



LASER ENGRAVING/CUTTING
MOTION CONTROL SYSTEM

NcStudio-V9

Users' Manual

Preface

Thank you for choosing our products.

This manual will acquaint you with such detailed information of our products as system components, settings, functions, and operations, etc.

Please read it carefully before using our products and other relative machine equipments to make a better use of our products.

Because of continuous updating, the products you bought may differ from the written in this manual, and we apologize for this situation.

Company address, phone number and our website are listed here for your convenience. Any questions, please feel free to contact with us. We will always be here and welcome you.

Company Name:	Weihong Electronic Technology Co., Ltd.
Company Address:	No. 29, 2338 Duhui Rd., Minhang, Shanghai
Zip Code:	201108
Tel:	021-33587550
Fax:	021-33587519
Website:	http://www.weihong.com.cn
E-mail:	support_weihong@188.com; sales_weihong@188.com

Warnings:

1. Don't plug or pull out the cable connected with the computer when power on.
2. Please don't plug or pull out the cable of manipulation boxes when power on.
3. For safety and disturbance reduction, the shell of computer or laser cutting/engraving machine must be grounded.
4. If machine is not in using, please shut off the power timely.
5. Please pull out the attaching plugs when the machine is not used for a period of time.
6. The spindle's life-span is inversely proportional to its rotation speed.

Fast Master Chapter

This chapter applies to the condition that electrical circuit has been connected and the control card and NcStudio system have been well installed.

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

Join the terminal board to port JP1 of the card with DB37 M/F cable, add 24V power supply to the terminal board, and power on 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 and 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 a metal block 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 open, LED should be normally out, and by artificially touching the switch, LED will become light, which shows the origin signals have been received by terminal board. The same method can be taken to test other ports to make sure the correctness of the connection between terminal board and machine tool, greatly shortening the debugging time.

- Debugging Signals

Open the computer and run NcStudio software. Open [Diagnoses]→[IOPort] window, which displays 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. Polarity of normally open InPorts is “N”, and that of normally closed InPorts is “P”. After all modification has been finished, exit from NcStudio software and restart it to validate polarity modification. Afterwards, supply power to the electrical box. At this time, dots which are in front of signals of program start, program stop, zero points of the three axes and E-STOP InPorts should be in red, indicating all these signals are invalid. If one of the dots shows in green, you must examine the circuitry in electrical box and the polarity of the signal. 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 a processing program, then press [Start] button on the operation panel under [Auto] mode, and observe if the color of the dot in front of “Program Start” changes after no error is detected. This dot should be in green when the button [Start] is being pressed and in red when the button [Stop] pressed. If no change occurs to this dot, please turn to step one to check if its LED on terminal board functions normally, check if the connection of DB37M/F cable and DB50M/F cable with the card and

the terminal board becomes loose. To check the reference point signals of X-axis, Y-axis and Z-axis, imitate to touch the origin signals according to the first step. When one of the origin signals is being touched, the corresponding signal dot should be in green.

➤ Parameters Setting

Set the manufacturer parameter [pulse equivalent]. The smaller the pulse equivalent is, the higher the resolution is. But too small pulse equivalent will affect the maximum feedrate. Users of model machines can generally set the pulse equivalent as 0.001 mm/P (The corresponding maximum feedrate is 9600 mm/min) or as 0.0005mm/P (The corresponding maximum feed speed is 4800mm/min). For users who are not critical about the accuracy, pulse equivalent can be set a little larger, such as 0.002mm/P (The corresponding maximum feed speed is 19200mm/min) or 0.005mm/P (The corresponding maximum feed speed is 48000mm/min); After the pulse equivalent has been confirmed, please compute the electronic gear ratio of servo driver in accordance with the value of pulse equivalent. $\text{Electronic gear ratio} = \text{encoder resolution} \times \text{pulse equivalent} \times \text{mechanical deceleration ratio} / \text{pitch}$; Thereinto, $\text{mechanical deceleration ratio} = \text{reducer rotation speed input} / \text{rotation speed output} = \text{driven gear tooth number} / \text{driving gear tooth number}$. Then move the machine manually to make sure the correctness of moving direction of each axis, be noted NcStudio 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 moving far away from the operator. (If Y-axis movement is the worktable movement, the positive direction is the direction of the worktable moving towards the operator). If the direction is not right, you can revise [Axis Direction] of manufacturer parameters or the relative servo driver parameters. Then set [Worktable stroke range] of manufacturer parameters on the basis of the actual machine tool dimension to make the software limit into function. Then 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 [direction of backing to mechanical reference point].

➤ Load a processing file and start machining (or perform manual machining).

Contents

1. Summarization of Laser Cutting.....	- 1 -
1.1. Software Features	- 1 -
2. Installation of Laser Cutting System.....	- 4 -
2.1. The Basic Configuration of NcStudio.....	- 4 -
2.2. Installation of NcStudio	- 4 -
2.2.1. Installation of NcStudio Software	- 4 -
2.2.2. Installation of Control Card & Other Devices	- 6 -
2.2.3. Other Installation Problems	- 9 -
2.2.4. Uninstall of NcStudio System.....	- 9 -
3. Basic Concepts of Laser Cutting System.....	- 10 -
3.1. Operation Mode and State	- 10 -
3.1.1. Operation Mode	- 10 -
3.1.2. Operation State	- 11 -
3.2. Machine Coordinate System	- 11 -
3.2.1. Mechanical Coordinate System	- 12 -
3.2.2. Workpiece Coordinate System	- 12 -
4. Functions & Features of NcEditor in Laser Cutting.....	- 13 -
4.1. Integral Interface.....	- 15 -
4.2. Object List Window	- 15 -
4.3. Object Attribute Window.....	- 16 -
4.3.1. Classification of Lead-in/out Line.....	- 16 -
4.3.2. Set Introduce (Lead-in/out Line)	- 17 -
4.3.3. Fill or Not.....	- 20 -
4.3.4. Selection of Machining Direction	- 21 -
4.3.5. Precise Positioning of Machining Objects.....	- 21 -
4.3.6. Object Information Box	- 22 -
4.4. Origin Setting.....	- 23 -
4.5. Machining Parameters Attribute Bar	- 23 -
4.5.1. Color Window of Machining Parameters	- 23 -
4.5.2. Modification Window of Machining Parameters.....	- 23 -
4.6. Figure Selection.....	- 24 -

4.7. Menu Function	- 25 -
4.7.1. File Menu	- 25 -
4.7.2. Edit Menu	- 28 -
4.7.3. View Menu	- 29 -
4.7.4. Draw Menu	- 36 -
4.7.5. Object Menu	- 40 -
4.7.6. Work Mode Menu	- 70 -
4.7.7. Operate Menu	- 70 -
4.7.8. Language Menu	- 74 -
4.7.9. Help Menu	- 74 -
4.8. Machine Control Section	- 75 -
4.8.1. Auto Mode	- 75 -
4.8.2. Manual Mode	- 80 -
4.8.3. BKRef Mode	- 82 -
5. Parameters of NcStudio in Laser Cutting	- 84 -
5.1. Methods for Parameters Modification	- 84 -
5.2. Parameter List of Manufactures (Including Operators)	- 84 -
5.2.1. General Parameters	- 86 -
5.2.2. LaserJet Parameters	- 105 -
6. General Overview of Laser Engraving	- 108 -
7. Features of NcEditor in Laser Engraving	- 109 -
7.1. Main Interface	- 109 -
7.2. Machine Control Bar	- 109 -
7.2.1. Color Window of Machining Parameters	- 110 -
7.2.2. Machine Control Section	- 111 -
8. Menu of NcEditor in Laser Engraving	- 115 -
8.1. File Menu	- 115 -
8.2. Edit Menu	- 117 -
8.3. View Menu	- 118 -
8.4. Draw Menu	- 118 -
8.5. Object Menu	- 118 -
8.6. Operate Menu	- 118 -
8.7. Language Menu	- 119 -

8.8. Help Menu	- 119 -
9. Parameters of NcEditor in Laser Engraving.....	- 121 -
9.1. Parameters Setting.....	- 121 -
9.2. Operators' Parameters.....	- 121 -
9.2.1. Program Parameters	- 121 -
9.2.2. Machining Parameters.....	- 124 -
9.2.3. Control Option.....	- 125 -
9.3. Manufacturers' Parameters	- 126 -
9.3.1. Workbench Parameters	- 126 -
10. Appendix	- 130 -
10.1. Shortcut Key Appendix.....	- 130 -
10.2. Software Operation Procedure	- 132 -

1. Summarization of Laser Cutting

Laser cutting refers to a thermal cutting method, which makes the stainless steel melt and evaporated with the energy released by laser beam irradiating surface of steel plate. A blank can be precisely cut complicated shapes by laser without further handling.

As the motion control system for laser engravers (laser cutting machines), NcStudio laser cutting system, copyrighted by Weihong, is independently developed by Weihong Electronic Technology Co., Ltd.. It can directly support G code, DXF format, PLT code format and ENG code of JDPaint processing files that are created by CAD/CAM applications such as UG, MasterCAM, CASMate, ArtCAM, AutoCAD, CorelDraw, and so on.

NcStudio takes great advantage of 32-bit computing and multi-tasks. Meanwhile, its standard interface is dependable and user-friendly.

Apart from functions of automatic mode, manual mode and backing to the reference point mode, this CNC system features the functions as following: simulation, processing time forecast, loading processing track in advance, dynamic tracking, breakpoints resuming (program block skip function), etc.

1.1. Software Features

NcStudio laser cutting system includes the following functions:

- Automatic processing. It completely supports G code, HP PLT format, DXF format and JDPaint ENG format of ISO standard.
- Manual processing. The 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.
- Incremental feeding. It is convenient for the user to set accurate feed length, and the step-size can be adjusted.
- Supporting special user data input function. The user can directly input G code online and immediately execute it.
- Supporting processing wizard. To execute functions such as contour milling and special lines, the user only needs to input several parameters.
- Supporting single block mode. The user can set the program to be executed as [Single Block], which provides very good support for error diagnoses and failure recovery.
- Supporting advanced auto functions of breakpoint memory and program block skip.

- Supporting “save / restore workpiece origin”.
- Supporting exactly returning to machine origin (reference point).
- Supporting feedrate override on-line adjustment. During the machining process, the user can adjust the feedrate override at any time. The least value is “0” that indicates the processing is suspended, and the maximum value is 120%.
- Supporting new function of high-smooth speed connection algorithm. In a usual numerical control system, the connection speed between two G codes is commonly a fixed value (e.g., it may be “0” or a certain small fixed value). However, in this new version of CNC 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, eliminating the speed chatter marks left on the processing surface.
- Supporting analog display. Processing results can be observed from different angles through simple operations. Thus, the user can learn the processing results more intuitively.
- Adding new function of loading processing path in advance. After the processing file is loaded, the system will automatically analyze and output the processing tracks. The user can directly call out the tracks to view.
- Supporting simulation function. This function can simulate machining process and finish in a short time. Meanwhile, it can check if there is any mistake with the program and if the processing result can meet the user’s satisfaction. Moreover, it can count the actual processing time precisely.
- Supporting keyboard operation. To satisfy the user’s need, we offer a strong support to the keyboard operation.
- New function of log. The system offers powerful log function to meet the user’s demand.
- Adding an embedded processing file manager. The user only needs to save the processing file into the specified directory; NcStudio will then administrate it in an enclosed file manager.
- Adding a built-in processing file editor. The user can introduce a processing file into file editor at any time to edit and modify it.
- Supporting displaying file processing information function. By simulation or actual processing,

[file processing information] window can help the user to count some important info, such as cutting time, machining range and workpiece quantity completed.

- Supporting particular [Parameters Auto backup] function. In this window, the system can automatically save parameters' settings information, which saves the user a lot of time and trouble to set parameters time and time again.
- Supporting backup for 8 parameter items in [Parameters Back Up] window. The user can save different parameters' settings of different processing files, and read them at any time.
- Particular [I/O Port] window can do map simulation, test and regulate polarity, which helps the user well realize hardware detection, processing supervision, and fault diagnoses.
- Adding PLC module, through which the user can control I/O ports.
- Supporting intelligent prediction; the system will analyze situation in the forward or backward distance, in order to decide the interpolation strategy and improve the integral smoothness of workpiece.
- The maximum length of machining file is 2 G byte.
- Supporting [auto center]; the user can record current mechanical coordinate after setting machine origin, after the execution of [auto center], the recorded value will be set as current workpiece coordinate origin.
- Supporting reverse cutting, used in repetition cutting.
- Supporting MPG (Manual Pulse Generator); the user can determine whether to use MPG or not according to his needs.
- Supporting multi-language; currently, English and Chinese have been embedded in the software, other languages will be added in demands.
- Supporting backlash compensation, lead screw error compensation, quadrant error compensation, etc.

2. Installation of Laser Cutting System

The installation of NcStudio laser cutting application is quite simple. Advanced users or those who have been familiar with the application can refer to the fast master chapter.

2.1. The Basic Configuration of NcStudio

◆ The Host Computer

CPU:	Main frequency 1G or above.
Memory:	512M or above.
Hard disk:	20G or above.
Display adapter:	1024*768 at least
Display:	14" VGA or above
CD-ROM:	4X or higher (optional)
Mainboard expansion slot:	More than 2 PCI slots

2.2. Installation of NcStudio

If there is already an old version of NcStudio, please delete it, and then install the new version. Regarding how to delete the old version, please turn to chapter 2.2.4 “uninstall of NcStudio system”. Or directly install the new NcStudio, which will lead to overriding the former data.

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

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, for which please refer to “Manufacturers' Manual” for detailed information.

2.2.1. Installation of NcStudio Software

Software installation can be divided into several steps as following:

- 1) Supply power to the computer, start it up to enter the operation system automatically.
- 2) 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 pop-up dialogue box is the [Language of the theme] dialogue, see Fig. 2-1. If you click

[English], the system will take the English interface as the default interface after installation. For the same reason, if you click [Chinese], the system will take the Chinese interface as the default interface. In V9 version, English and Chinese can also be switched in the system. If there is already a version of 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, as Fig. 2-2.



Fig. 2-1 Language selection dialogue

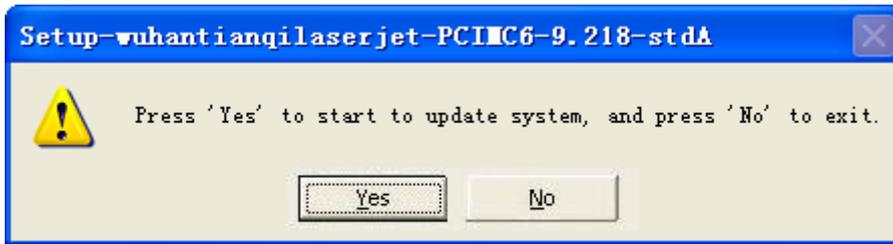


Fig. 2-2 Update reminding dialogue

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

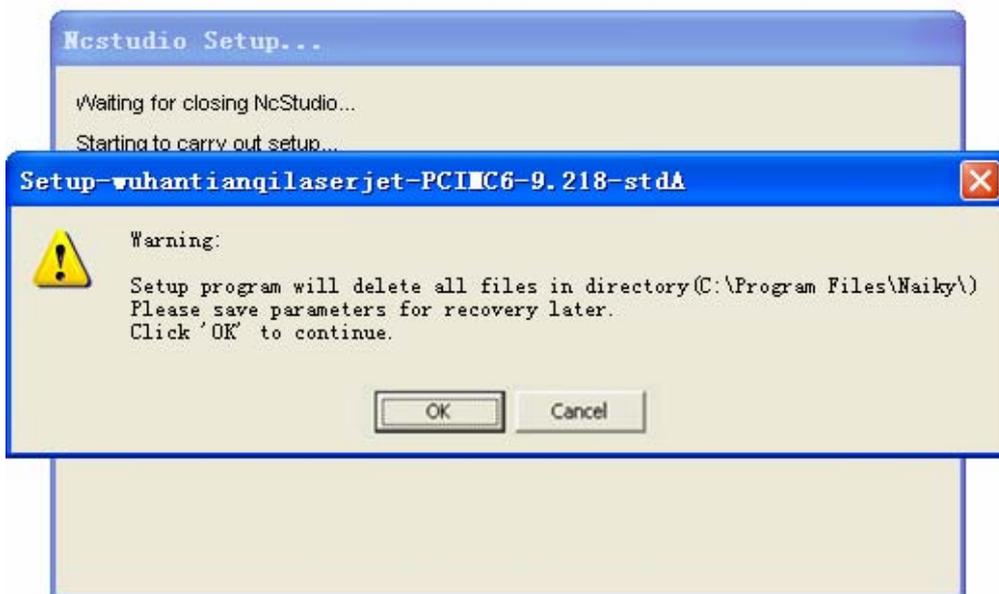


Fig. 2-3 Remind to save parameters before installation

- 4) Click [OK]. Installation begins. NcStudio will be installed to the default directory, C:\Program Files\Naiky. Installation progress bar will show the installation speed. See Fig. 2-4. After installation, a prompt will ask the user whether to shut down the computer (We recommend closing the computer to facilitate the installation of the control card.) See Fig. 2-5.

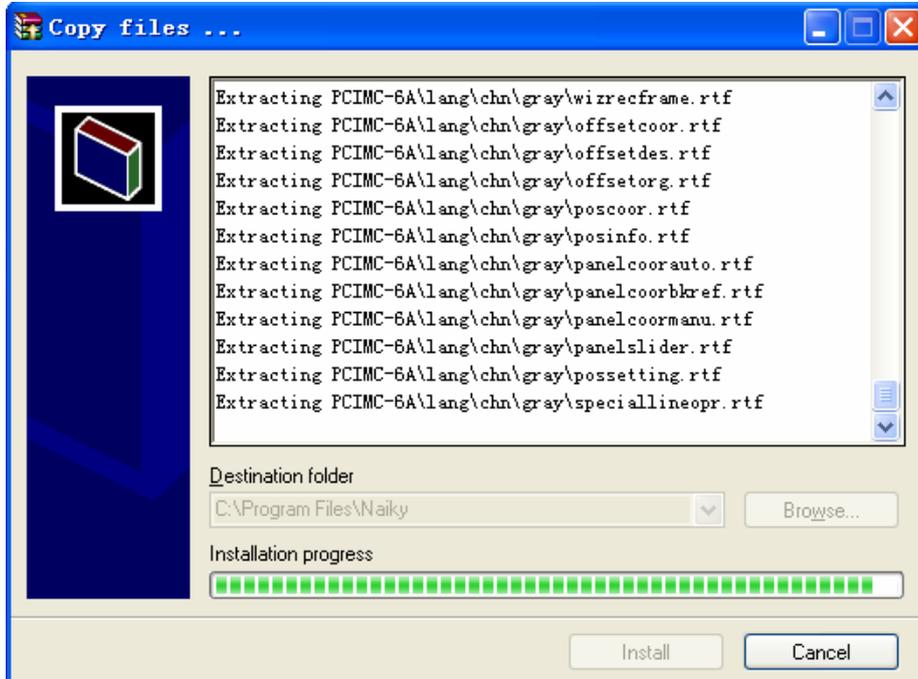


Fig. 2-4 Software installation interface



Fig. 2-5 Prompt whether to shutdown the computer (recommend: OK)

- 5) Then the software installation has been completed.

