

Selected the first side of the object, the tip will prompt as Fig.10-63. Then select the second side adjacent to first side. Then the rectangle in Fig.10-61 will change into the figures as Fig.10-65.

Please select the second side.The two side must be two lines not parallel with each other or two lines in a X:100.000 Y:150.000

Fig. 10-64 second edge selecting tip
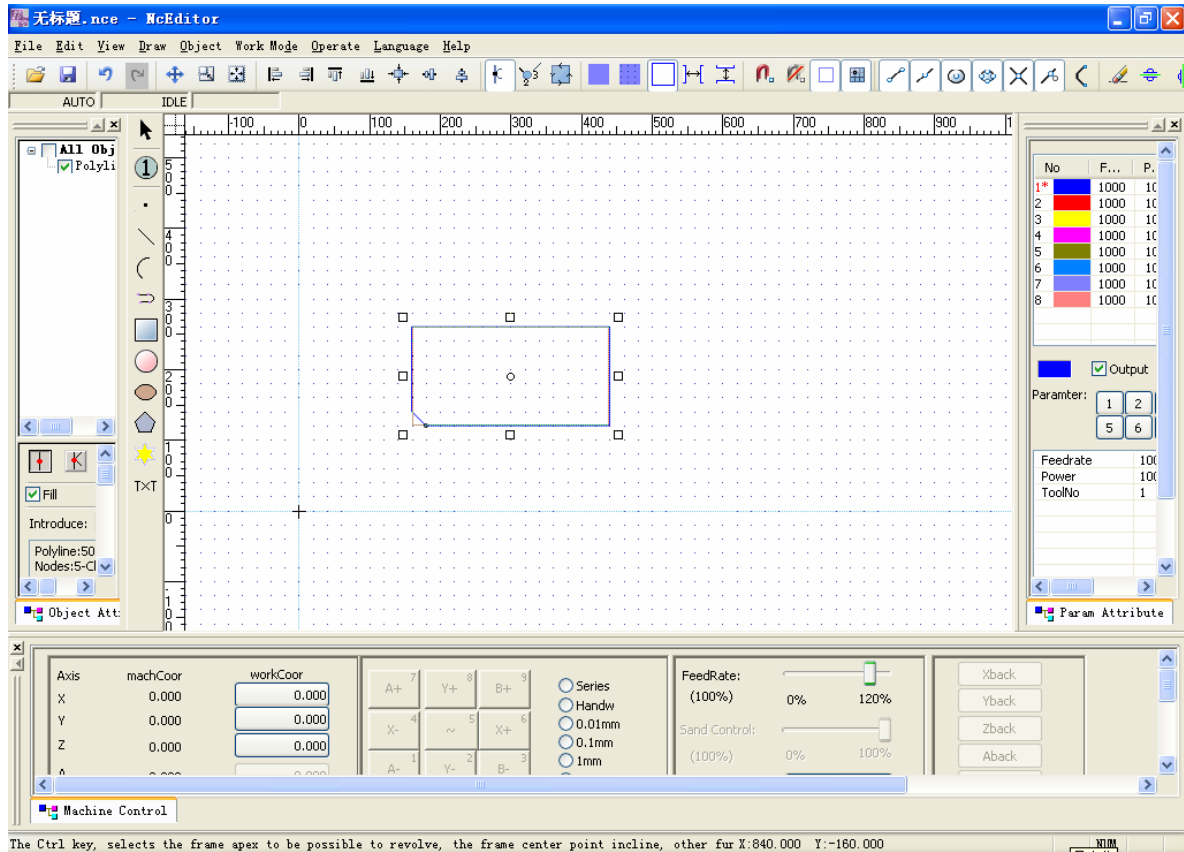
Fig. 10-65 output after executing [chamfer]

Input following data under [Angle] mode of [chamfer], in Fig.10-66.
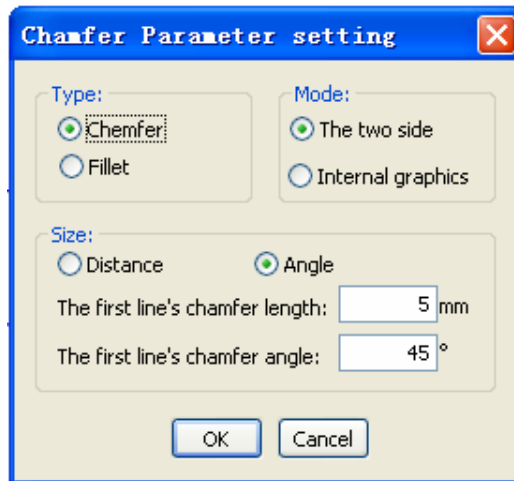
Fig. 10-66 [Angle] mode under [Chamfer] dialogue

After setting the [Chamfer], please select the two adjacent sides, the manipulation is the same as that under [Distance] mode. The output is the same as in Fig.10-65.