2.2.2. Installation of 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 and matched expansion slot.

Note:

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

- 2) Similar to the installation of the control card, please insert peripheral equipments into the corresponding slots or ports and make sure they are firmly fixed without fluttering.
- 3) The installation has been completed now. Restart the computer.

Restart the computer after installation of the motion control card & other devices. The operation system will report a new hardware has been found, and eject a hardware update guide dialogue. See Fig. 2-6. Choose [No, not at this time] and click [Next] to go to next page. See Fig. 2-7. Choose [Auto Install the Software] and click [Next]. See Fig. 2-8. Then the operation system will install the driver application for NcStudio automatically. Up to this point, the whole installation of NcStudio software and the motion control card have been completed. To start NcStudio software, click [Start] menu on the desktop →click [Applications] →click [NcStudio]. Or start it by the icon on the desktop or by other shortcuts.



Fig. 2-6 Page one of hardware update guide



Fig. 2-7 Page two of hardware update guide

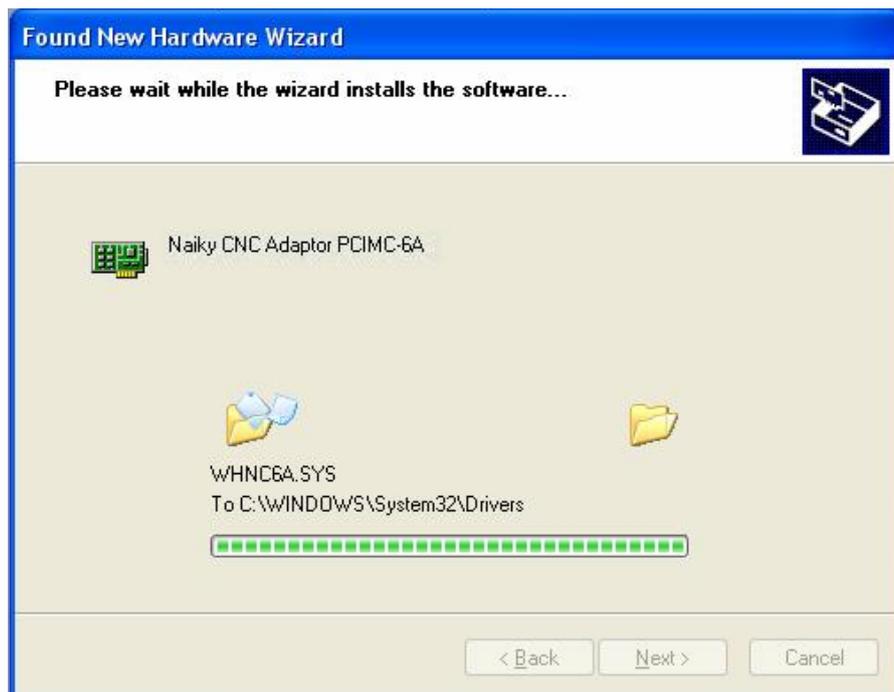


Fig. 2-8 Page three of hardware update guide

Note:

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

2.2.3. Other Installation Problems

If you don't follow the above installation procedures and insert the motion control card before software installation, the computer will remind you to install driver for motion control card installation after restarted. Please cancel the dialogue of installing driver, and directly install NcStudio software, then restart the computer. After restart, the operation system will request once again to install driver for the motion control card, at this time, you can set it according to the hardware guide of the last chapter. After all the installation has been finished, you can use NcStudio software.

2.2.4. Uninstall of NcStudio System

NcStudio is green software which has the following advantages: easy to install and uninstall, and no need to write installation information into registry. To uninstall NcStudio system, you only need to delete its document folder under "C:\Program Files\Naiky", NcStudio item in [Applications] of [Start] menu (as shown in Fig. 2-9) and the shortcut menu on the desktop.



Fig. 2-9 NcStudio menu under [Applications] of [Start]

3. Basic Concepts of Laser Cutting System

Laser cutting system is a comparably complicated system involving many concepts, such as workpiece coordinate system, mechanical coordinate system, operation mode, operation state, etc, which are unavoidable. Therefore, the user should grasp these concepts before using this laser cutting system.

3.1. Operation Mode and State

3.1.1. Operation Mode

It is necessary for the user to understand operation mode correctly, since there are only three modes of machining operation in all as below.

◆ **Reference Point Mode (Backing to the Reference Point/Machine Origin)**

[Back to the Reference Point] mode is the process to unify the internal coordinates and the machine actual coordinates. Therefore, in the process of machine tool start-up, it is essential to “back to the reference point”. Every time the system is started, the default operation mode is reference point mode, which is very convenient for the user to implement [Back to the Reference Point] action.

◆ **Auto Mode**

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

◆ **Manual Mode**

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

- Under [Jog] mode, the machine will start motioning once a manual button is pressed, and won't stop until the button is released.
- Under [Stepping] mode, pressing down a manual button and then releasing it, the machine will move a specified distance. The distance is thought as step-size, which is divided into five grades: 0.01mm, 0.1mm, 1mm, 5mm, 10mm. Moreover, step-size can be customized. Therefore, the displacement of the machine tool can be accurately controlled under [Stepping] mode.
- Under [Handwheel] mode, the user can control the machine tool by handwheel. Each time the user turns the handwheel a lattice, the machine will move a step distance.

3.1.2. Operation State

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

◆ IDLE State

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

◆ ESTOP State

This is an abnormal state. When there is a hardware fault or when the [Reset] button is pressed, the system will enter into ESTOP state and implement the predetermined protection actions, such as closing spindle motor 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, the 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, the system enters into Running State.

◆ PAUSE State

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

◆ LOCK State

Lock state is an internal state which occurs at the time of status switching. Normally, the user may not meet with this state.

3.2. Machine Coordinate System

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

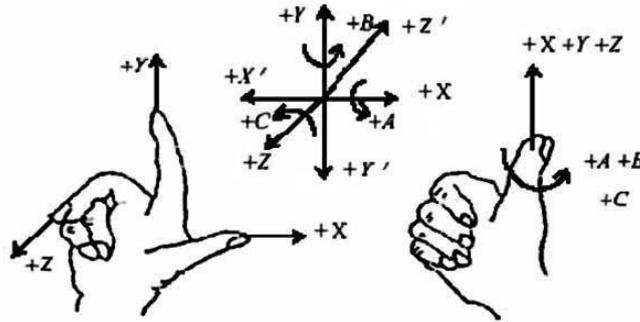


Fig. 3-1 Demonstration for right hand principle

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

—Z-axis is identical with axial line of spindle, and 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 a single column milling machine, if the user faces the tool and spindle and looks in the column direction, right moving direction is the positive direction of X-axis (+ X).

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

3.2.1. Mechanical Coordinate System

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

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

3.2.2. Workpiece Coordinate System

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

4. Functions & Features of NcEditor in Laser Cutting

The features of NcEditor are as following:

- Designed for planar cutting, editing planar pictures and outputting data of planar machining objects;
- Supporting directly processing and mating with NcStudio;
- Supporting the element creation of basic machining objects, including point, line, polyline, rectangle, circle, ellipse, arc, etc;
- Supporting setting processing condition for a single object, distinguished by color, and parameters setting function;
- Supporting object change function, including translation, rotation, mirror image, zoom, slant, combination and break-up of objects;
- Supporting view change function, including translation, zooming in, zooming out, fit to window;
- Supporting spot catch function, such as catch of node and midpoint of a straight line, catch of center and quadrant point of a circle;
- Supporting view grid and ruler displaying function;
- Supporting grid catch function;
- Supporting mouse dragging editing of nodes function to adjust the size of an object;
- Supporting displaying and modification functions of attribute of an object;
- Supporting multi-object alignment functions, including “align right”, “align left”, “align top”, “align bottom”, “align vertical line”, “align center point”, “align level dispersion” and “align vertical dispersion”;
- Supporting object selection function and mouse dragging function of objects;
- Supporting “ManualSort” & “AutoSort” (sorted by start point and end point as the benchmark) in machining, which optimizes thorough cutting jobs;
- Supporting auto and manual bridge addition function;
- Supporting removal of duplicate objects;
- Supporting inserting and editing a file of other format into the existed processing file;

- Supporting embedded photo point of MVA (machine visual aid) system;
- Supporting saving parts of currently loaded processing file as another processing file;
- Supporting “Undo” and “Redo” which can undo or redo almost all previous operations;
- Supporting basic editing functions: copy, cut and paste;
- Supporting machining object format display, and display & modification of object name;
- Supporting analog processing function;
- Supporting read and transformation of ENG files, G code files, DXF files and PLT files;
- Supporting Chinese and English which can be switched;
- With round spot that can be reset and adjusted; after load of a file, the round spot can be set as the start point of machining;
- Supporting viewing and editing of a text file;
- Easy to operate, fast and quick to learn and master.

4.1. Integral Interface

The integral interface of NcEditor includes: [title bar], [menu bar], [toolbar], [drawing toolbar], [status bar], [machining parameters attribute column], [object editor space], [object list bar], [ruler bar]. Among them, the [object list bar] includes: [object list window] and [object property window]; [machining parameters attribute column] includes: [machining parameters color window] and [machining parameters modification window]. Please refer to Fig. 4-1 for the concrete interface.

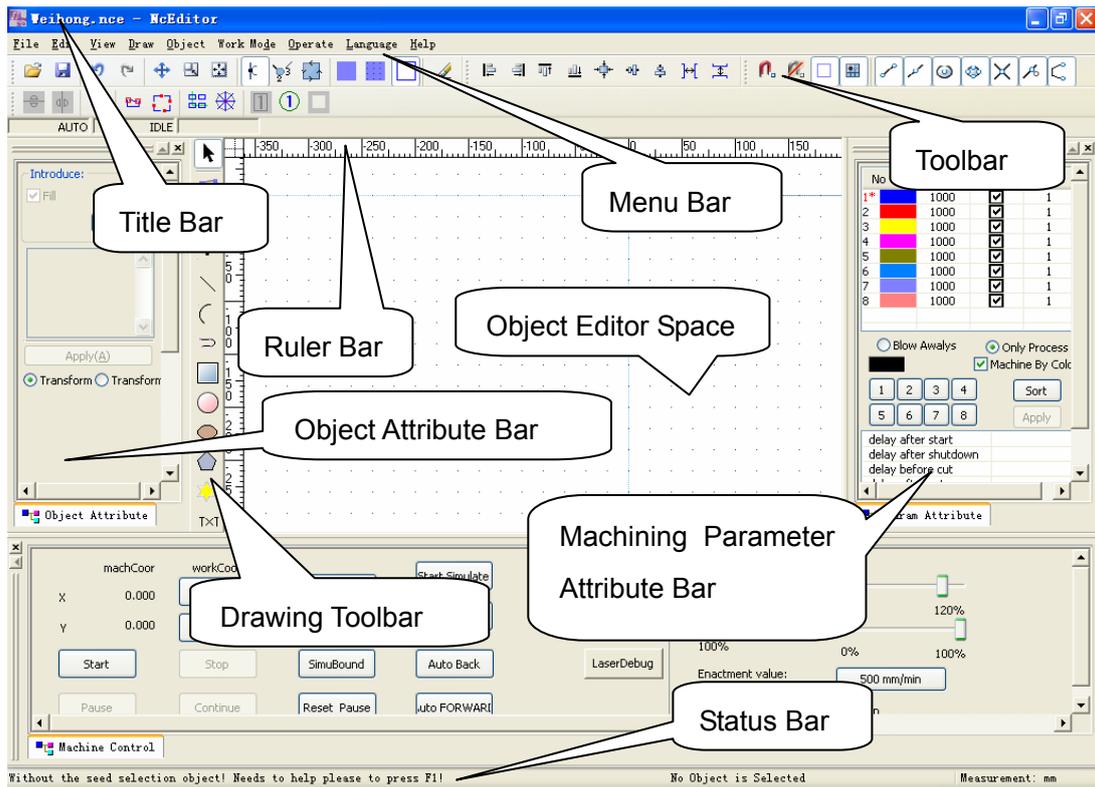


Fig. 4-1 Integral interface

4.2. Object List Window

Objects in [Object list window] are corresponding to what in [Object editor space]; when an object is drawn in the [Object editor space], a corresponding object will be added automatically in [Object list window]; when an object is selected or deselected by [Object list window], the corresponding object in [Object editor space] will be executed the same operation.

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

- Selecting a or several check boxes ahead of object (s) can select one, or several objects;
- Cancelling a or several selected check boxes ahead of object (s) can deselect one or several objects;

- The multiple selected objects can be set as one group;
- Breaking the objects in one group will make them back to the previous group;
- Setting a group as the current working group: when a file includes multiple groups, to edit one of these groups, set the group as the current group.

4.3. Object Attribute Window

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

4.3.1. Classification of Lead-in/out Line

[Lead-in/out line] is exclusive for closed figures, mainly classified into 2 types: straight-line type and arc type, which can also be sub-divided according to different parameter setting. The type of [lead-in/out line] is chosen in terms of the cutting.

1) [None Introduce (Lead-in/out Line)]

It means no lead-in/out line; the user can decide whether to set lead-in/out line or not.



Fig. 4-2 None [lead-in/out line]

2) [Straight-line Type Introduce (Lead-in/out Line)]

Opening introduce (lead-in/out line): both cutting-in line and cutting-out line are straight-lines, and the cutter keeps processing until reaching a certain distance away from cutting-in point, and then it cuts out along the cutting-out line, not falling from the blank.

Tine introduce (lead-in/out line): cutting-in edge and cutting-out edge are all straight lines, the tool cuts in along one straight line of the taper angle and cuts out along the other straight line. For instance, [lead-in/out line] is on the peak of a rectangle, or on the top of other cross points.

Kerf introduce (lead-in/out line): when the tool cuts to the tail (cutting-in point), the tool will return back along the same path and cut out after traveling a certain distance.

The above three types can be achieved by setting the related parameters of lead-in/out line.

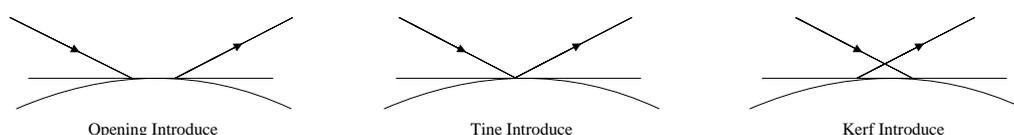


Fig. 4-3 Lead-in/out line of straight-line type

3) [Arc Lead-in Line]

Cutting-in edge and cutting-out edge are both arcs; the tool cuts in along one edge of the arc and cuts out along the other edge, such as, lead-in line of a circle, an ellipse and an arc, etc. Just like the lead-in/out line of straight-line type, the distance between cutting-in point and cutting-out point in [Arc Lead-in/out Line] can also be custom, for instance, the sketch map of lead-in line is as below when the distance between cutting-in point and cutting-out point is “0”.

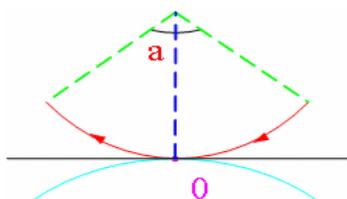


Fig. 4-4 [Arc lead-in/out line]

Note: there is no definite limitation on the usage of straight-line type lead-in/out line and arc lead-in/out line. They are interchangeable. For instance, setting an arc lead-in/out line for a rectangle vertex, the system will figure out the fine lead-in/out line automatically; the type of lead-in/out line is chosen in terms of the cutting; besides, the use of lead-in/out line is for precise machining, the user can decide whether to set it or not.

[Lead-in/out line] can be divided into internal lead-in/out line and external lead-in/out line according to position, which mainly depends on the reserved parts after machining.

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

- External [Lead-in/out line]: the inner part of object will be reserved, while the outer one will be abandoned.
- Internal [Lead-in/out line]: the outer part of object will be reserved, while the inner part will be abandoned.

4.3.2. Set Introduce (Lead-in/out Line)

Method One: press the button  on [drawing toolbar] and select a closed object →press the button  to eject “Set Introduce” dialog.

Method Two: press the button  on [drawing toolbar] and select a closed object →right-click the object to eject a menu bar → select [Set Introduce] to eject “Set Introduce” dialog.

Method Three: select a closed object by the button  on [drawing toolbar] →choose the [Object] menu and select [Set Introduce] option to eject the following dialog.

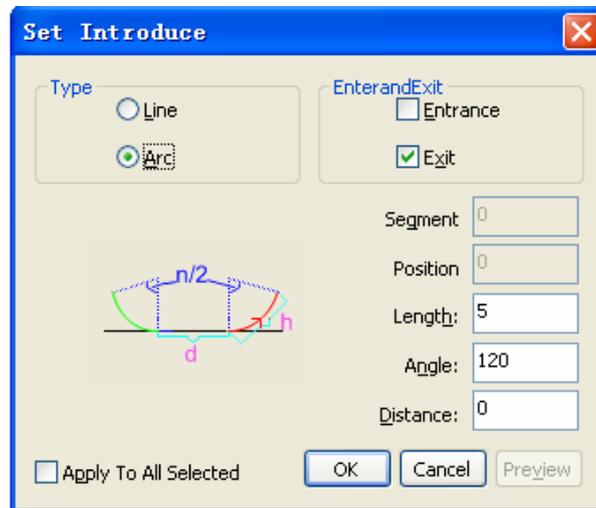


Fig. 4-5 Lead-in/out line setting dialogue

In the dialogue of [Set Introduce], the user can set lead-in/out line by parameter setting, and the concrete definitions of each parameter are as following:

[Segment]: there are probable N straight lines (N is a natural number) on the selected object. "Segment" refers to the sequence number of the segment where the lead-in/out line is on among all the segments of the object (It is used to fix on which segment the location of lead-in line is).

Sort segment: choose a closed object or the combination of multiple objects, there will be a mark on the object (s). Choose [View direction] under [View] menu; the direction can be modified in [object attribute window]. Sort begins from the mark, in line with the machining direction. The first order number is "0", and the user can set the segment number according to his own desire.

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

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

[Angle]: the angle size between the two sides of lead-in/out line;

[Distance]: the distance between cutting-in point and cutting-out point of lead-in/out line; if the value of this item is positive, the lead-in line will be opening one, if "0", fine one, if "negative", kerf one.

[Apply to All Selected]: if this function is not selected, after pressing [OK], the user needs to manually set lead-in/out line by clicking on each object. If selected, after pressing [OK], all selected objects will be produced lead-in/out line automatically. When this function selected, after pressing "OK" and "Preview", there will be a pop-up dialogue as following if the segment setting is not fit for all the objects (different range will be displayed for various objects, which depends on the concrete situation).

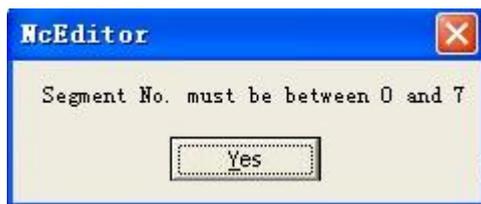


Fig. 4-6 Inquiry dialogue

An example for setting the lead-in/out line:

Note:

[Lead-in/out line] is only available to closed objects. For polylines, only polylines finished with shortcut key "C" can be set lead-in/out line.

Draw a triangle with polylines, and then finish drawing it by shortcut key "C" (note: the input method should be English), as following:

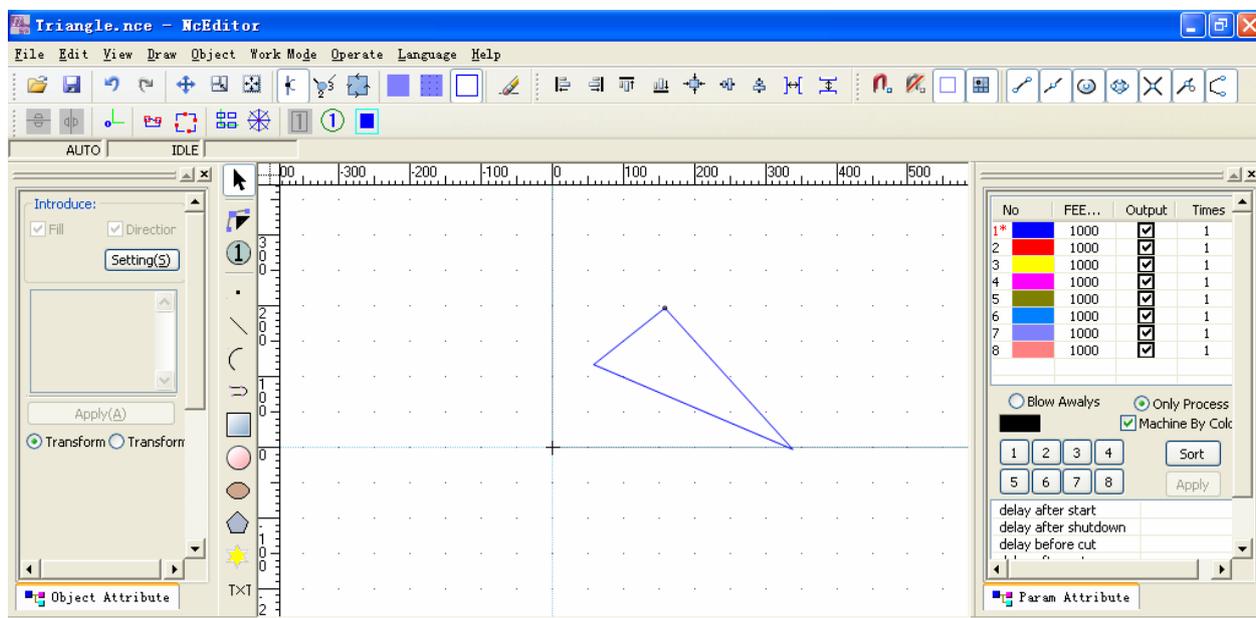


Fig. 4-7 Draw a closed triangle with polylines

Select this closed object, then select **Setting(S)** in the object attribute window or select "Set Introduce" under right-click menu or under "Object" menu, and then input the data as shown in the following dialog.

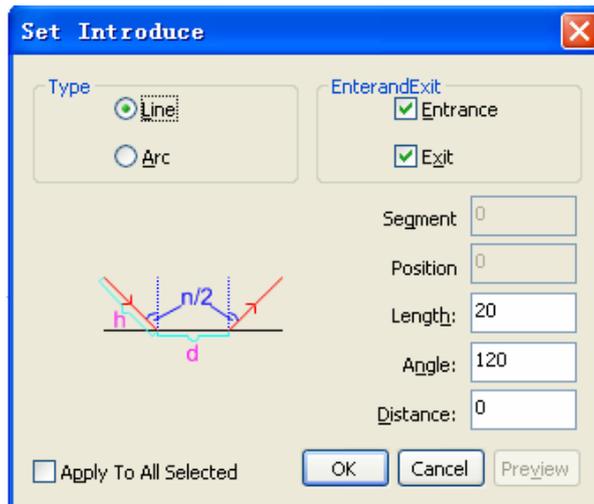


Fig. 4-8 Set lead-in/out line

After clicking [OK], click a position on the object as the position of lead-in/out line, the output is as following.

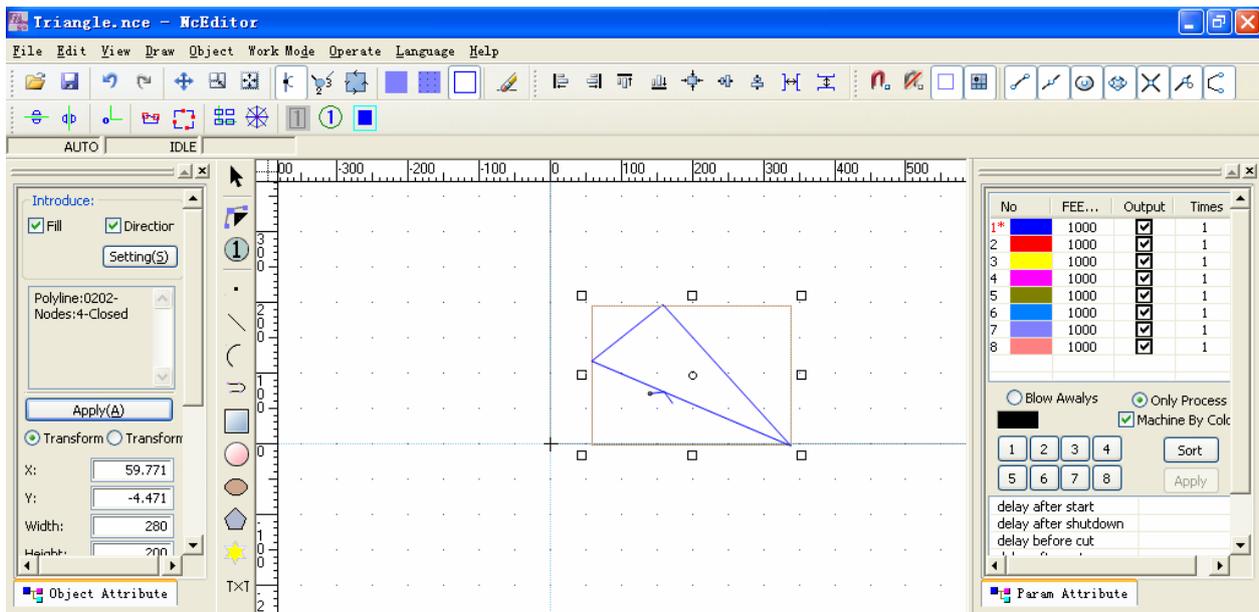


Fig. 4-9 Output after setting lead-in/out line

4.3.3. Fill or Not

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

- If the inner parts of objects are to be reserved, select [fill], shown as Fill;
- If the outlines of objects are to be reserved, select not fill, shown as Fill, the inner parts of objects to be milled.

4.3.4. Selection of Machining Direction

Selection of machining direction is used to change the machining direction of objects, so the user can adjust it according to cutting method, texture of workpiece, cutting direction, etc.

The concrete procedure is as following:

- Open “View” menu, and then select “View Direction” to show the machining direction of all objects in [Object editor space];
- Select an object;
- In [Object attribute window], select Director , the selected object will reverse its machining direction.
- To cancel the alteration of direction, cancel the check state of the choice box, i.e. Director.

4.3.5. Precise Positioning of Machining Objects

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

For example: draw a circle in [Object editor space], and then select this circle, as following:

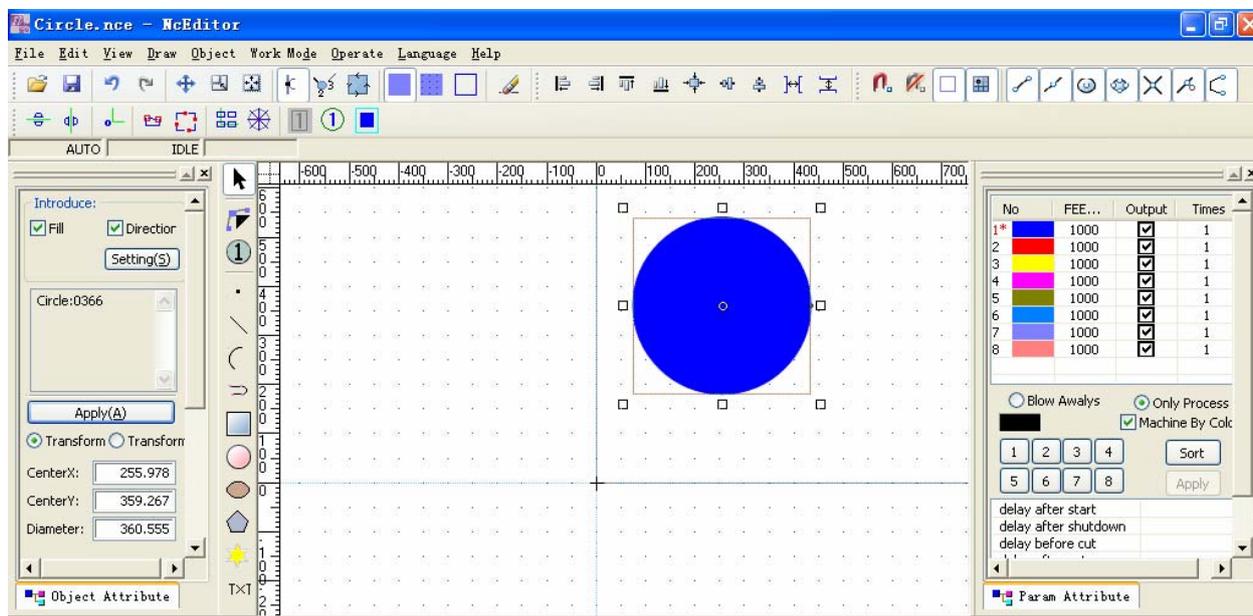


Fig. 4-10 Positioning of a circle

Input the data as following in [Object attribute window]:



Fig. 4-11 Precise positioning window

After inputting the data, press “Apply” to precisely locate this circle to the position centered at (-300, -300) with radius 212.132.

If the selected object is a straight-line, a rectangle, an arc or a point, the precise positioning dialogue will alter accordingly.

Note that precise positioning is unavailable for objects drawn by polylines currently.

4.3.6. Object Information Box

When one or multiple objects in [object editor space] are selected, its (their) detailed information will be displayed in object information box, such as [name], [node] and [fill or not], so as to help the user check the detail information of the object (s) to avoid misunderstanding of its (their) attribute under different views.

For instance, draw a figure, and select it, as below.

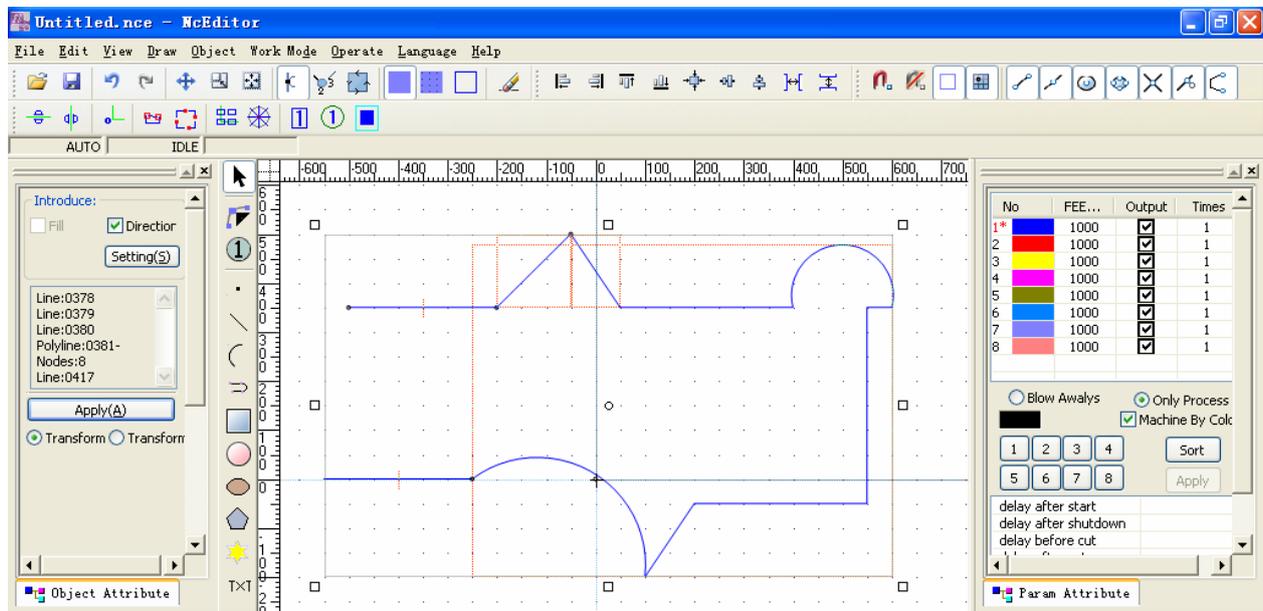


Fig. 4-12 An example

From the picture above, the [object information box] has displayed the name and node of objects, etc, as following.

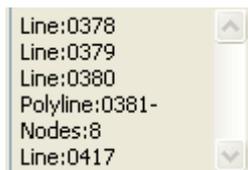


Fig. 4-13 Object information box

4.4. Origin Setting

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

In the crossing of vertical ruler and horizontal ruler, there is a mark ; click the cross center and drag it into [object editor space] to release it at the required origin position, which is also the origin of workpiece coordinate.

The advantages of setting an origin:

- The position of objects in workpiece coordinate can be positioned more precisely;
- When loading a file, the workpiece position in WCS can be positioned more precisely;
- It will be more convenient to set origin as the beginning of machining.

4.5. Machining Parameters Attribute Bar

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

4.5.1. Color Window of Machining Parameters

Currently, there are 8 colors in color window with their corresponding number, and each color is corresponding to different values of machining parameters, including feedrate, output, times, etc.

4.5.2. Modification Window of Machining Parameters

Machining objects of the same color are regarded as having the same values of machining parameters under two-dimension mode in NcEditor, so if the value of one of the machining parameters related to a certain color is changed, the value of this machining parameter of all the other objects with this color will be altered accordingly.

Choose a color from “Color window of machining parameters”, for example, select color “1”, as shown in Fig. 4-14:

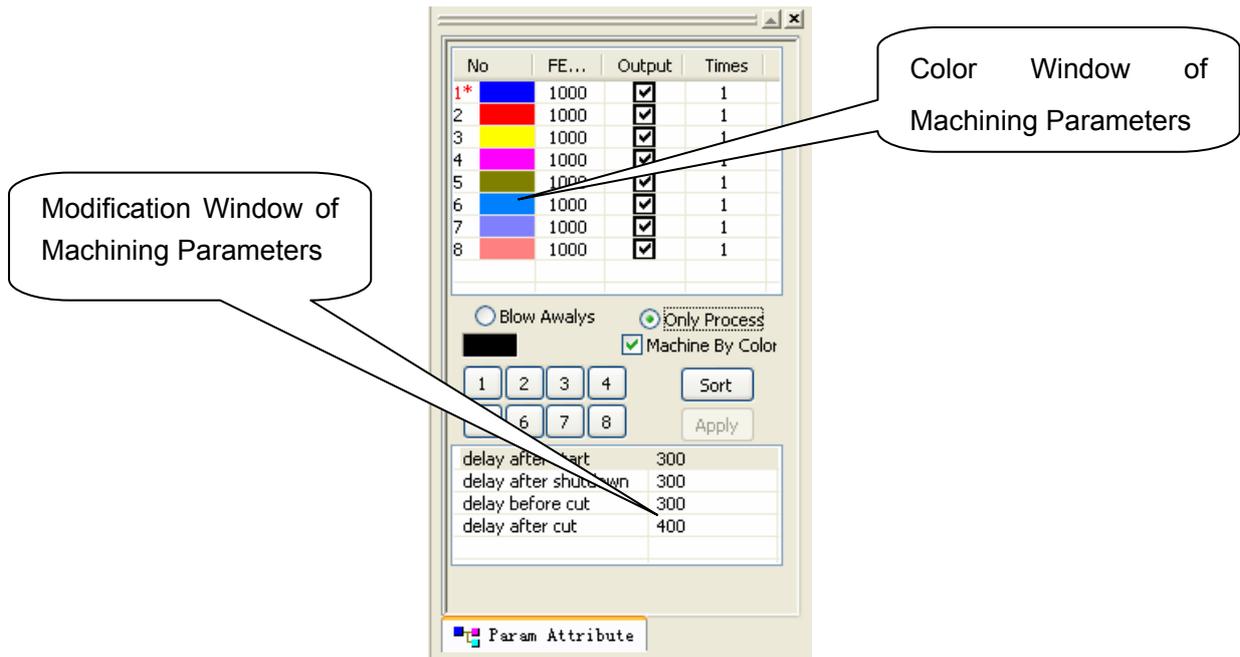


Fig. 4-14 Machining parameters attribute bar

When color 2 is clicked, after pressing “Apply”, the No. of color 2 will become red, and a mark “*” will be displayed on the top right corner of No. 2, i.e. this color is selected.

Double-clicking the color bar on the right side of the color No. will eject the parameter setting dialogue, in which clicking one of the “Value” columns can modify the corresponding parameter value, as shown in Fig. 4-15.

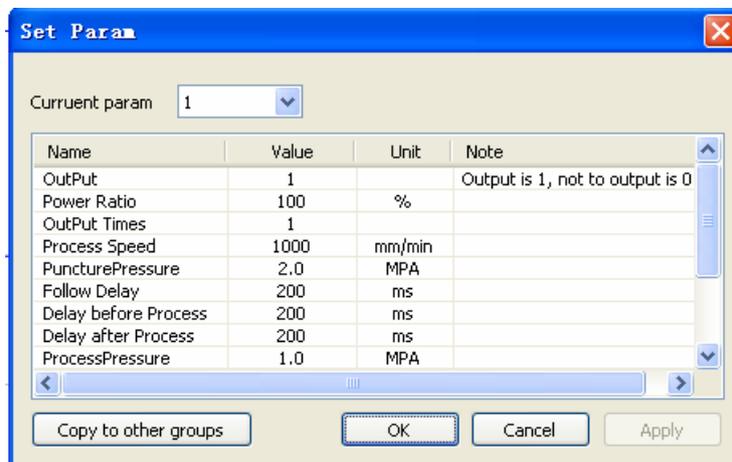


Fig. 4-15 Parameter setting

4.6. Figure Selection

To select a figure, you first need to ensure that there isn't other running command. Then press down the selection button, shown as .