Setting parameters under [Two sides] of [Fillet] is used to set the two adjacent sides round angle. The only way for setting size is [Radius], and input the data as Fig.10-67. Press "ok" and select two adjacent sides, the output is as following, Fig.10-68.
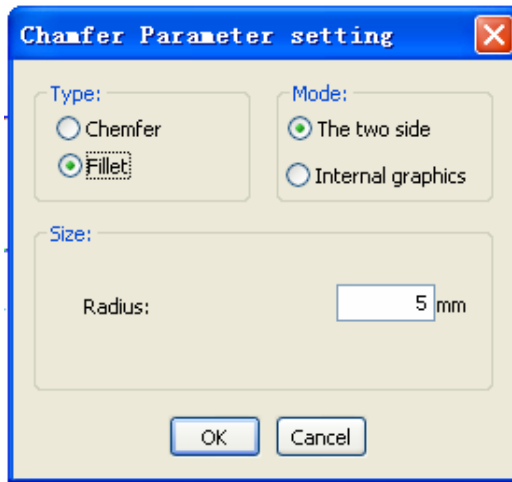
Fig. 10-67 setting chamfer parameters
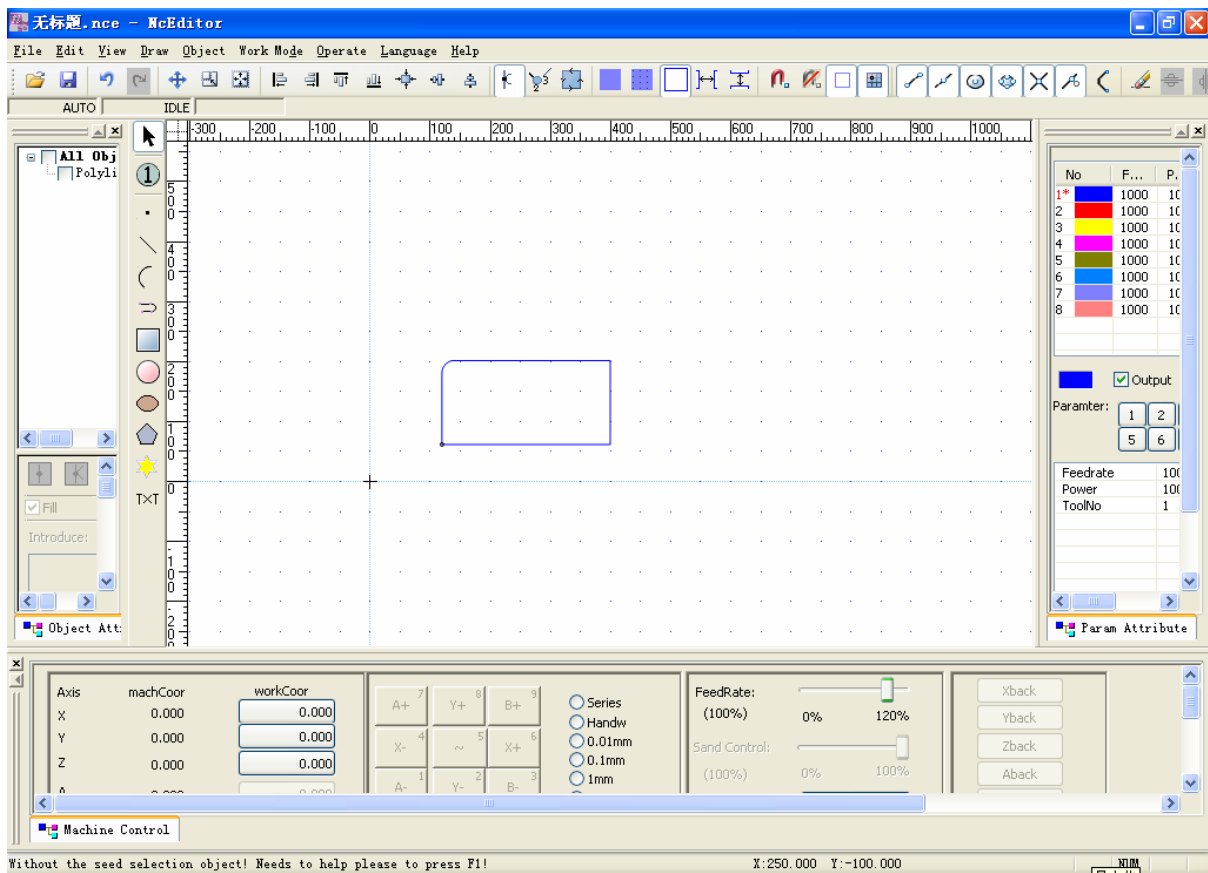
Fig. 10-68 output after executing [Fillet]

◆ [Internal graphics]

[Internal graphics] is ensuring the all the qualified chamfers inner the graphics. Select [Internal graphics] mode, input data as Fig.10-69. After pressing "ok", selecting any edge of the figure, system will automatically process all the qualified included angles, and the output is as Fig.10-70. Under [Internal graphics], there are two methods, [Distance] and [Angle], which are identical to the above and will be omitted.
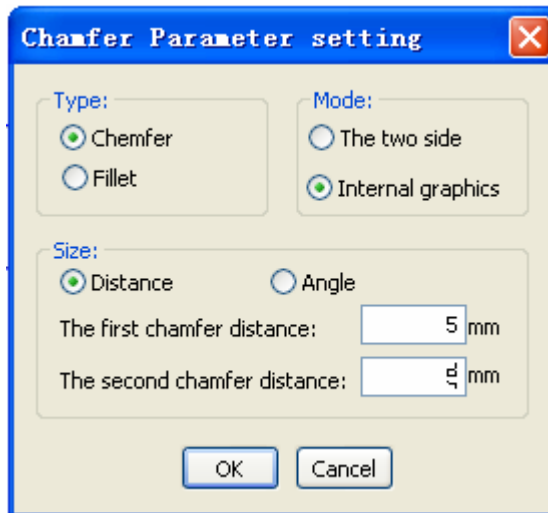
Fig. 10-69 [Internal graphics] mode of [Chamfer]
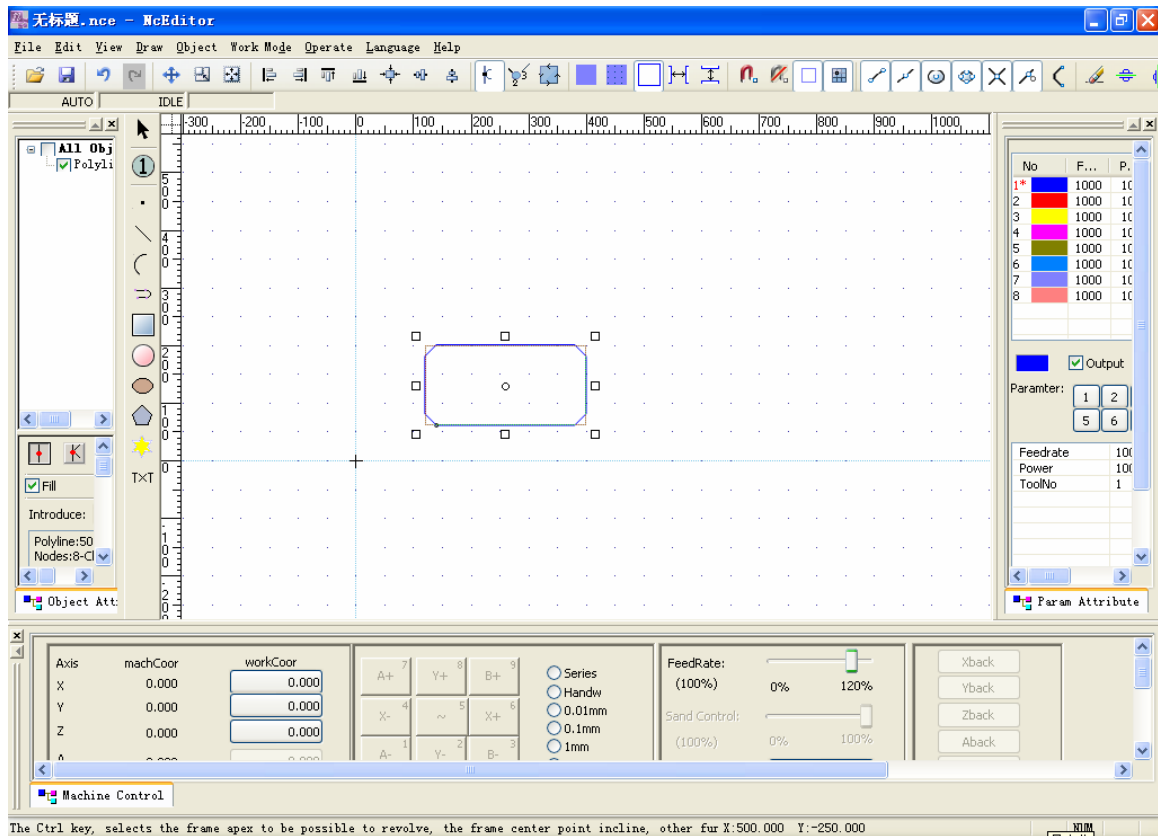
Fig. 10-70 output after [Internal graphics]