You can select a single object in [Object editor space] by the following methods:

- Click on a frame of an object to select it;
- Select the required object in [Object list window];
- Hold down the left mouse button and move the mouse, there to be a rectangle box with dotted line in the space to ensure the selection of object profile.
- If you want to select multiple objects or object groups, the following methods can be used:
- Click on a frame of object and press the button Shift / Ctrl at the same time; this new object will be added into the selection.
- Select a group or the required objects in [Object list window];
- Holding down the left mouse button and moving the mouse, there will be a rectangle box with dotted line in the space. The objects total in the box will be selected, and there will be red dotted-line rectangle around the selected objects. Besides, you can hold down the left mouse button to drag the selected objects to the required place, which is also fit for a single selected object.

4.7. Menu Function

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

4.7.1. File Menu

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

◆ [New]

This menu is used to create a new NcEditor editing file (.nce).

◆ [Open]

The menu is used for opening the processing files from the hard disk, “NC” format & “G code” files available. The user can directly open a file and then edit it, and then save it after finishing editing, at this time, the system equals to an Editor. To back to the NcEditor interface, click the “New” item under [File] menu.

After this item is selected, [Open] dialogue will pop up, as below. After selecting a machining file, press “Open” to load it automatically.

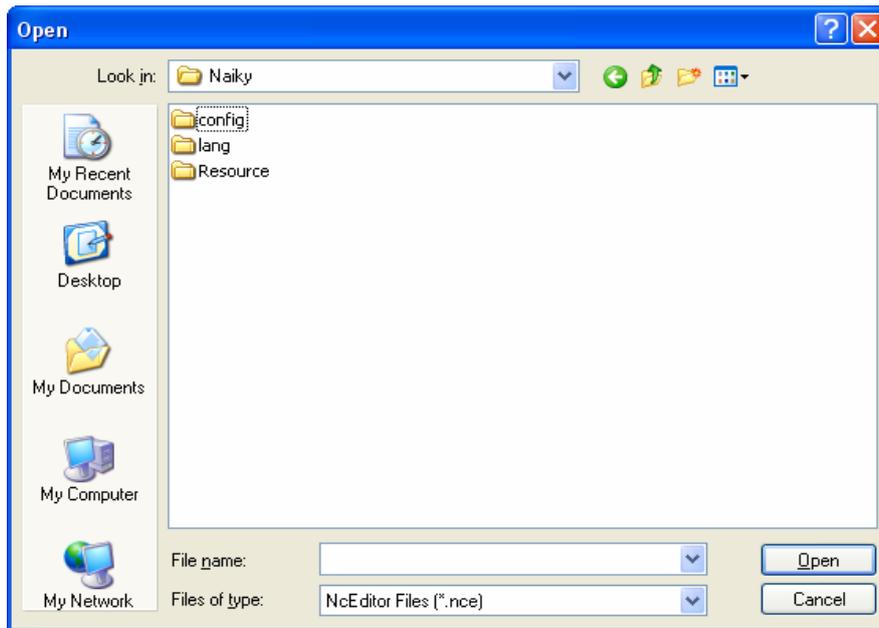


Fig. 4-16 Open dialogue

◆ [Open text file]

The main function of this menu item is to open all the file formats supported by system, for instance, G code files of .txt, .g, .nc formats, etc.. If an unsupported file is opened forcibly, the warning box “File error” will pop up. Just like [Open] menu item, the system equals to a file editor after a processing program is opened by [Open text file]. The user can edit the opened program file.

◆ [Import G / DXF/ ENG/ PLT code file]

Shortcut key: Ctrl+G/ D/ E/ P, the function of theses menu items are to open G code / DXF/ ENG/ PLT files. Method is as following (take G code as an example):

- Open or draw a machining file;
- Select [Import G code file] under [File] menu to eject [Open] dialogue, as below.

The method of importing DXF code file, ENG code file and PLT code file in NcEditor is the same as above.

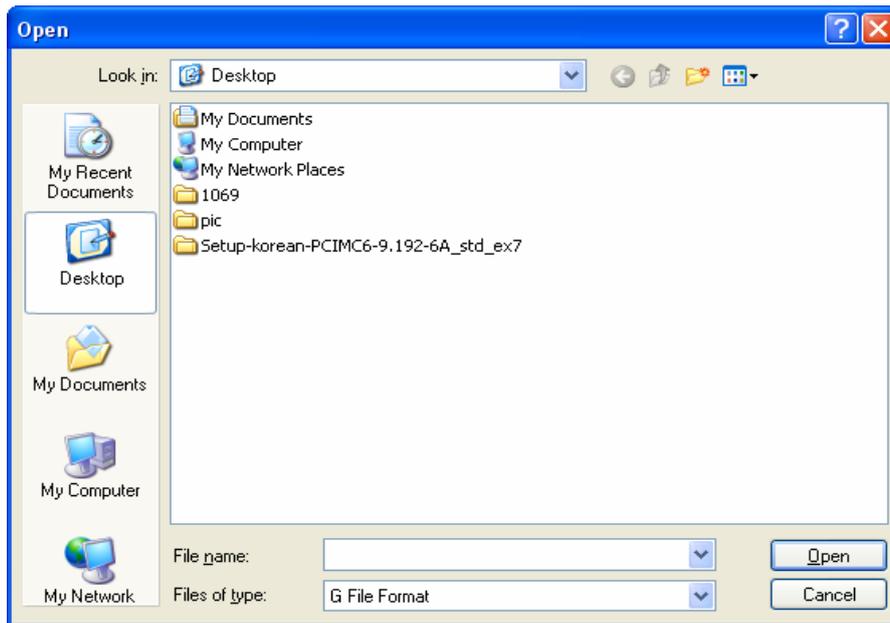


Fig. 4-17 [Import G Code File] dialogue

Currently, only NCE format files can be machined by a laser cutting machine; when other format file is loaded, it should be transformed to NCE format first, or, the system will prompt no machining file after pressing [Start] button.

◆ **[Save]**

The main function is to save the currently edited file.

◆ **[Save as]**

The function is using another file name to save the currently edited file.

◆ **[Save selected objects as]**

This function is used for a file with multiple machining objects; the user can save parts of objects as a single file for processing.

For instance: open a file, select some figures, as following.

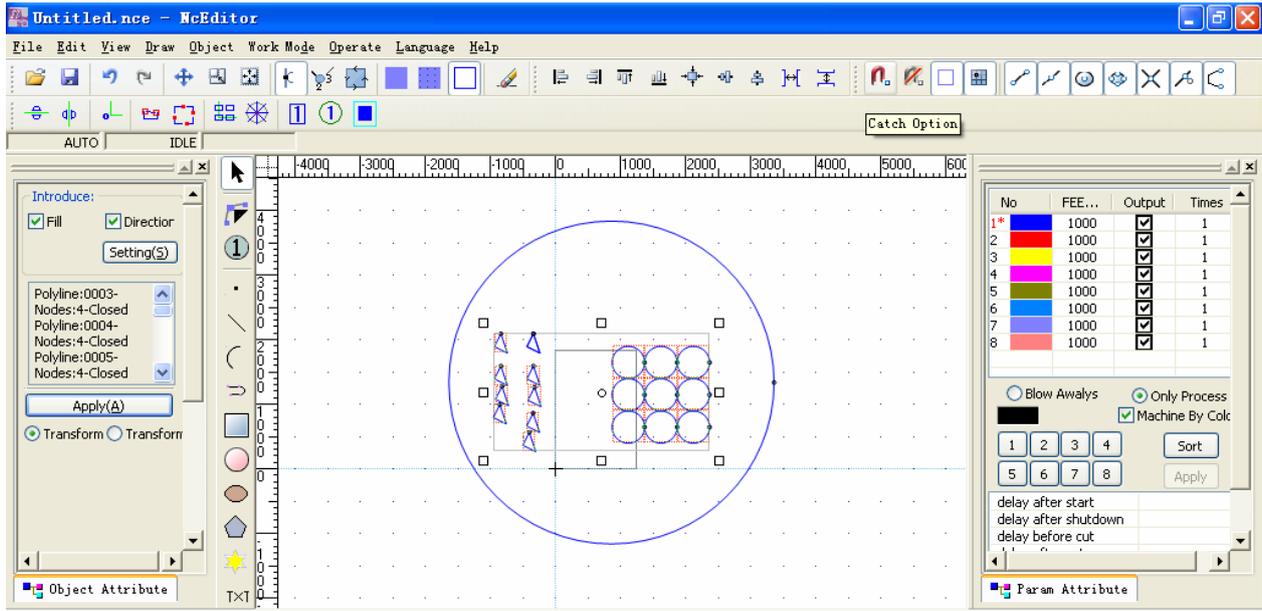


Fig. 4-18 Selection of part of objects

Then choose the option of [Save selected objects as] under [File] menu. After a dialog pop-up, input the file name and press “OK”; the selected objects are saved as a single file.

◆ [Recent loaded NCE files]

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

◆ [Recent loaded text files]

This function enables the user to view the recently loaded text files and open these files.

◆ [Exit]

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

4.7.2. Edit Menu

There are many submenus under Edit menu, which will be introduced in the following part.

◆ [Undo/ Redo (Ctrl +Z), (Ctrl + Y)]

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

The same function can also be fulfilled by [undo] button  and [redo] button  on toolbar.

◆ [Select all]

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

◆ **[Insert NCE/ G/ DXF/ ENG/ PLT code files]**

This function inserts machining objects of other file into the currently loaded file, taking DXF file as an example.

The method is as following:

- Open or draw a machining file;
- Select “Insert DXF code file” under [File] menu to eject the following dialogue.

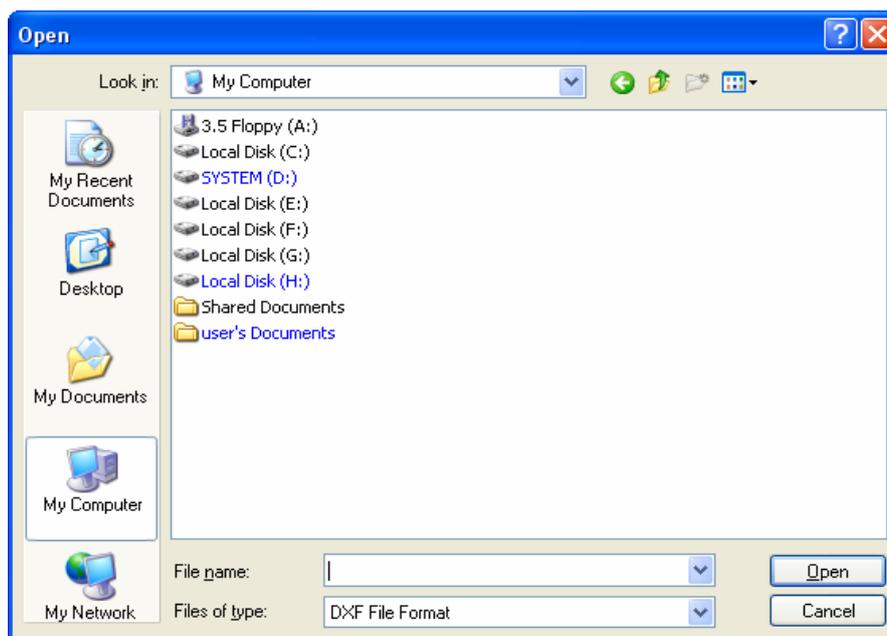


Fig. 4-19 Insert file dialogue

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

4.7.3. View Menu

Main options of View menu are: [Toolbar], [Status bar], [Object list], [Draw toolbar], [Machine control bar], [Parameters bar], [View translate], [View zoom], [Fit to window], [Catching option], etc.

◆ **[Toolbar]**

This function is used to display / hide the toolbar.

◆ **[Status Bar]**

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

◆ **[Object List]**

This function is used to display / hide the object list window and 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 machining parameters window.

◆ **[Machine Control Bar]**

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

◆ **[NcStudio]**

This function is used to display / hide the NcStudio interface.

◆ **[System Parameters]**

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

◆ **[Normal Parameters]**

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

◆ **[I / O Ports]**

This function is used to display / hide the hardware port dialogue, which is identical with the sub-function of [I/O Ports] in [DIAG] under [Auto] of NcStudio.

◆ **[Config]**

This item is used to select a laser cutting configuration, including “Plate config”, “Armor config”, “Circular config”, “Armor plate HUS2020” and “High armor plate HUS2020”.

◆ **[View Translate]**

The user can reposition a figure to clearly view other parts of the figure with this item. This command will not change the position or ratio of the figure in the coordinate, only the view. The user can translate the figure up and down, left and right.

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

➤ Method two: select the translation button  on toolbar.

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

◆ **[View Zoom]**

The user can zoom the track by menu + mouse or toolbar + mouse; the actual size of the figure won't change, only the view zooms 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, 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 the user wants to fit the view to window, this menu item can be used. This function can make the 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: select the button  on toolbar.

◆ [Catching Option]

[Catch Options] is used to better position to a feature point of objects during drawing, so a feature point can be easily caught when approached.

When [Catching Close] item is not selected, this item is valid.

Select [Catching option] under [View] menu to eject the following [catch option] dialogue.

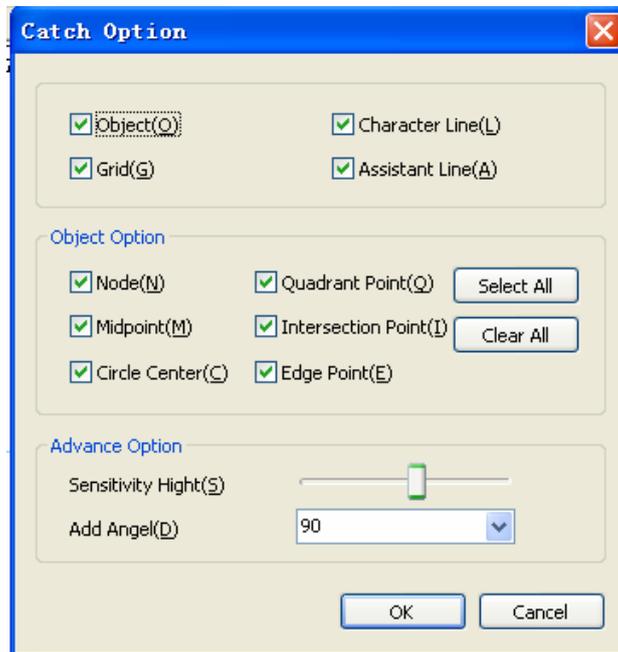


Fig. 4-20 [Catch Option] dialogue

In the above dialog, any choice box checked, its corresponding item is displayed in the [Object Editor Space], otherwise, hidden.

- The feature point of each figure:
 - 1) The character point of a spot: one point;
 - 2) The character point of a straight line: two endpoints and one midpoint;

- 3) The character point of a polyline: every node and every circle center;
 - 4) The character point of a rectangle: four vertexes
 - 5) The character point of a circle: four quadrant points and one circle center;
 - 6) The character point of an ellipse: four quadrant points and one circle center;
 - 7) The character point of an arc: start point, end point, midpoint and one circle center;
- The advantage of [Catching option]

The user can directly use cursor to position to an above-mentioned feature point precisely without knowing the concrete coordinates, which is convenient for the accurate connection of objects. This function is always used together with [Combine Object] function.

Note: in order to better introduce [Catching option], it is recommended to uncheck [Grid] option.

For instance: introduce another straight line from the midpoint of a straight-line.

- 1) Select the option of [Catching option] under [View] menu.
 - 2) Draw a straight-line in [object editor space]; a little pane will appear when the cursor is on the straight-line near the midpoint, which means a character point (midpoint of straight-line) is caught, so the midpoint is found accurately; click the pane to introduce another straight-line from the midpoint.
- [Catch grid]

The advantage of [Catch grid]:

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

The manipulation method:

Do not select the option of [Catching close] under [View] menu.

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

- [Sensitivity Height], [Add angle], [Character line] and [Assistant line]

[Sensitivity Height] is used to specify the reflection intensity of catching when the cursor is approaching the character point.

[Character line] is used to prompt the related position.

[Add angle] is to catch an object with a certain angle. For instance, choose “45”, in drawing a

straight-line, a red polar axis will be caught when the straight line to be drawn moves anticlockwise near 45 degree each time with the positive X axis as the start direction. In addition, the software prompts horizontal and vertical direction.

Note: close [Grid] function before using [Add Angle] function.

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

◆ **[View Introduce]**

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

The user can decide whether to add lead-in/out line or not as well as the type according to the cutting method. Refer to [setting lead-in/out line] function for detail.

◆ **[View Order]**

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

◆ **[View Direction]**

This function is used to display / hide machining direction.

◆ **[Drawing Mode]**

To facilitate drawing, this software set three views, i.e. ordinary, translucent and line rim mode.

1) **[Ordinary view mode]**

It is used frequently. When there are many layers, the overlapping parts are concealed, shown in Fig. 4-21, in order to distinguish the objects, different layers are in different colors, and in parameter attribute column, the layer color can be changed and machining by color can be selected.