Under [Internal graphics] mode of [Fillet], system will automatically process all the qualified included angles, inputting radius of 5 mm into the input frame and the output is as Fig.10-71.
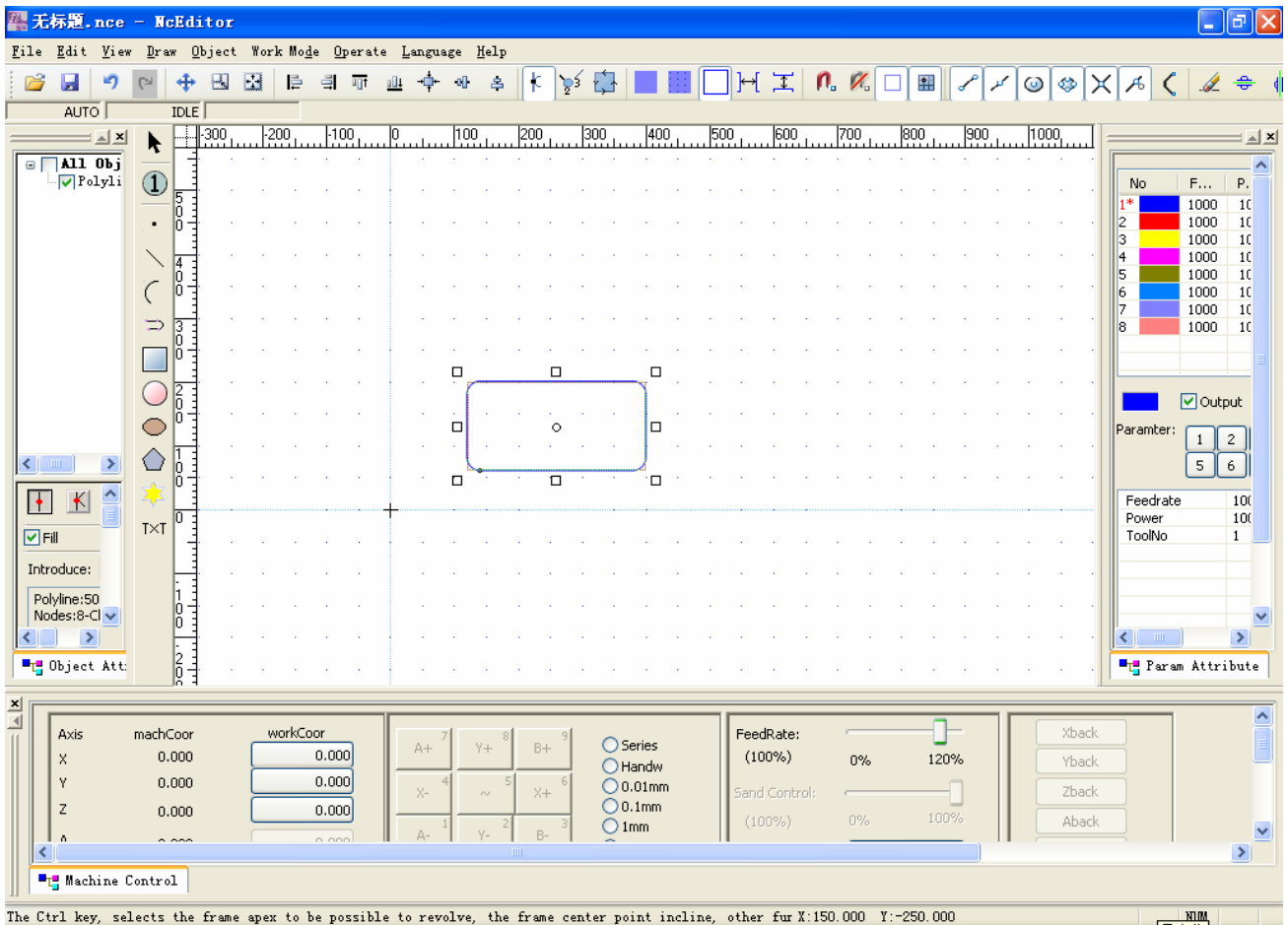
Fig. 10-71 output after [Internal graphics] mode of [Fillet]

◆ [Tool path]

After selecting object, users 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 selecting this option, a dialogue will pop up as Fig.10-72. The default tool compensation is off; When the compensation type is "computer", user can input the value of tool diameter, compensation direction and chord.
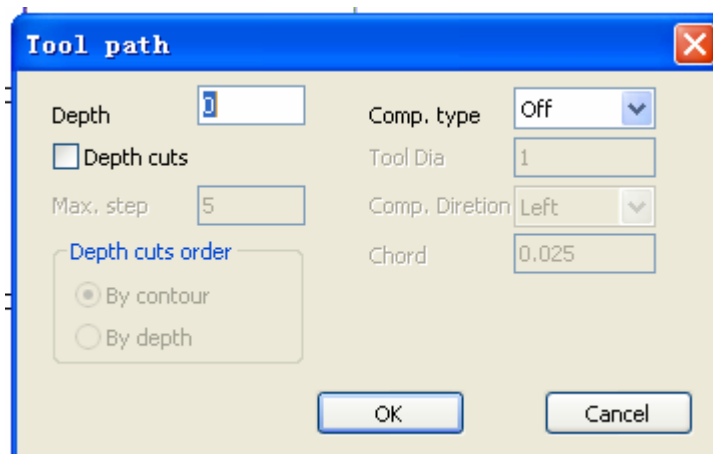
Fig. 10-72 [Tool path] dialogue

When the value of left compensation is "1", the tool path is as following, Fig.10-73.
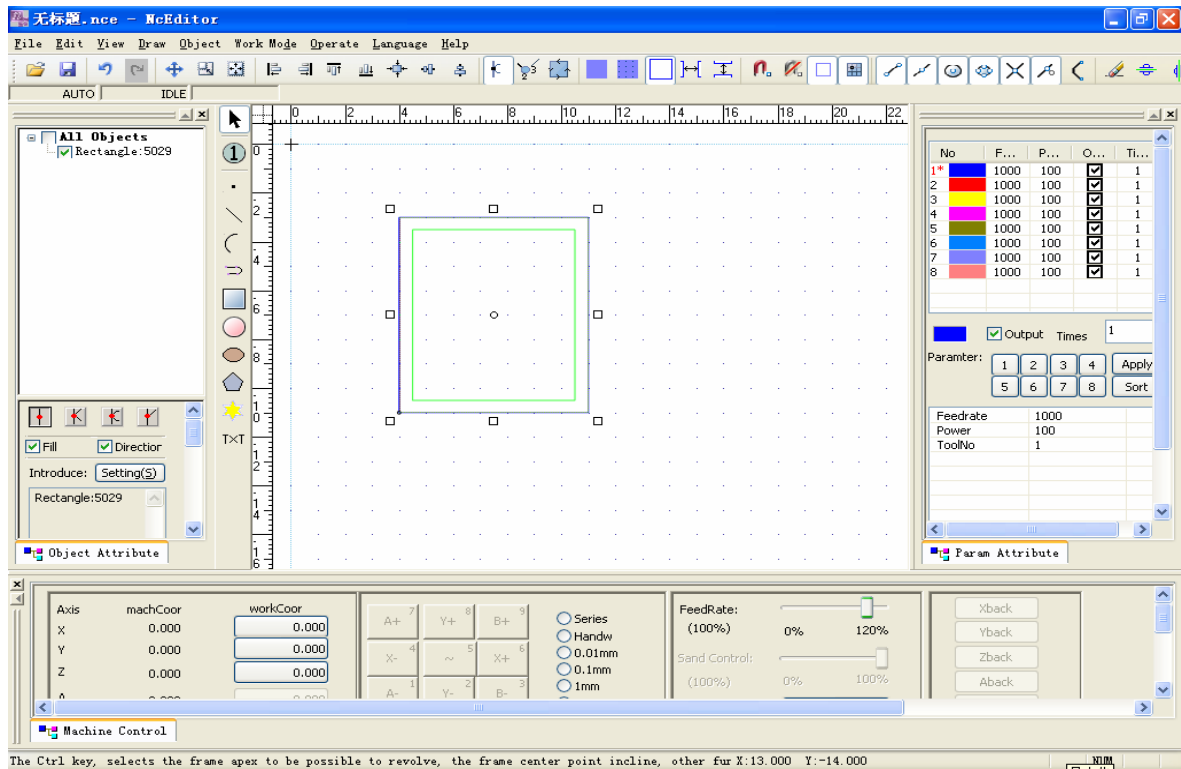
Fig. 10-73 [Tool path] of left-compensation as "1"

When the value of right compensation is "1", the tool path is as following, Fig.10-74.
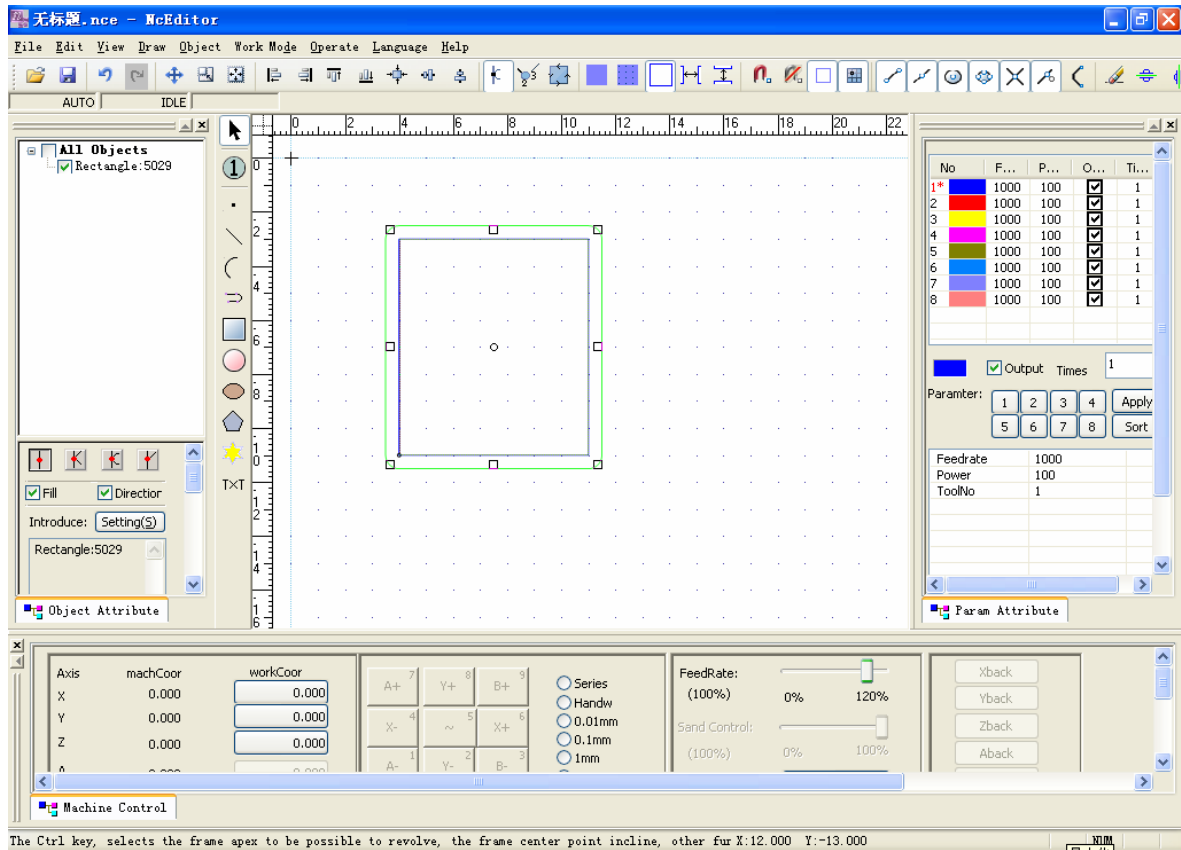
Fig. 10-74 [Tool path] of right-compensation as "1"

◆ [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 object, users can fulfill this function by following two methods.

➢ Choose the option of [Expand Bidirectionally] under [Object].

➢ Choose [Expand Bidirectionally] under right-click menu.