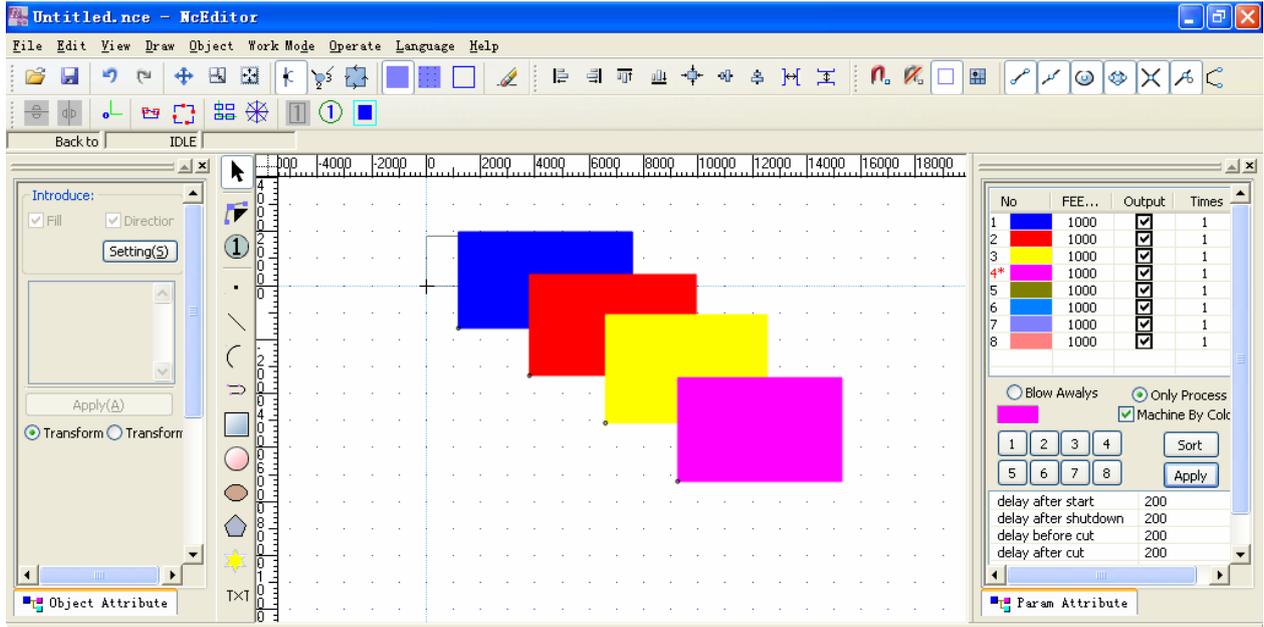


Fig. 4-21 Ordinary view mode

2) [Translucent view mode]

In order to make drawing more convenient, NcEditor can make layer translucent, that is the backward grid background or overlapping parts will be concealed translucently, as shown in Fig. 4-22.

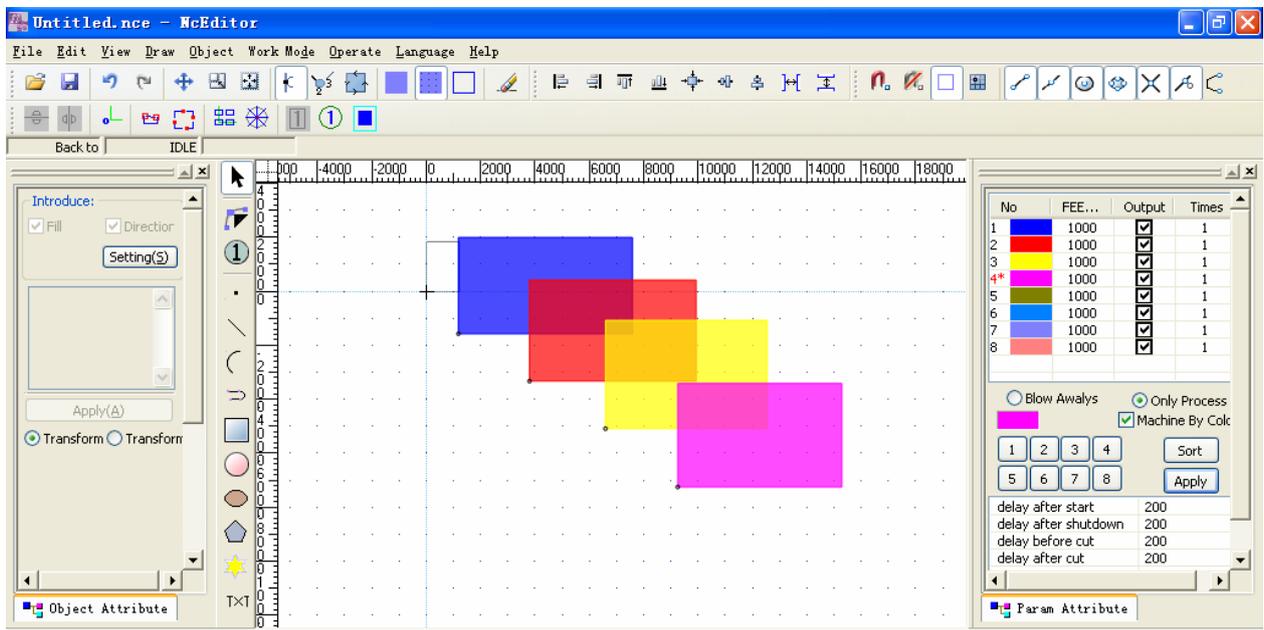


Fig. 4-22 Translucent view mode

3) [Line rim mode]

This is the default view mode; all the figures are shown in wireframe mode, that is the overlapping parts are revealed, as shown in Fig. 4-23.

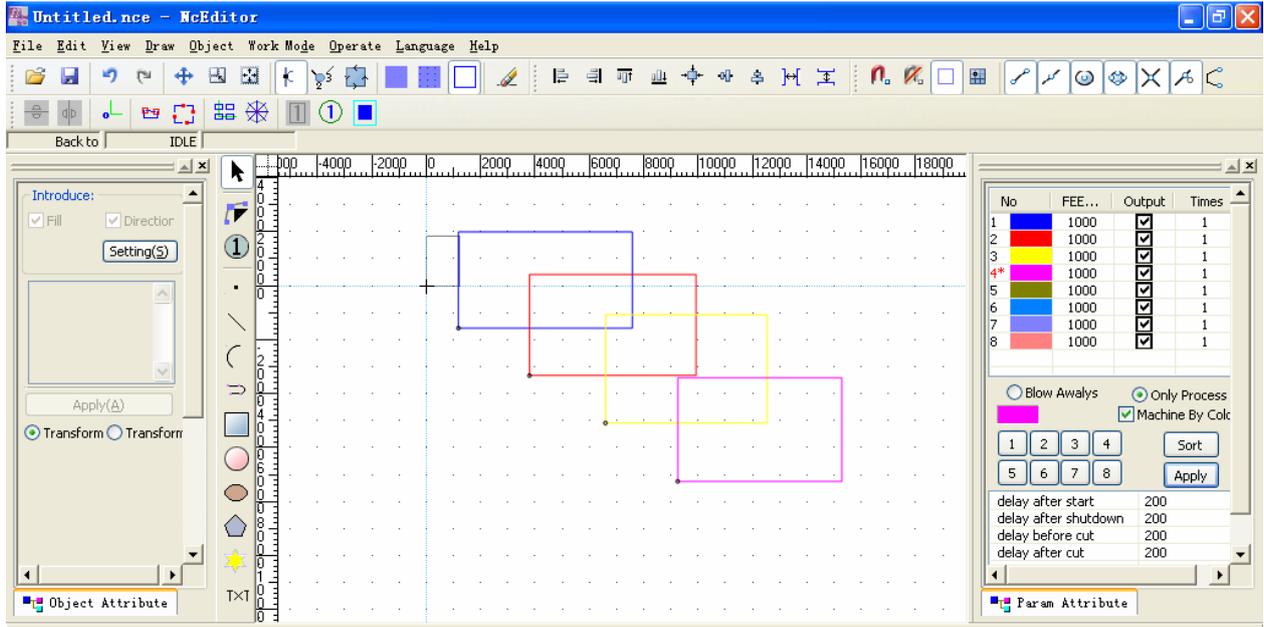


Fig. 4-23 Wireframe view mode

◆ [Set Machining Order]

This option can set machining sequence manually, with its counterpart [Manual Set Machining Order] under [Object] menu.

◆ [View Machining File Info]

The file information can be checked after machining. [View Machining File Info] displays the information of file name, Gxx time, G00 time, and Gxx length, G00 length, unclosed figure, closed figure, and total figure.

After finishing machining, click the option [View Machining File Info] to eject the dialogue, as shown in Fig. 4-24.

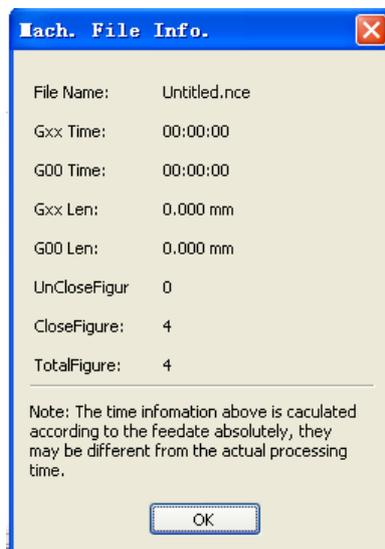


Fig. 4-24 Machining file information

There will be information shown only after machining.

◆ **[Set Default Length Unit]**

There are two kinds of length unit in NcEditor, metric and imperial. This is for the different measurement units of different nations; the user can set measurement unit according to his needs.

Select [Set default length unit] option under [View] to eject the following dialogue.

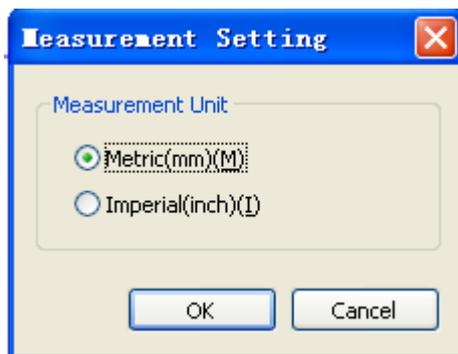


Fig. 4-25 Setting length unit dialogue

4.7.4. Draw Menu

[Draw] menu is used to draw ordinary figures, including points, straight lines, polylines, rectangles, circles, ellipses, arcs, and texts, etc. All the menu items have their counterparts on the drawing toolbar, as shown in Fig. 4-26.

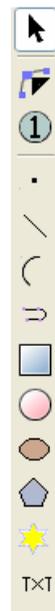


Fig. 4-26 Drawing toolbar

Note: the figure drawn below can be precisely positioned by object attribute window, and precise positioning and drawing is not finished until [Apply] is clicked or Enter is pressed.

◆ [Draw Point (D)]

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

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

If the user needs to draw a point more precisely, he can input the coordinates in [object attribute window], and then press [Enter] or click [Apply].

◆ [Draw Line (L)]

If the user wants to draw a straight line, he can select the option [Draw line] under [Draw] or single-click icon .

Two points can determine one straight line. Draw two points in [object editor space] as the beginning and end points of a straight line by left mouse key.

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

◆ [Draw Arc (A)]

If the user wants to draw an arc, he can select [Draw Arc] under [Draw] or single-click icon .

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

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

Note: 1 rad. = $180/\pi$ degree

◆ [Draw Polyline (P)]

A polyline refers to a curve connected end to end by different straight lines and arcs, which enables the user to reduce operation times of segments and improve the velocity of modeling via finishing manipulation of multiple arcs and segments once.

If a polyline is needed, the user can select [Draw Polyline] under [Draw] or single-click icon .

Shortcut key: "A"→ arc;" L"→ line; "C"→ close the figure; "F"→ finish drawing; "Esc"→ cancel drawing.

Note:

In drawing a polyline, the input state of system should be English.

◆ [Draw Rectangle (R)]

To draw a rectangle, the user can select [Draw Rectangle] under [Draw] or single-click icon .

NcEditor determines the position and shape of a rectangle by width, height, and coordinates of the left bottom vertex.

After selecting a rectangle, the user can accurately redraw and position it by inputting the left bottom vertex coordinates, height and width in [Object attribute window].

◆ [Draw Circle (C)]

To draw a circle, the user can select [Draw Circle] under [Draw] or single-click icon .

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

After selecting a circle, the user can accurately redraw and position it by inputting circle center coordinates and radius in [Object attribute window].

◆ [Draw Ellipse (E)]

If an ellipse is needed, the user can select [Draw Ellipse] under [Draw] or single-click icon .

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

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

◆ [Draw Polygon (G)]

If a polygon is needed, the user can select [Draw Polygon] under [Draw] or single-click icon .

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

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

◆ [Draw Star (S)]

If the user wants to draw a star, he can select [Draw Star] under [Draw] or single-click icon .

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

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

◆ [Draw Text]

If the user wants to directly write a paragraph, he can select [Draw Text] under [Draw] or

single-click icon

Select [Draw text] item, and then single-click in [object editor space], "TEXT" characters appearing, as shown in Fig. 4-27.

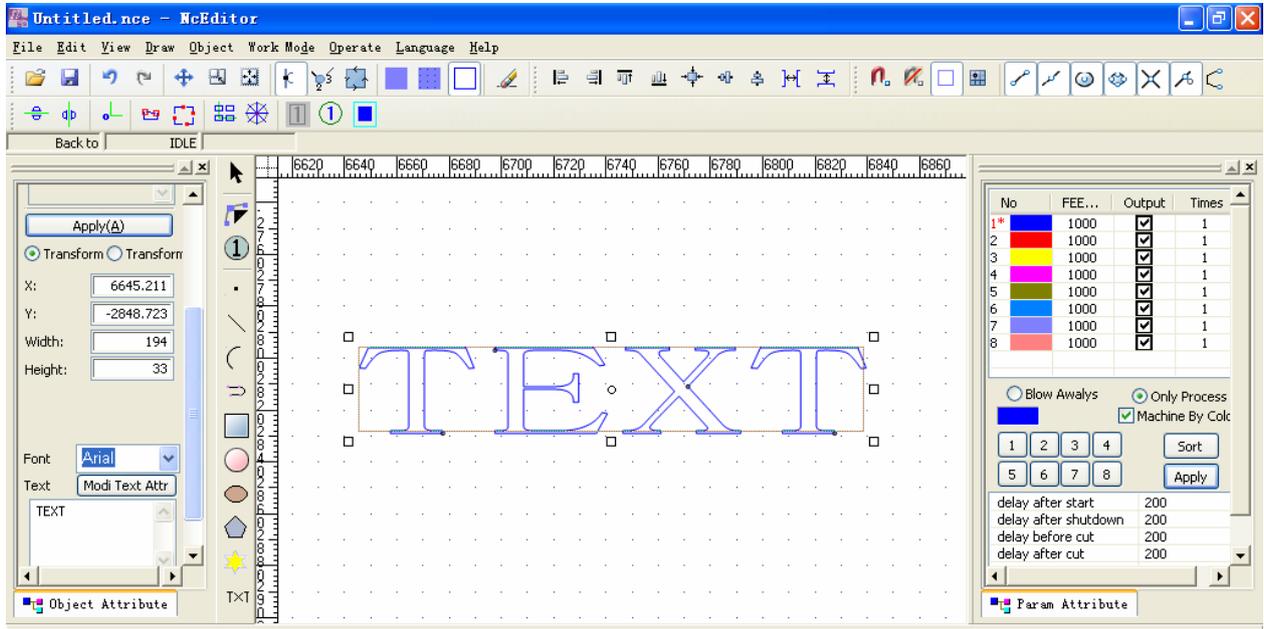


Fig. 4-27 Editing text

As shown above, clicking "TEXT" in "editor space" will make the character modification box appear in [object attribute window], shown in Fig. 4-28.

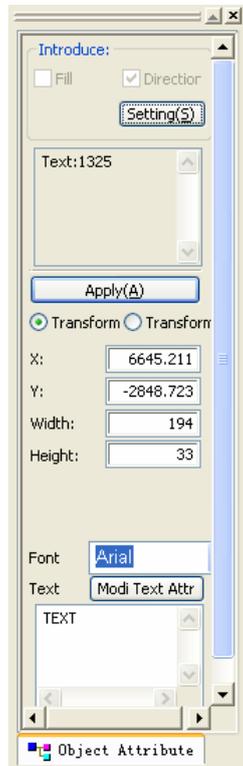


Fig. 4-28 Text editing zone

Change the word “TEXT” to the necessary words in text zone, then set font and size, and then click “Apply”.

Besides, in comparison with [Drawing Toolbar], [Draw] menu adds two extra functions, i.e. [Measure Distance] and [Set Origin].

◆ **[Measure Distance]**

This item is used for measuring a certain distance, mainly adopted in dispenser software, barely in laser cutting software.

◆ **[Set Origin]**

This function is used to set workpiece origin. After selecting this menu item, a dialogue will pop up, as shown in Fig. 4-29.

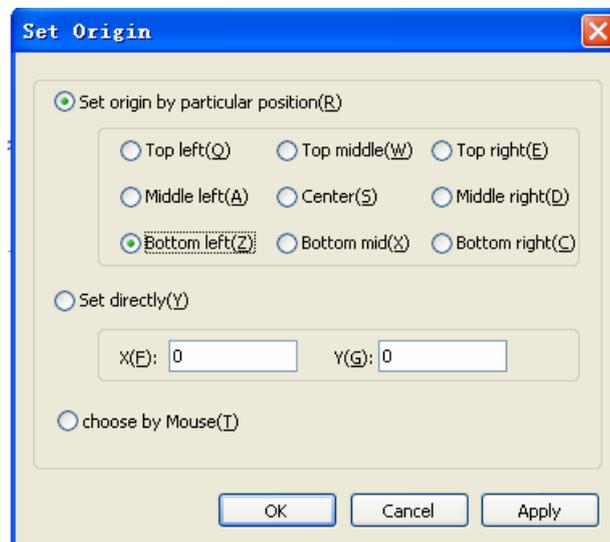


Fig. 4-29 [Set Origin]

Workpiece coordinate origin can be set by particular position, or directly inputting the coordinates, or by mouse.

4.7.5. Object Menu

Right-click menu is also the [Object] menu, which includes [Delete Object], [Combine Object], [Combine Object Tolerance Setting], and so on.

◆ **[Delete Object]**

Objects must be selected before deleted.

Following three methods can be used for deleting objects:

- Right-click on the selected object and select [Delete object];
- Select [Delete object] under [Object];

- Use the button [Delete] on the keyboard.

To delete all objects, press Ctrl + A to select all objects, and then press Delete to delete them.

◆ [Combine Object]

Note: the points of unclosed objects may not connect together precisely because of misoperation or other reasons. When the objects are needed integrated operation, the user can use [combine object] to consolidate several unclosed objects (straight line, arc and unclosed polyline) with nearby endpoints (<5) to a single path object (polyline or path). It is suggested to use [catching option] before using [combine object].

For instance, combine two straight lines by the function of [combine object].

First, draw the first line. Then open [catching option] to precisely position to an endpoint of the first line before drawing the second line. And then select option [combine object] under [Object] or under right-click menu. See Fig. 4-30.