After selecting this option, the following dialogue will pop up.
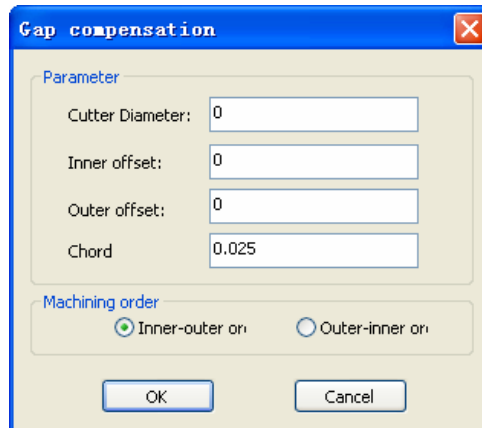
Fig. 10-75 [Gap compensation] Dialogue

The parameters in this function are the same meaning as in [Tool path]. [Inner offset] and [Outer offset] are equal to left compensation and right compensation. After setting the parameters, press "ok". The view under [Expand Bidirectionally] is as Fig.10-76, when values of [Inner offset] and [Outer offset] are both "2".
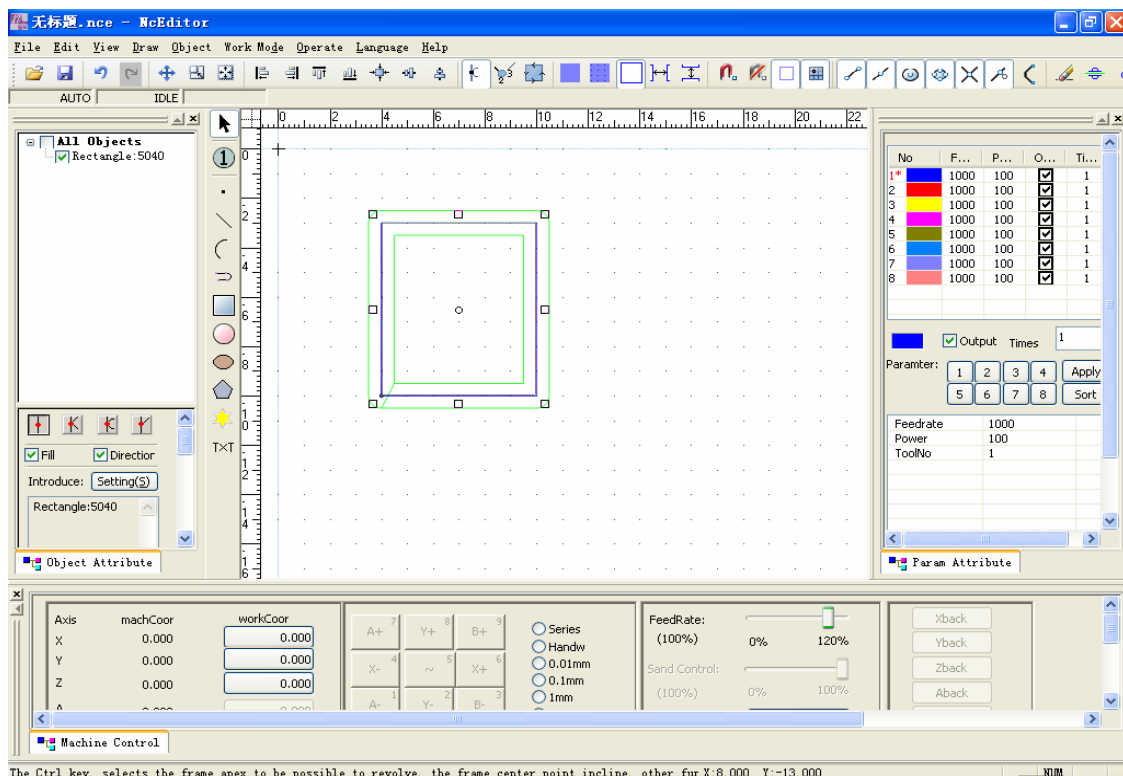

Fig. 10-76 after executing [Expand Bidirectionally]

◆ [Delete Tool path]

After executing [Tool path] or [Expand Bidirectionally], [Delete Tool path] is available. Concrete manipulation is as following:

After selecting object, users can fulfill this function by following two methods.

➢ Choose the option of [Delete Tool path] under [Object].

➢ Choose [Delete Tool path] under right-click menu.

◆ [Series machining]

When there are several graphics for machining, users want the machine goes on machining next figures without lifting tool after finishing machining first graphic, which requires all the graphics connected. After selecting [Series machining], system will connect the graphics automatically and go on machining continuously.

After selecting two or above two graphics, users can choose [Series machining] under [Object] menu or under right-click menu.

All the graphics are connected after pressing [Series machining], as Fig.10-77.
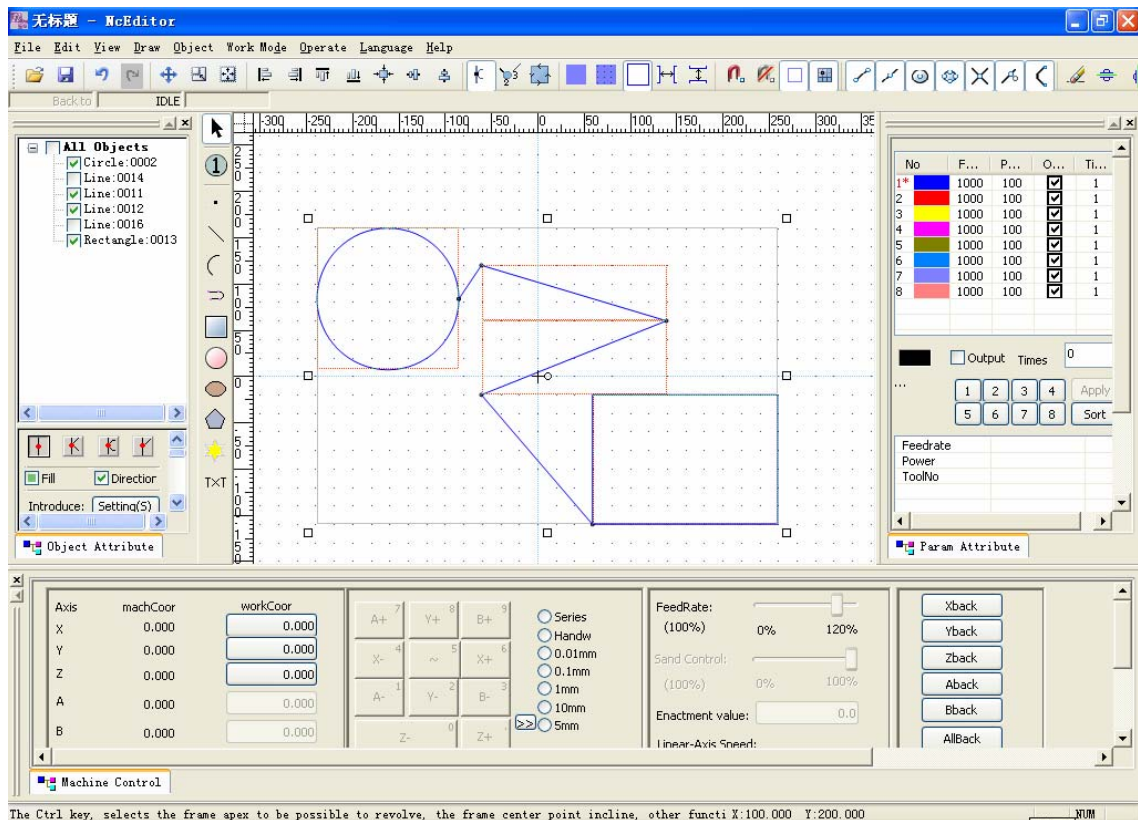

Fig. 10-77 example of [Series machining]

◆ [Clear ProcessTracks]

When machining object, the red tracks on graphics mean processed. After finishing machining, all the graphics turn to red. Actually, the color is the processing track, rather than the color of graphic. So, we can select [Clear Process Track] to clear track.

After finishing machining, there are several methods to fulfill [Clear Process Track] as following:

> ➢ Choose the option of [Clear Process Track] under [Object].

> ➢ Choose [Clear Process Track] under right-click menu.

> ➢ Pressing the icon ![icon] on toolbar;

Note: [Clear Process Track] is unavailable during machining.

◆ [Text translate to graphic]

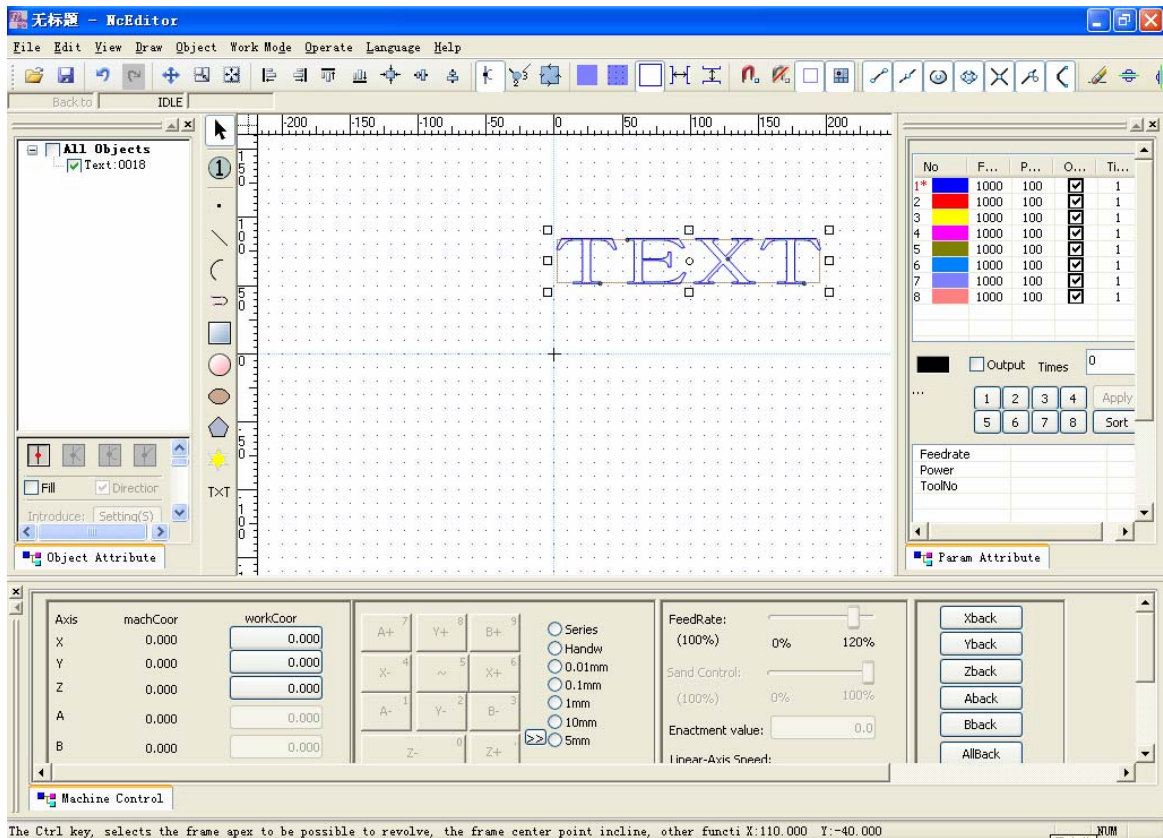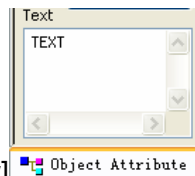Press button TxT and drag this button to the editor space, as following, Fig.10-78.

Fig. 10-78 step one of [Text translate to graphic]

Input the text in the [Object attribute window] Object Attribute → press [Apply] → the input text will display in the editor space, as following, Fig.10-79.
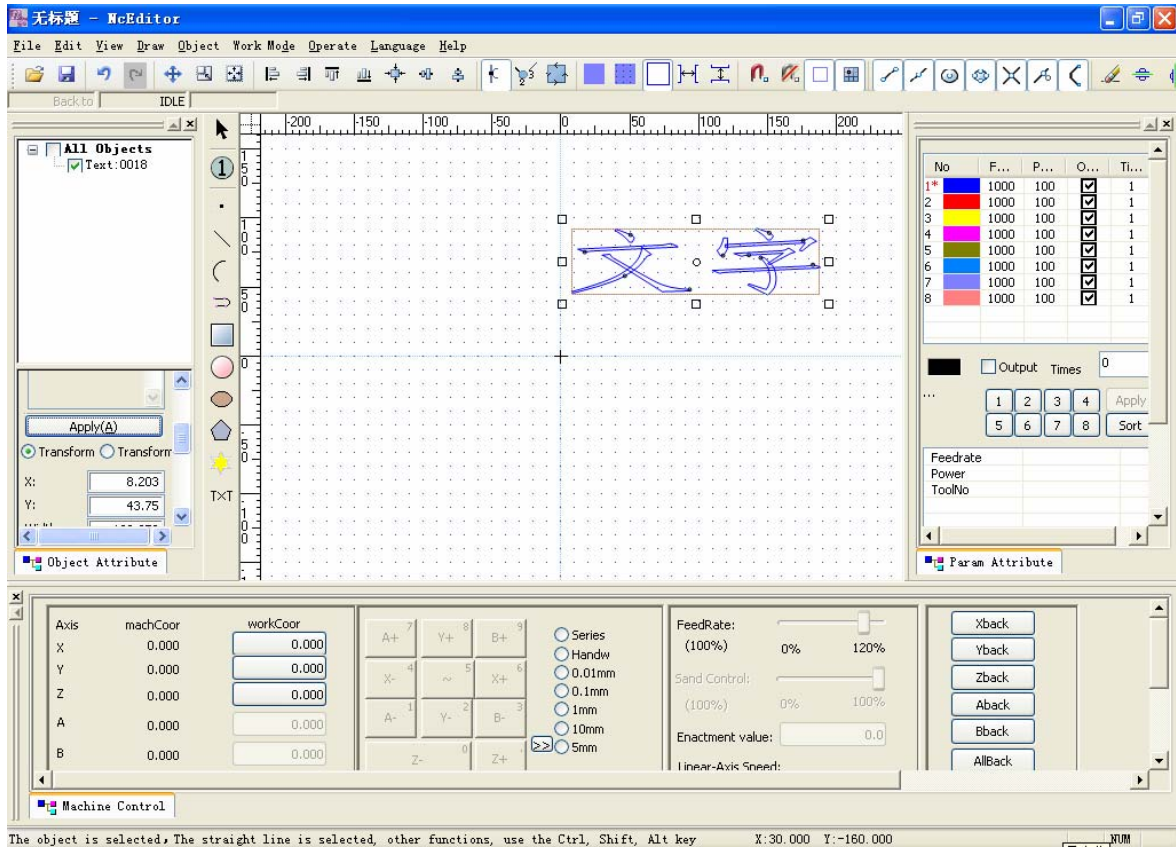
Fig. 10-79 after inputting text