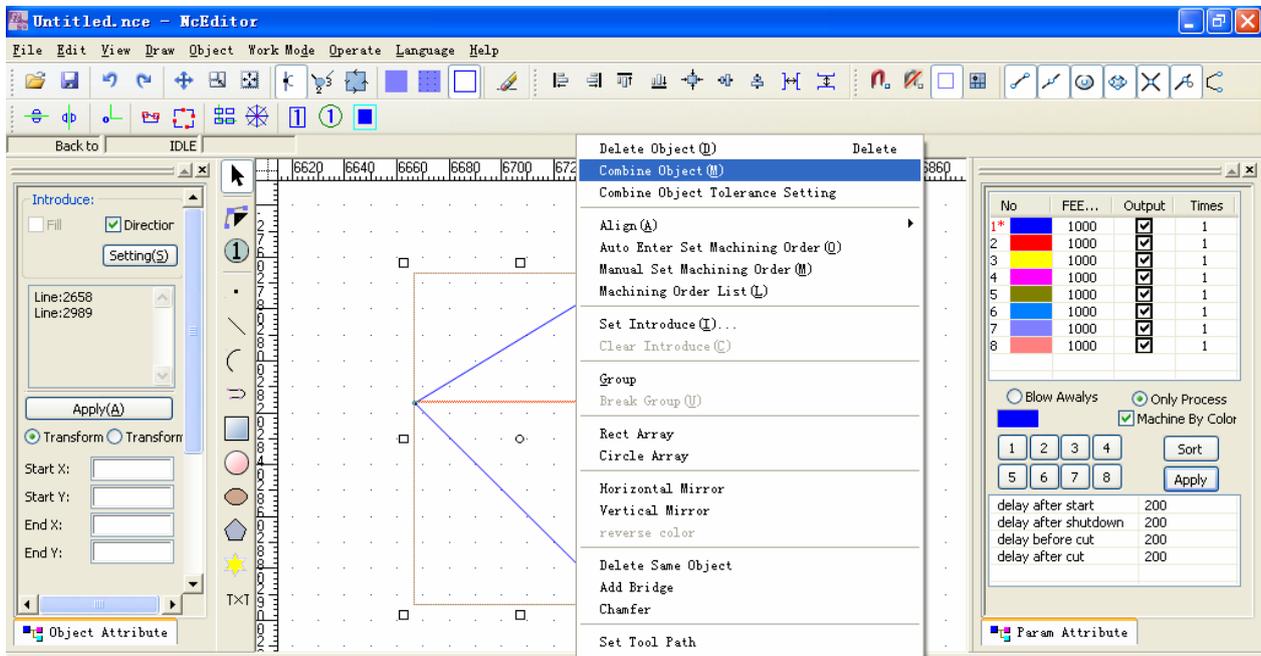


Fig. 4-30 Manipulation of combination of two straight lines

The combination output of two straight lines is as Fig. 4-31, and the two objects are combined to one single object (polyline).

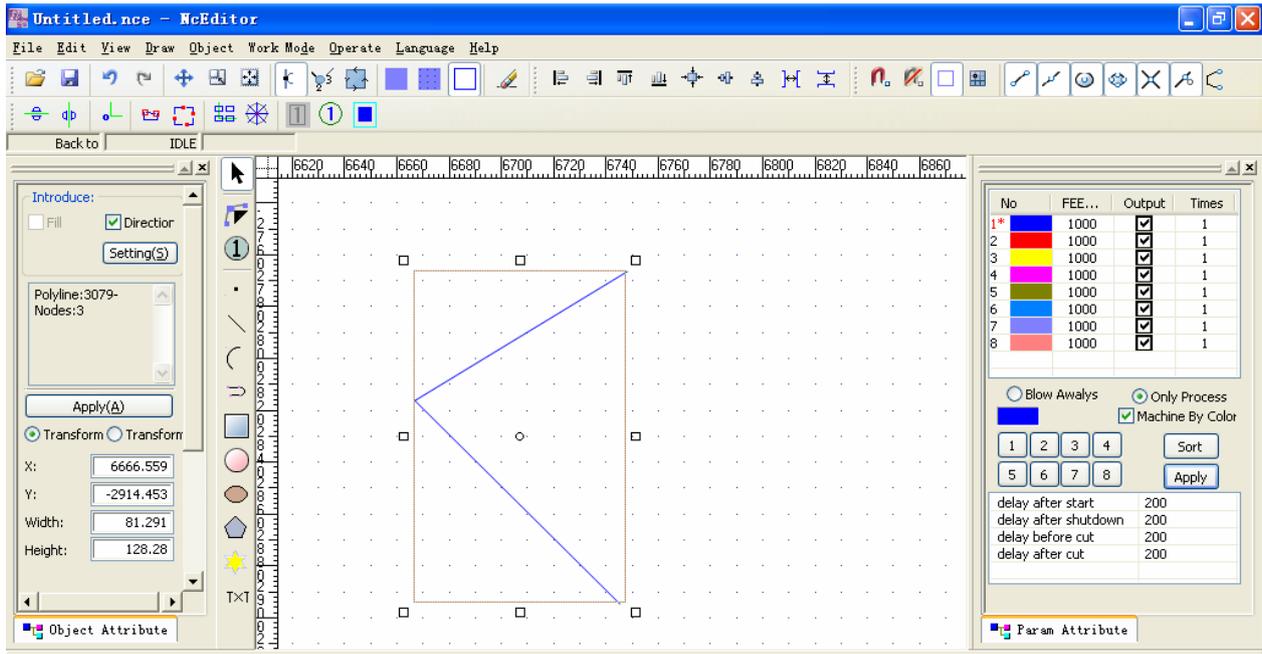


Fig. 4-31 The combination output of two straight lines

Referring to combination among straight lines, arcs and unclosed polylines, the user can better catch their endpoints by [Catching options] so as to improve the efficiency of [Combine object].

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

◆ [Combine Object Tolerance Setting]

In combination, the user needs to combine objects following a certain criterion, so combine tolerance is offered. When the distance between selected objects is within combine tolerance, objects will be combined.

The following dialogue will pop up in selecting [combine object tolerance setting]. Input a number between 0.001 ~ 5 into the input dialogue.



Fig. 4-32 [Combine object tolerance setting] dialogue

◆ [Align]

When two or above two objects are selected in [Object editor space], the [Align] function is enabled; this function can align all the selected objects in two-dimensional plane.

Note: all the following alignment functions are on the basis of top, bottom, left and right frame and center point of the rectangle of object selection.

See Fig. 4-43, all the following alignment functions are exemplified by the two below objects.

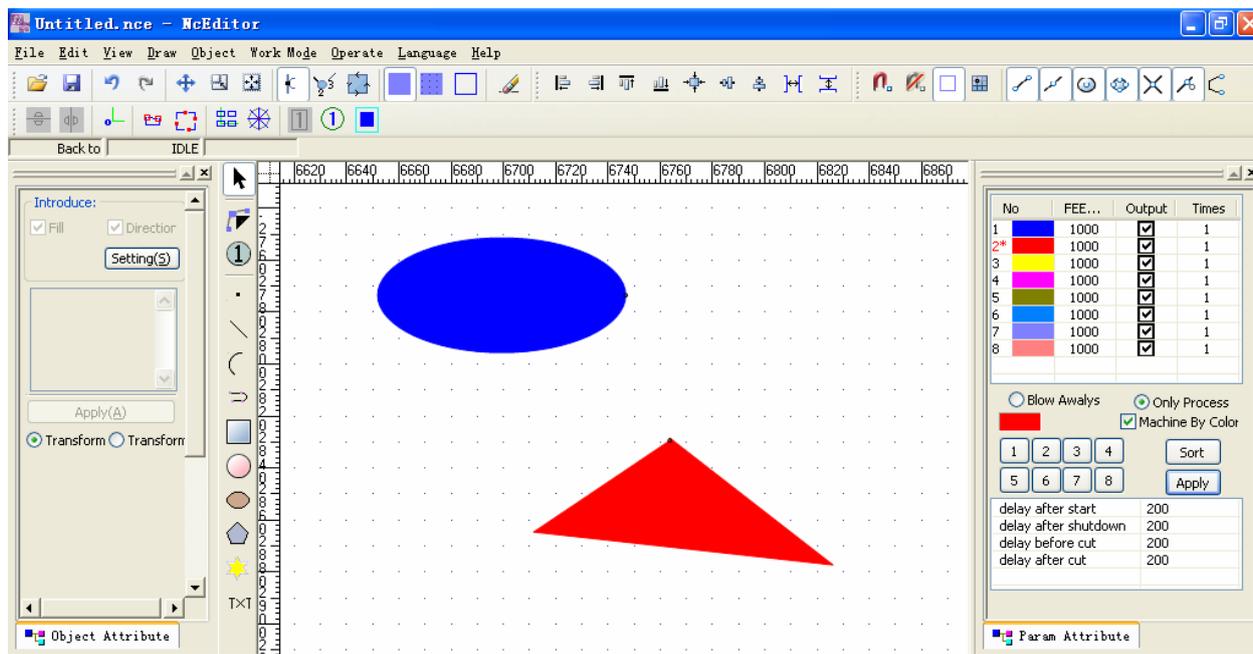


Fig. 4-33 Two objects used for alignment

◆ [Align Left]

The left edge of rectangle of object selection is regarded as the benchmark for alignment.

After selecting the objects, there are three methods to fulfill [Align left] function:

- Select option [Align left] under right-click menu;
- Select option [Align left] under [Object] menu;
- Click the [Align left] button  on the toolbar.

After [Align left] is executed, the view is as following:

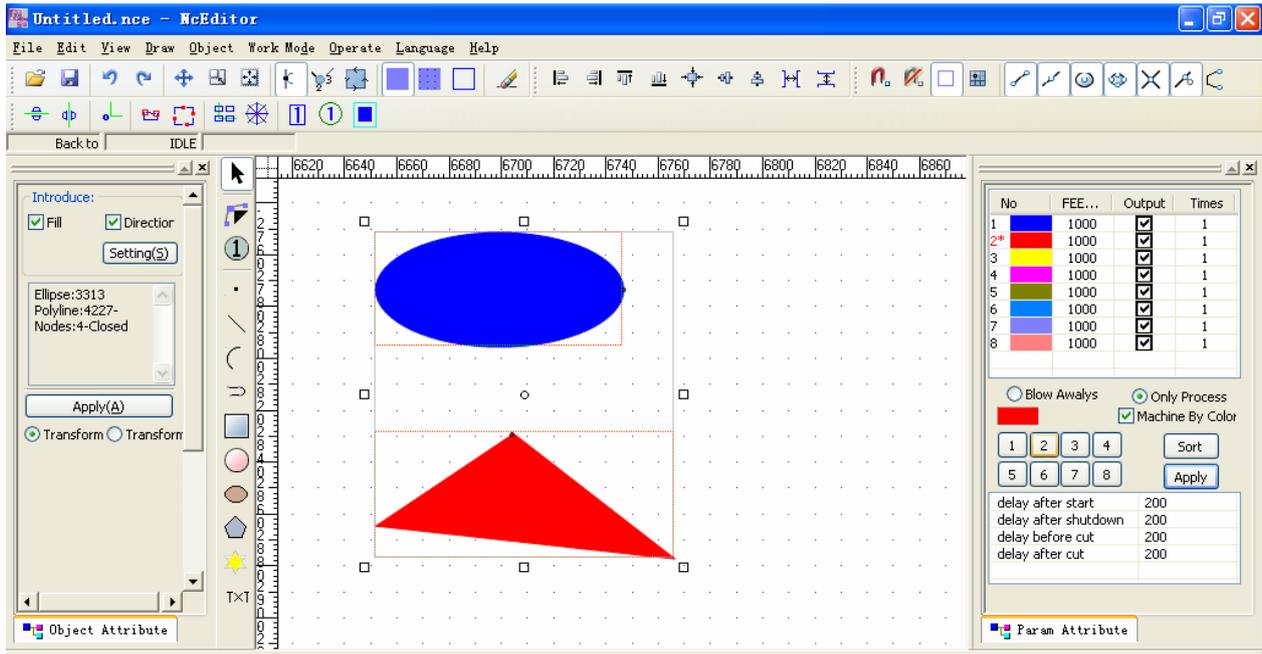


Fig. 4-34 [Align left] operation

◆ [Align Right]

The right edge of rectangle of object selection is regarded as the benchmark for alignment.

◆ [Align Top]

The top edge of rectangle of object selection is regarded as the datum for alignment.

◆ [Align Bottom]

The bottom edge of rectangle of object selection is regarded as the datum for alignment.

◆ [Align Center Point]

The center point of rectangle of object selection is regarded as the datum for alignment.

After the objects are selected, there are three methods to fulfill [Align Center Point] function:

- Select option [Align Center Point] under right-click menu;
- Select option [Align Center Point] under [Object] menu;
- Click the [Align Center Point] button  on the toolbar.

After [Align Center Point] is executed, the view is as following.

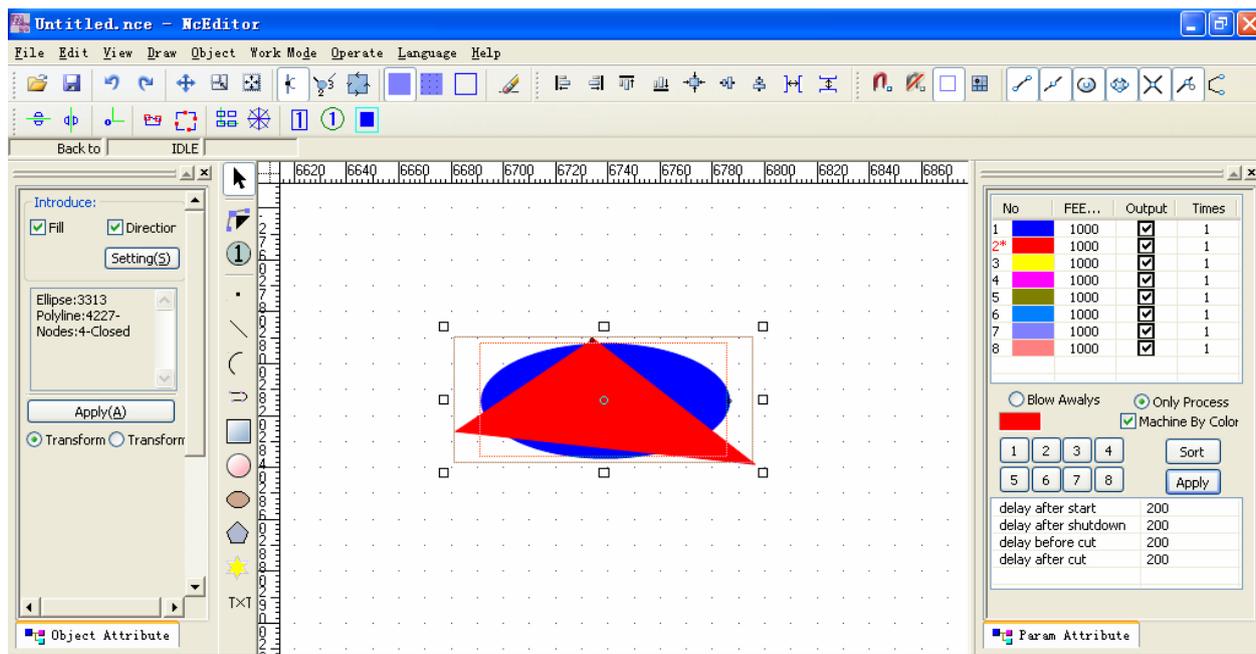


Fig. 4-35 [Align center point] operation

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

◆ [Align Vertical Line]

The X coordinate of center point of rectangle of object selection is regarded as the alignment reference for those of all the selected objects.

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

- Select option [Align vertical line] under right-click menu;
- Select option [Align vertical line] under [Object] menu;
- Press the [Align vertical line] button  on the toolbar.

After [Align vertical line] is executed, the view is as following:

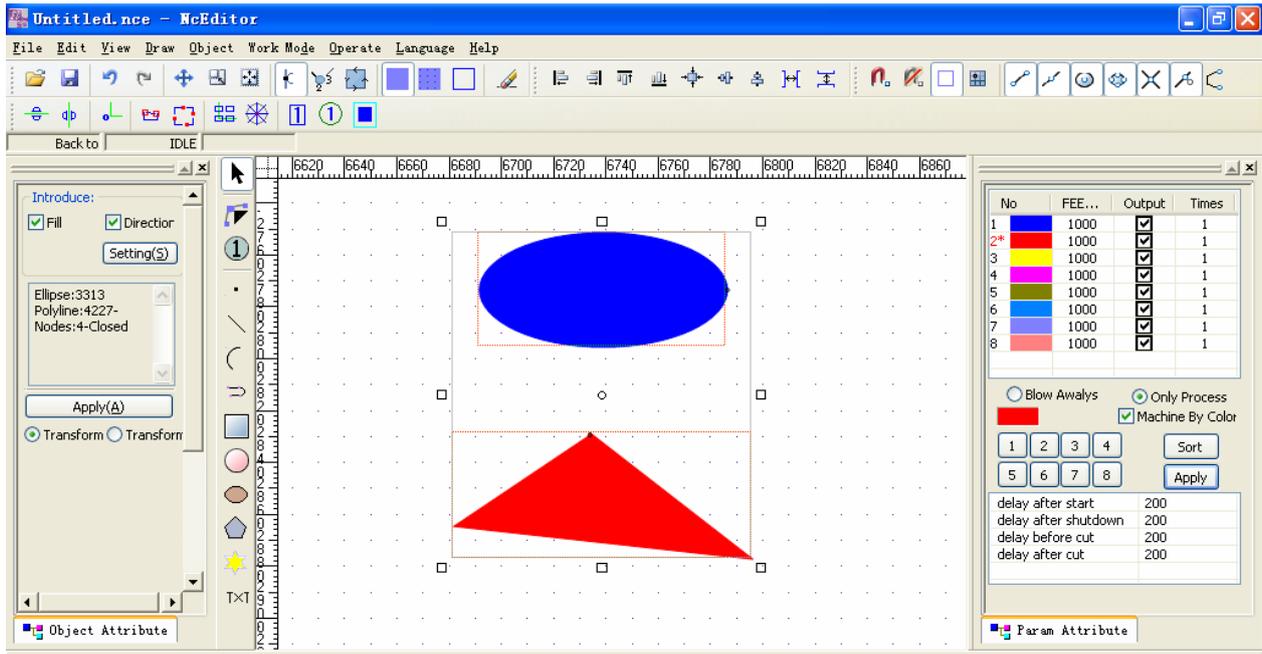


Fig. 4-36 [Align vertical line] operation

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

◆ [Align Level Line]

The Y-coordinate of center point of rectangle of object selection is regarded as the datum for those of all the selected objects.

◆ [Align Level Dispersion]

There are two alignment criteria for this method:

- Center point distance standard;
- Graphic distance standard;

In [center point distance standard], the horizontal distances between center points of objects are equal, and the positions of two objects next to the horizontal edge are fixed. In [Graphic distance standard], the horizontal blank spaces between each object are equal.

The selected objects must be three or above three. After the selection of objects, there are three methods to fulfill [Align level dispersion].

- Select option [Align level dispersion] under right-click menu;
- Select option [Align level dispersion] under [Object] menu;
- Press the icon  on toolbar;

After the [Align level dispersion] is activated via one of the above methods, the following dialogue will pop up.

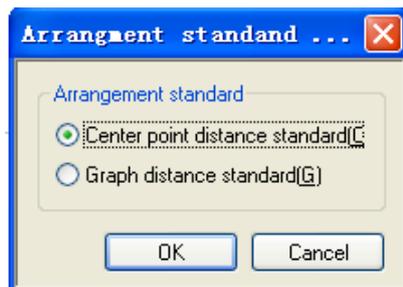


Fig. 4-37 [Arrangement standard setting]

Select [center point distance standard] and execute [Align level dispersion]. See Fig. 4-38 and Fig. 4-39.