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.
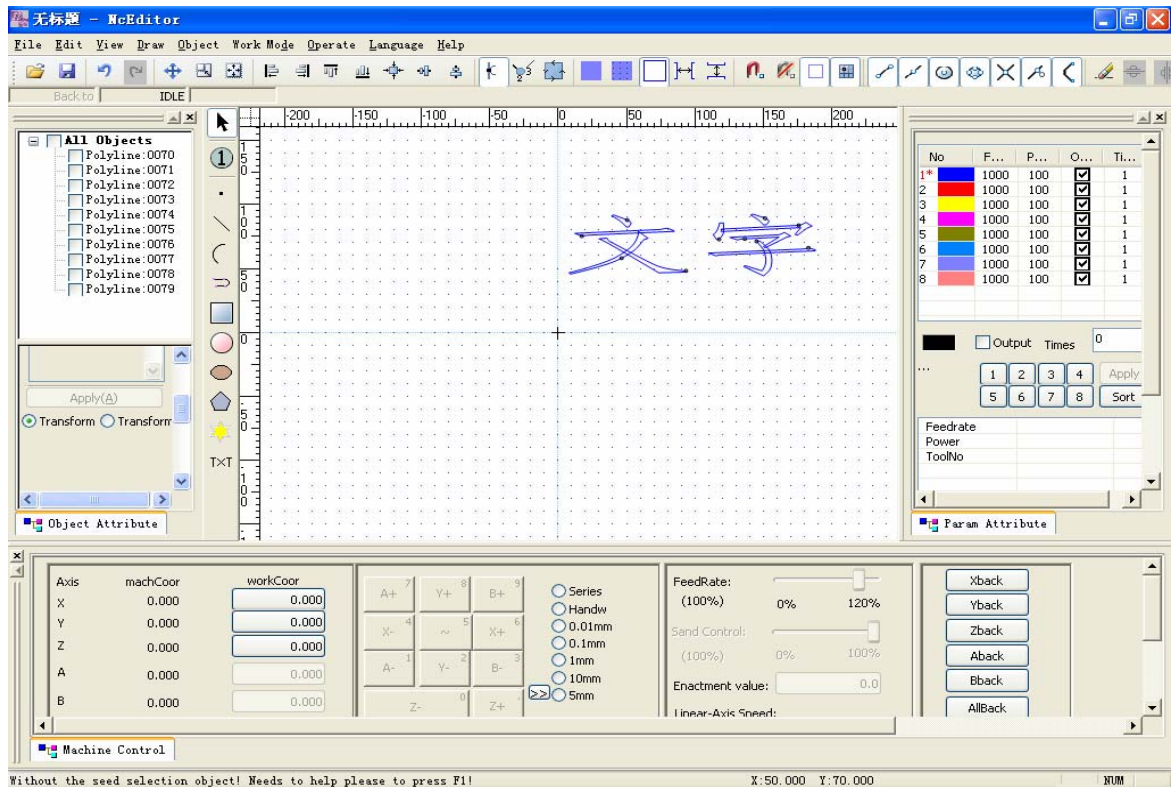

Fig. 10-80 output after [Text translate to graphics]

◆ [Dispart Cad Path]

[Dispart Cad Path] is assorting to [Manual Add Bridge], which is available after [Dispart Cadpath].

◆ [Ellipse translate to polyline]

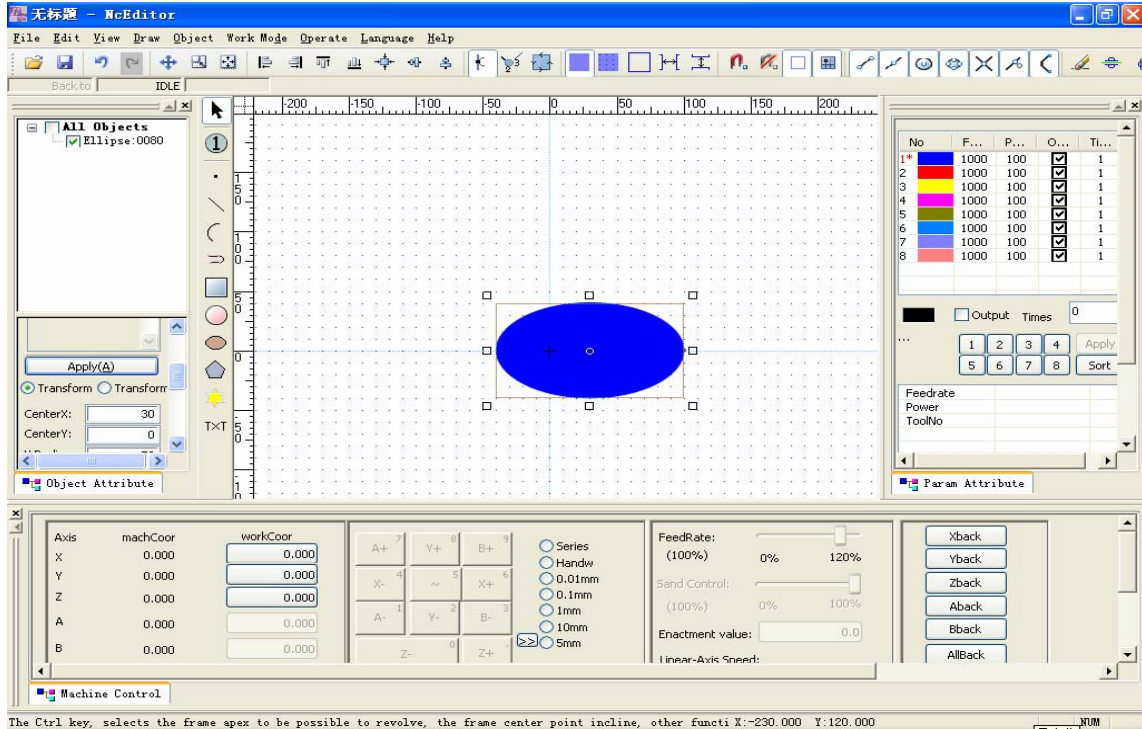Draw an ellipse as following, Fig.10-81

Fig. 10-81 example of [Ellipse translate to polyline]

Select this ellipse → choose [Ellipse translate to polyline] under right-click menu, as Fig.10-82.
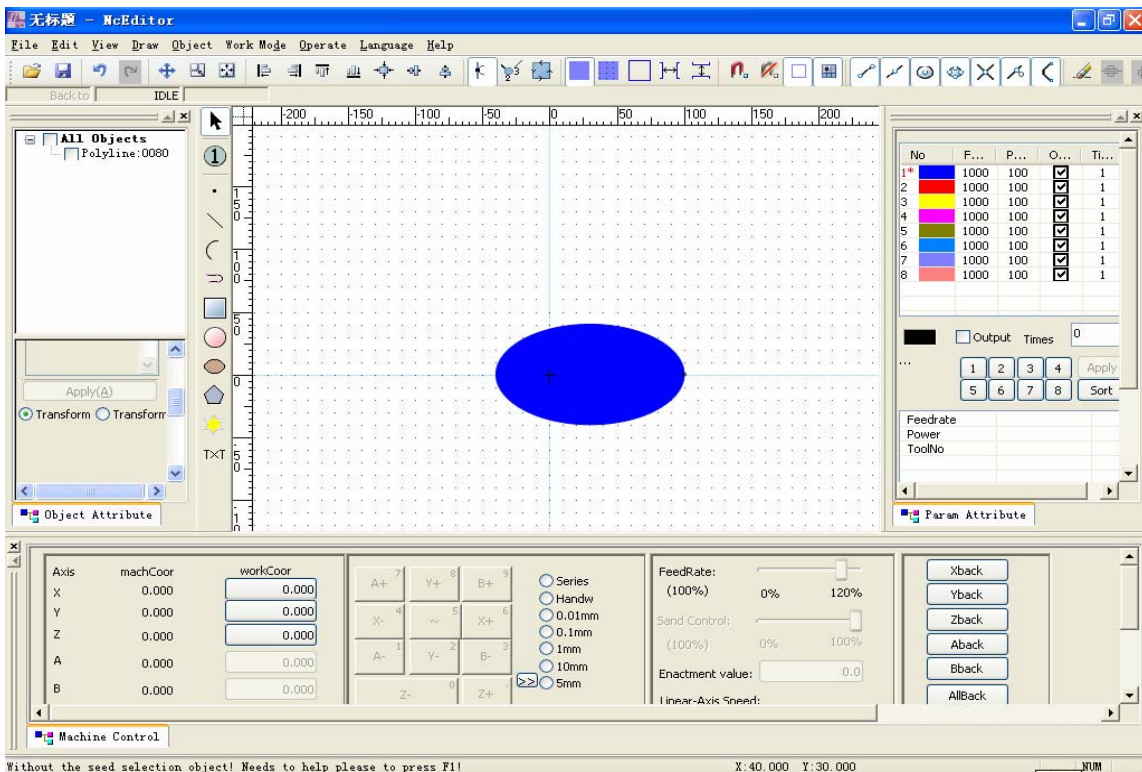

Fig. 10-82 after [Ellipse translate to polyline]

10.6.6   Operation Menu

Many options in [operation] menu are the same as that in Ncstudio, and the options with the same function won't be introduced any more.

◆   [Machining selected figure]

This function is used to machining the selected figures, after finishing machining the selected object, the software will stop, and will not process other unselected figures. User can selected one figure or multiple figures.

◆   [Machining by color]

Users can choose the color of object in drawing and also can order the color. Then the machining can be executed following the order of color, for instance, first red, then blue.

After selecting [Machining by color], a [Sort color line] dialogue will pop up, as following, Fig.10-83.
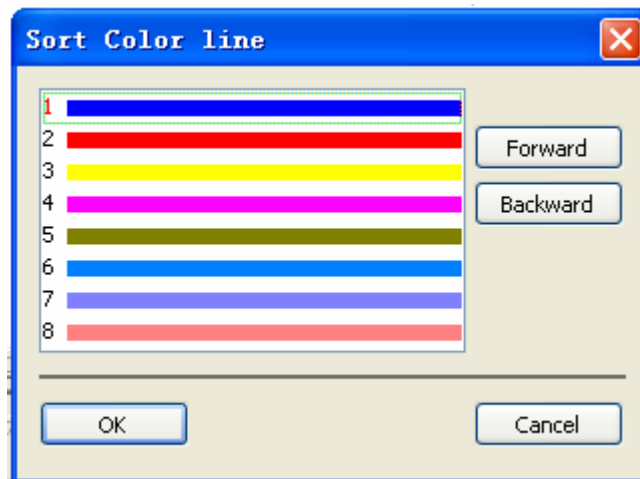

Fig. 10-83   [Sort color line] dialogue

In this dialogue, user can set the machining order of different color figures.

◆   [Machining current group]

If user chooses one group and only needs to machine this group rather than the other figures, this function can be selected. This function is only available in the current group, unavailable for single figure.

◆   [Machining from the selected objects]

There is an order for machining objects, after selecting [Machining from selected objects], only the selected objects and backward objects will be machining, the forward one will not be machining. For instance, there are 8 objects for machining, if the sequence of selected object is 3, the No. 3 to 8 objects will be machining, but No.1 and No.2 won't.

11 Ncstudio™ Appendix

There are many shortcuts in Ncstudio system for the convenience of the users. All the involved shortcuts will be listed in this Chapter.

11.1 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 its enabled precondition will be also provided.

Single number key (small keyboard is invalid) can be used to choose function section under the current mode (AUTO, MANUAL, REFER). List the key 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 key 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	**function screen 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