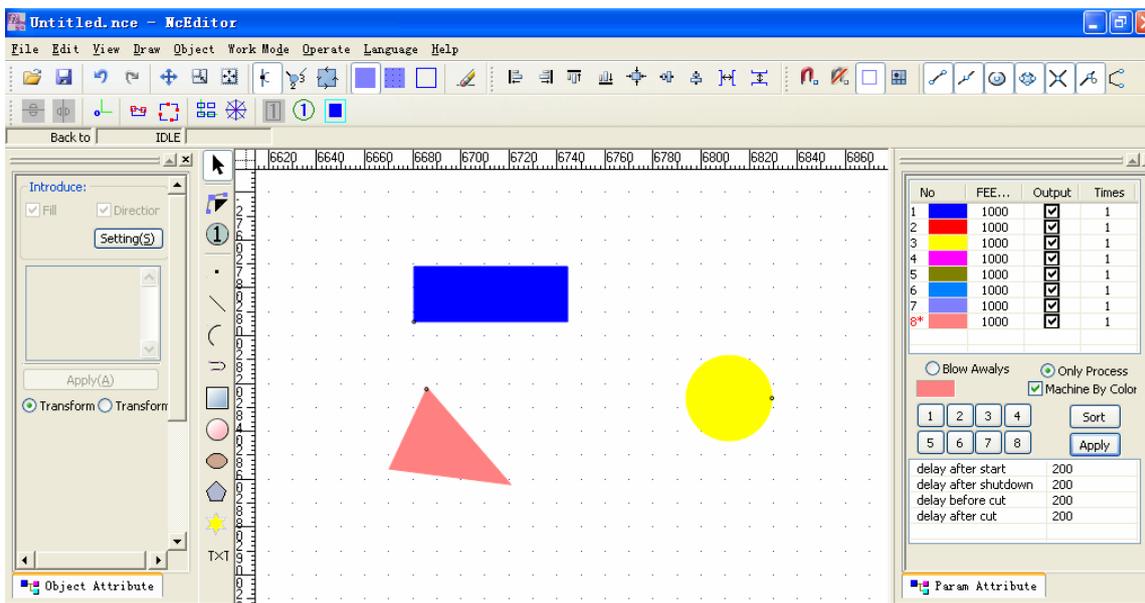


Fig. 4-38 Before [Align level dispersion]

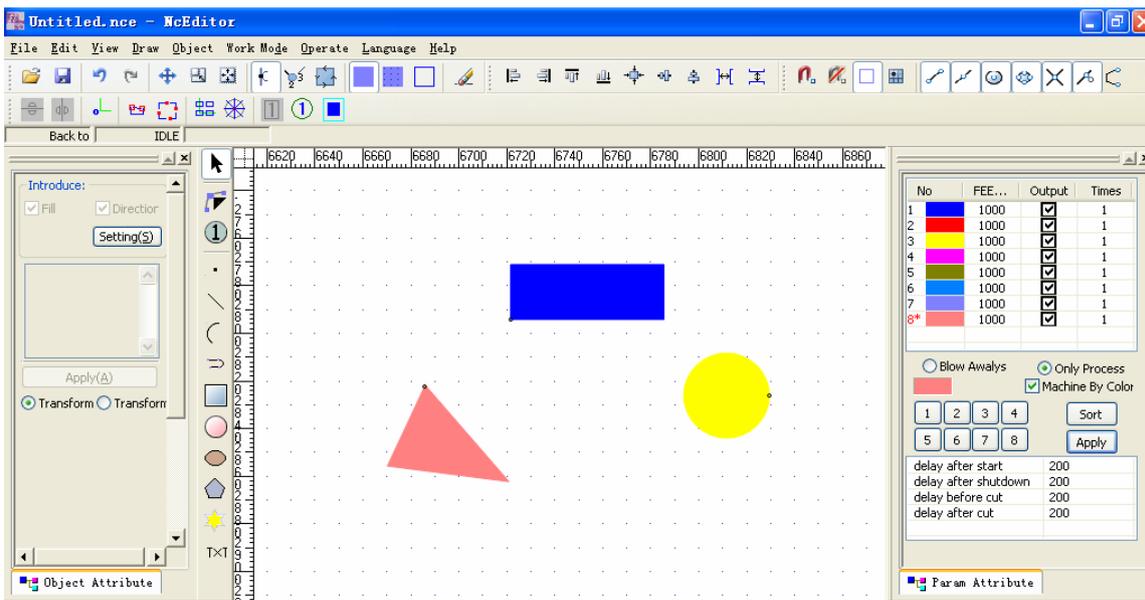


Fig. 4-39 After [Align level dispersion]

Note:

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

◆ [Align Vertical Dispersion]

Just like [Align level dispersion], there are two criteria for [Align vertical dispersion]:

- Center point distance standard;
- Graphic distance standard;

In [center point distance], the vertical distances between center points of objects are equal, and the positions of objects next to the vertical edge are fixed. In [Graphic distance standard], the vertical blank spaces between each object are equal.

◆ [Auto Enter Set Machining Order]

In order to save time and improve efficiency in machining, NcEditor holds the function of [Auto enter set machining order] in two-dimensional mode of NcEditor, so that the system can set machining order following the best path algorithm.

Manipulation method:

- Draw several different types of objects.
- Select [View order] option under [View] or click the icon  on toolbar to display the machining order of the objects, see Fig. 4-40.

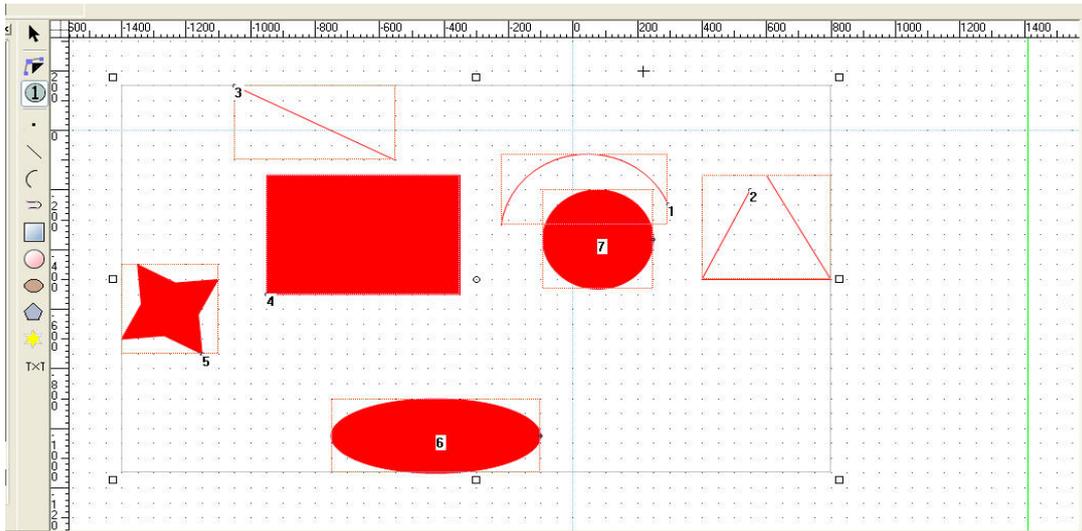


Fig. 4-40 Displaying the machining order of the objects

- Click the option [Auto enter set machining order] under [Object] or under right-click menu or click the button  on toolbar, the system automatically setting machining order following the best path algorithm, see Fig. 4-41.

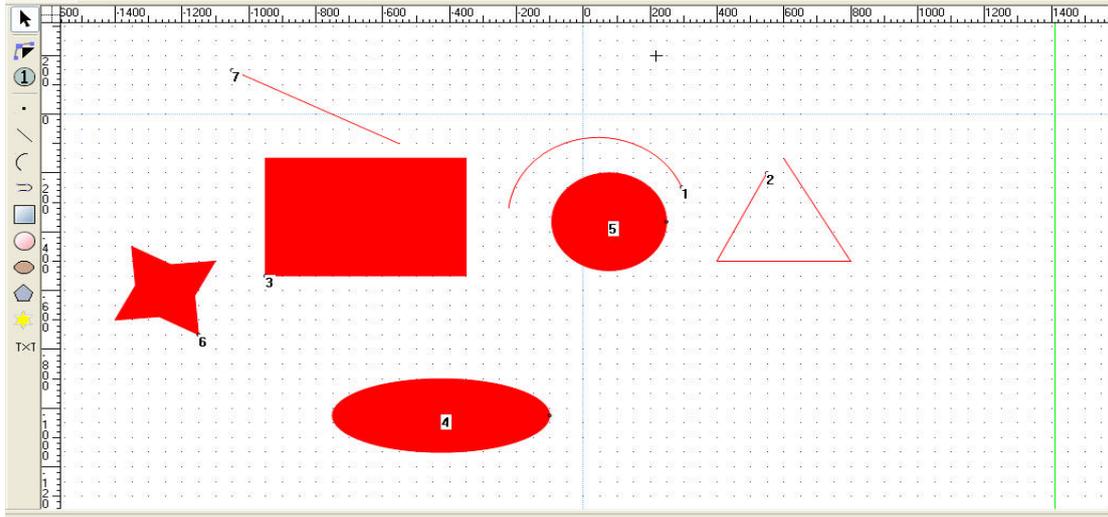


Fig. 4-41 Auto setting machining order

◆ [Manual Set Machining Order]

This function is to make it more convenient for the user to set a special machining order for a certain object.

After executing [Auto enter set machining order], to change the machining order of a certain object, the method is as following:

Select the object → choose [Manual set machining order] under [object] menu or under right-click menu → input the required sequence number into the following input dialog → press [Ok].

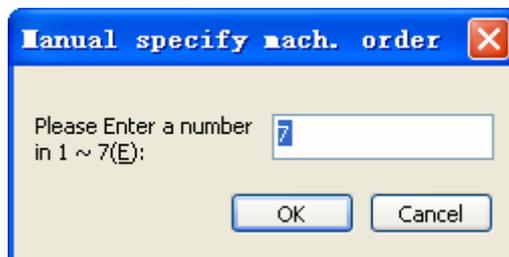


Fig. 4-42 [Manual set machining order] dialogue

Please specify a number between 1 and 7, "7" is the total number of objects in [Object editor space], which is variable.

◆ [Mouse Set Machining Order]

The icon on drawing toolbar can be used to change the machining sequence of objects.

For example, after clicking the icon , clicking on object 6 by mouse will change its machining sequence to 1; then clicking on object 5 will changing its sequence to 2, the machining sequence of the rest objects reordered according to the original ascending sequence.

To maintain the sequence of several previous objects and change the sequence of the later

objects, for example, to retain the sequence of No.1-4 objects unchanged and change the sequence of No.5-8 objects, firstly click the icon  , then press Ctrl key and click No.4 object simultaneously to keep the sequence of previous 4 objects unchanged, and then click the object to be set as No.5, No.6 and No.7 sequentially.

◆ [Machining Order List]

This function can also be used for sequence modification. The method is:

Click the item [Machining Order List] under [Object] menu →choose several objects in the pop-up dialogue→ double-click the other objects to change the sequence.

◆ [Set Introduce]

Refer to 4.3.2 for detailed introduction.

◆ [Clear Introduce]

If the set introduce is not appropriate, click this menu item to delete it after selecting the corresponding object.

◆ [Group]

If multiple objects, multiple groups of objects or objects and groups are needed integral operation, they should be set as a group firstly.

See Fig. 4-43, please select the required objects or object groups, and then choose the option [Group] under [Object] menu or under right-click menu.

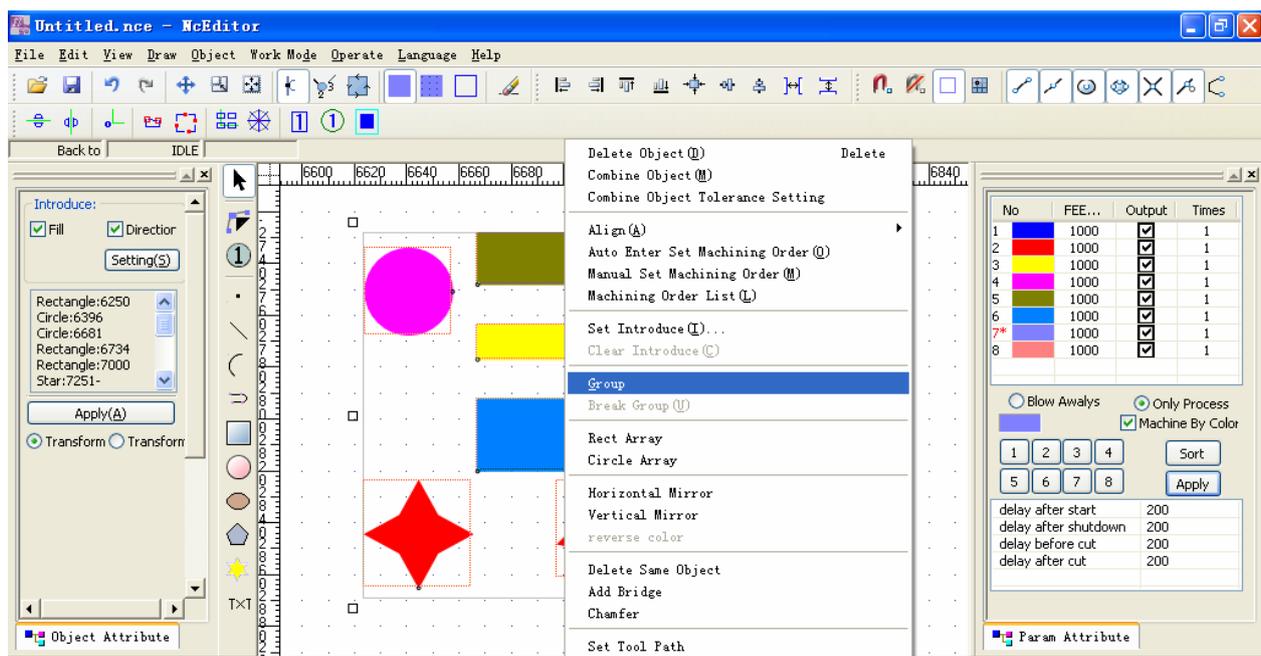


Fig. 4-43 Setting selected objects as a group

As Fig. 4-44, all the objects are set as a new group.

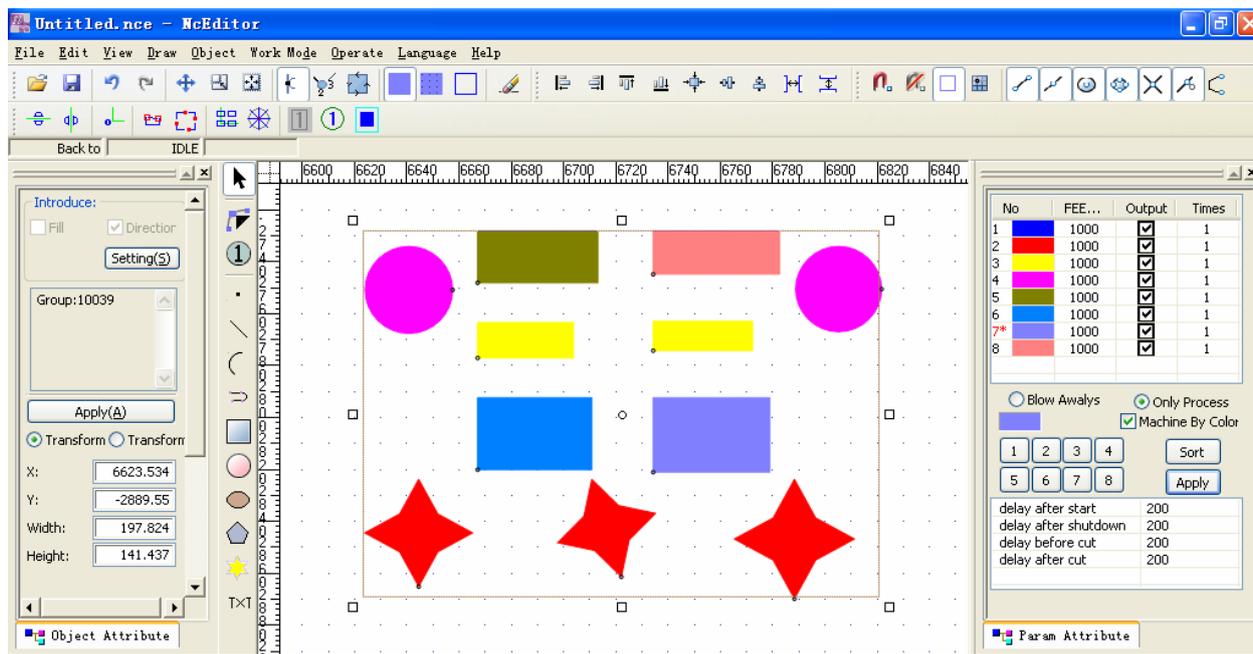


Fig. 4-44 Objects are in the same group

Now the new group can be edited as a whole.

◆ [Break Group]

If only a part of objects in a group are needed editing, the group should be dismissed first.

Select the group → choose [Break group] under [Object] or under right-click menu → the objects in the group back to the previous group.

◆ [Rect Array]

Select one or multiple objects or groups (objects in different groups should be set as a group first) → choose option [Rect array] under [Object] menu → input the array rows, columns, line space (offset), column pitch (offset) into the pop-up dialogue → press "OK".

For instance, draw a picture as following → set them as a selected group.

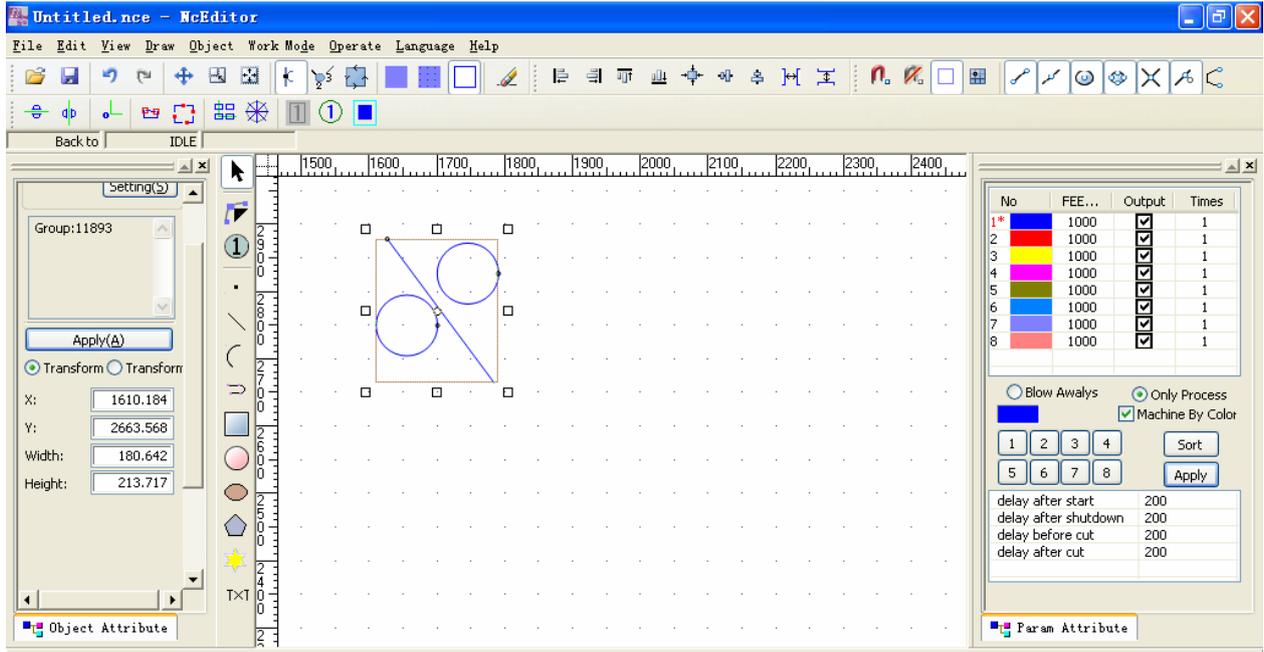


Fig. 4-45 Example figure

Select [Rect array], input the data and press “Ok”, as following.

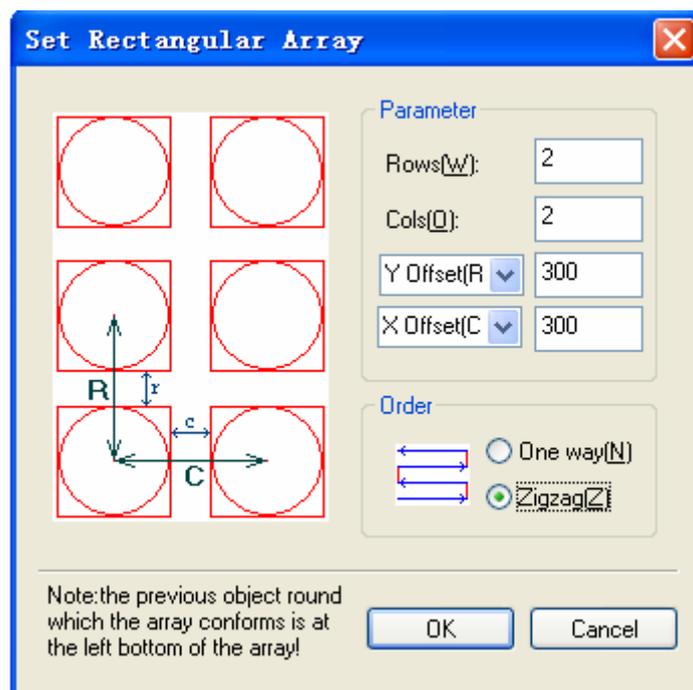


Fig. 4-46 [Rectangular array] dialogue

The output after setting rectangular array is as following in Fig. 4-47.

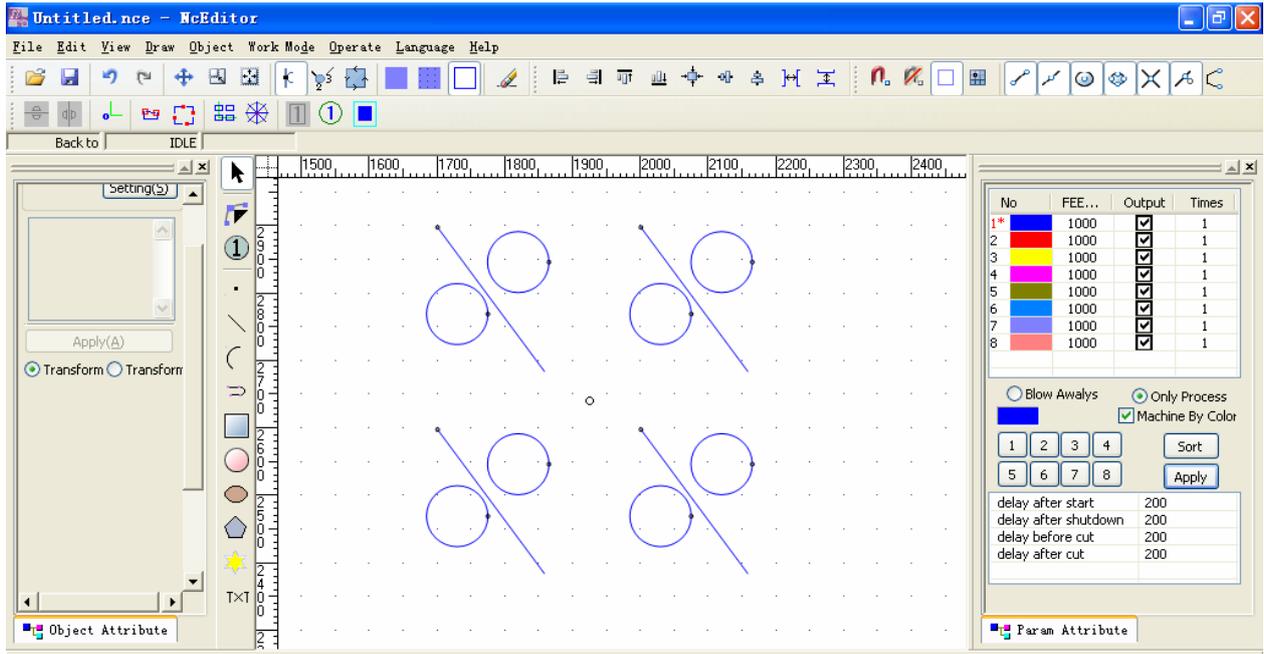


Fig. 4-47 The output after setting rectangular array

◆ [Circle Array]

Select one or multiple objects or groups (objects in different groups should be set as a group firstly) → choose [Circle array] under [Object] menu or click the button  on toolbar → input the radius, original angle and units on circle into the pop-up input frame → press “OK”.

For instance, select the group in Fig. 4-45 → choose [Circle array] → input the data as following and press “OK”

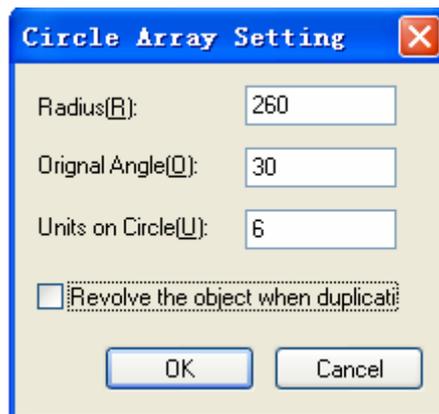


Fig. 4-48 [Circle array setting] dialogue

The output after setting circle array is as following in Fig. 4-49.

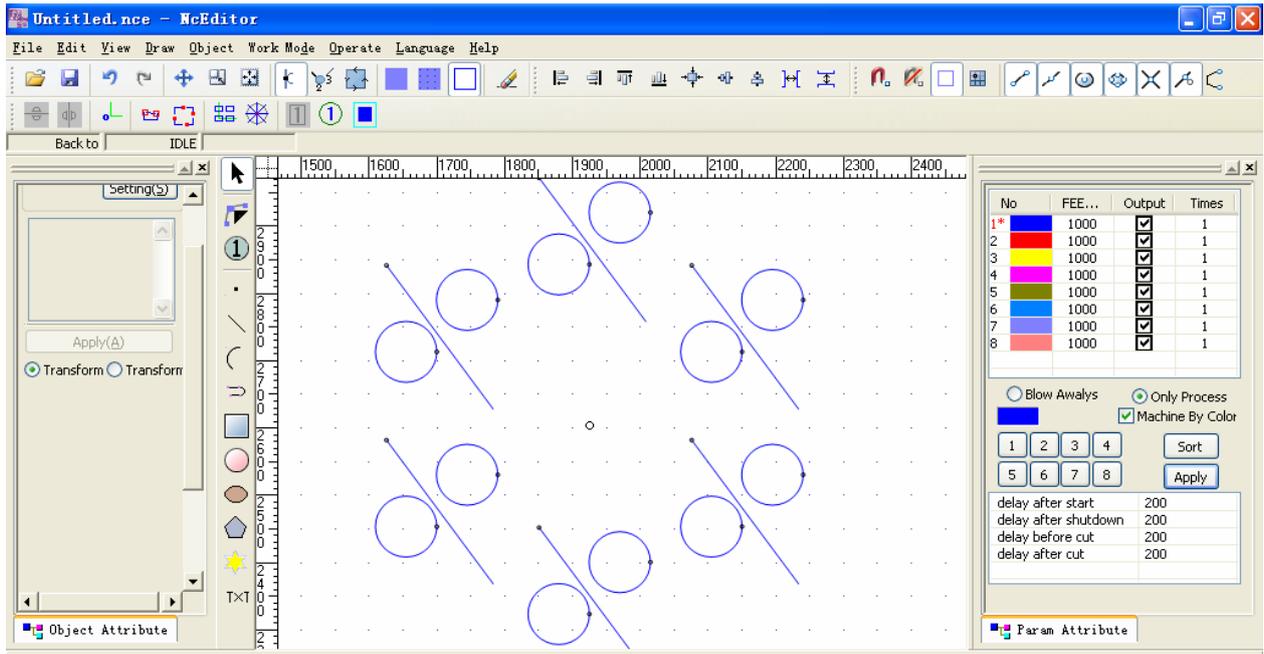


Fig. 4-49 The output after setting circle array

Note: multiple selected objects should be set as a group before using array function.

◆ [Horizontal Mirror]

Select one or multiple objects or groups (objects in different groups should be set as a group firstly) → choose [Horizontal Mirror] under [Object] menu or click the button  on toolbar → X mirror for the selected object is executed.

For a better view, Fig. 4-50 and Fig. 4-51 display the contrast before and after horizontal mirror.

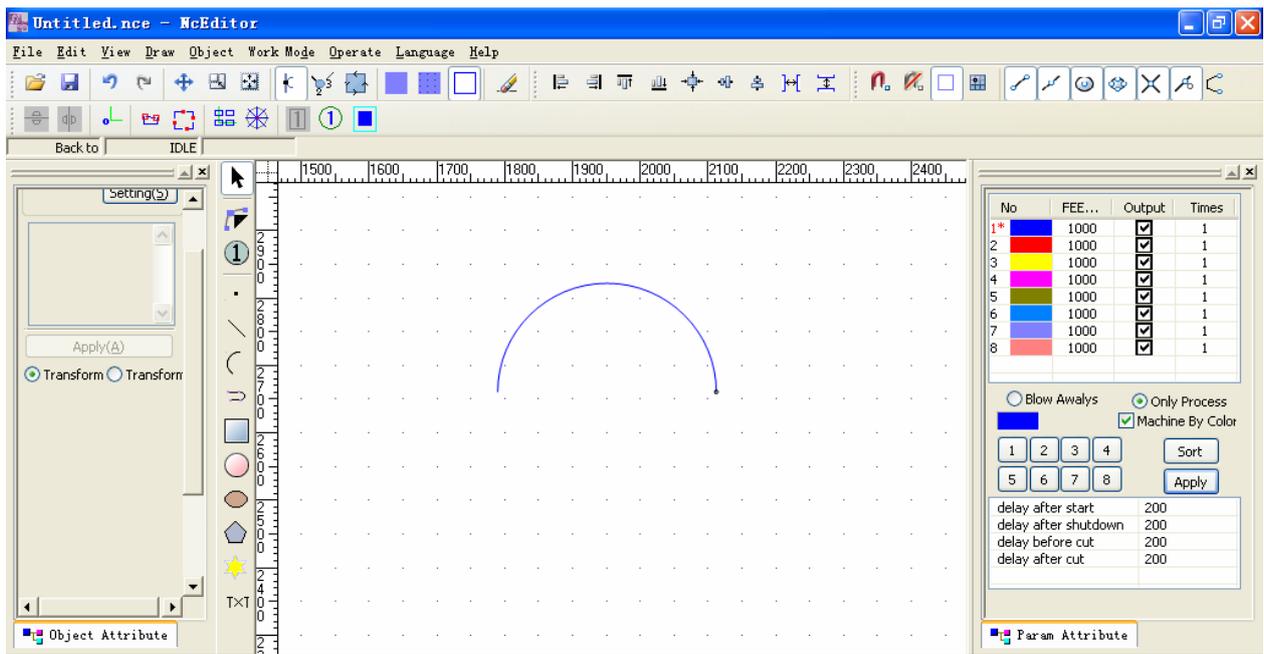


Fig. 4-50 Before [Horizontal Mirror]

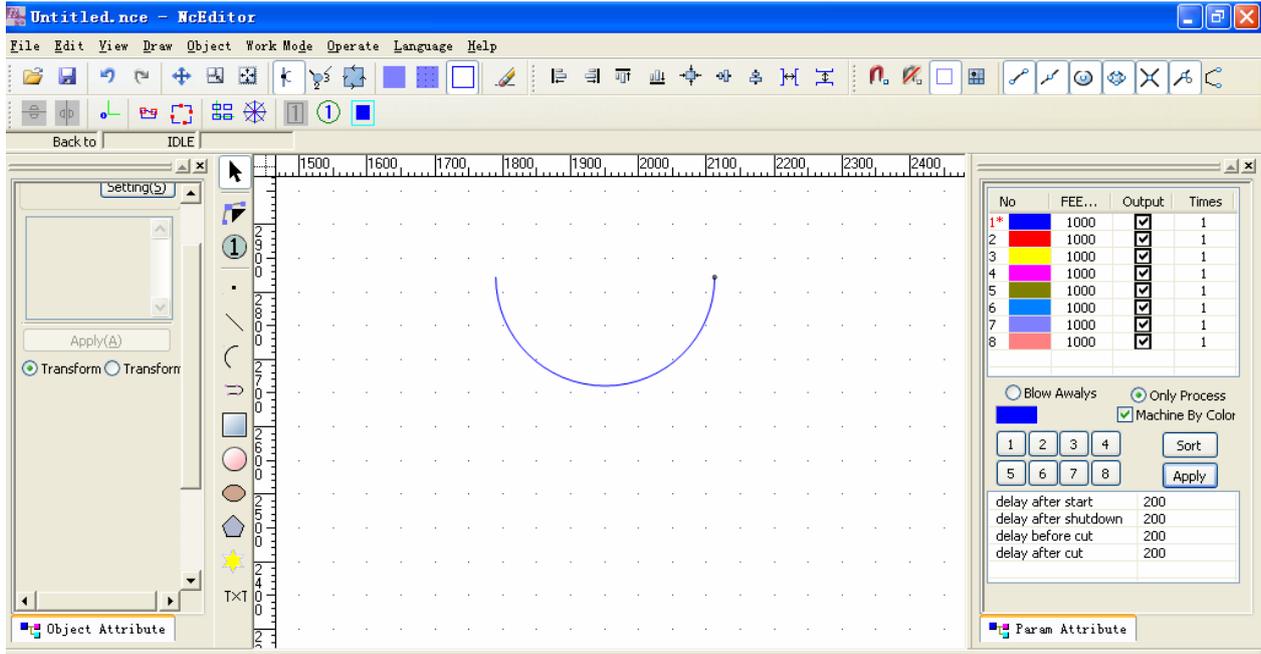


Fig. 4-51 After [Horizontal Mirror]

◆ [Vertical Mirror]

Select one or multiple objects or groups (objects in different groups should be set as a group firstly) → choose [Vertical Mirror] under [Object] menu or click the button  on toolbar → Y mirror for the selected object is executed.

To contrast, the views “Before [Vertical Mirror]” and “After [Vertical Mirror]” are exemplified as shown in Fig. 4-52 and Fig. 4-53 accordingly.

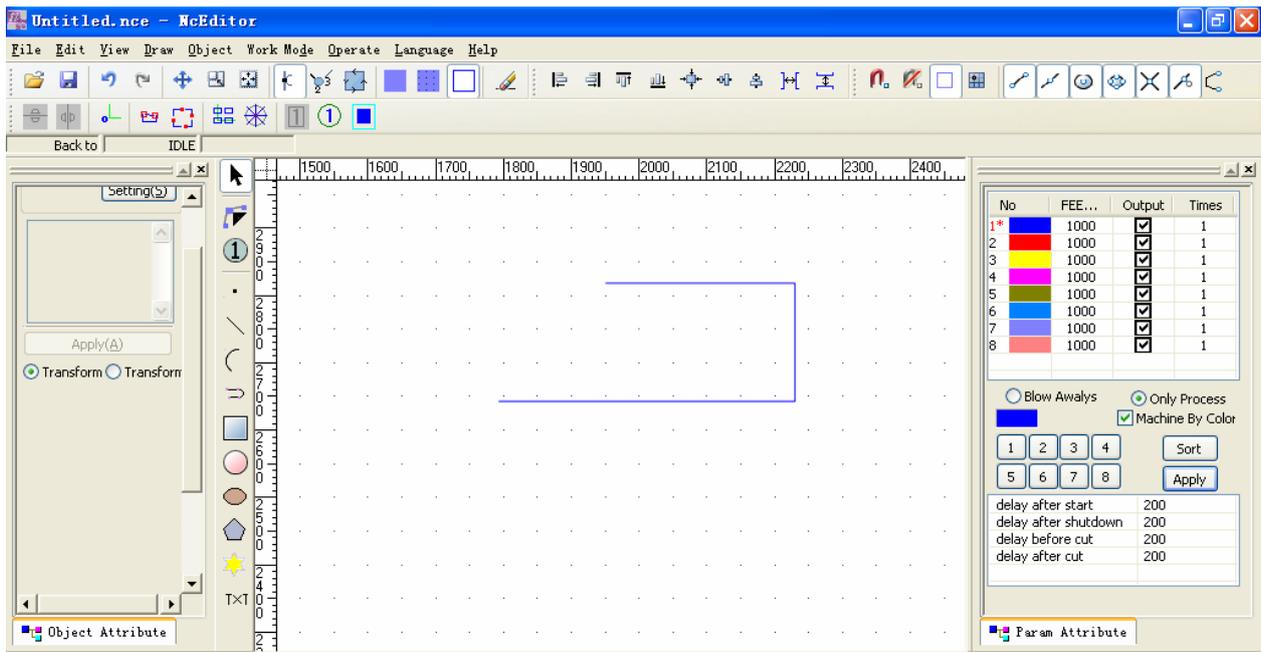


Fig. 4-52 Before [Vertical Mirror]

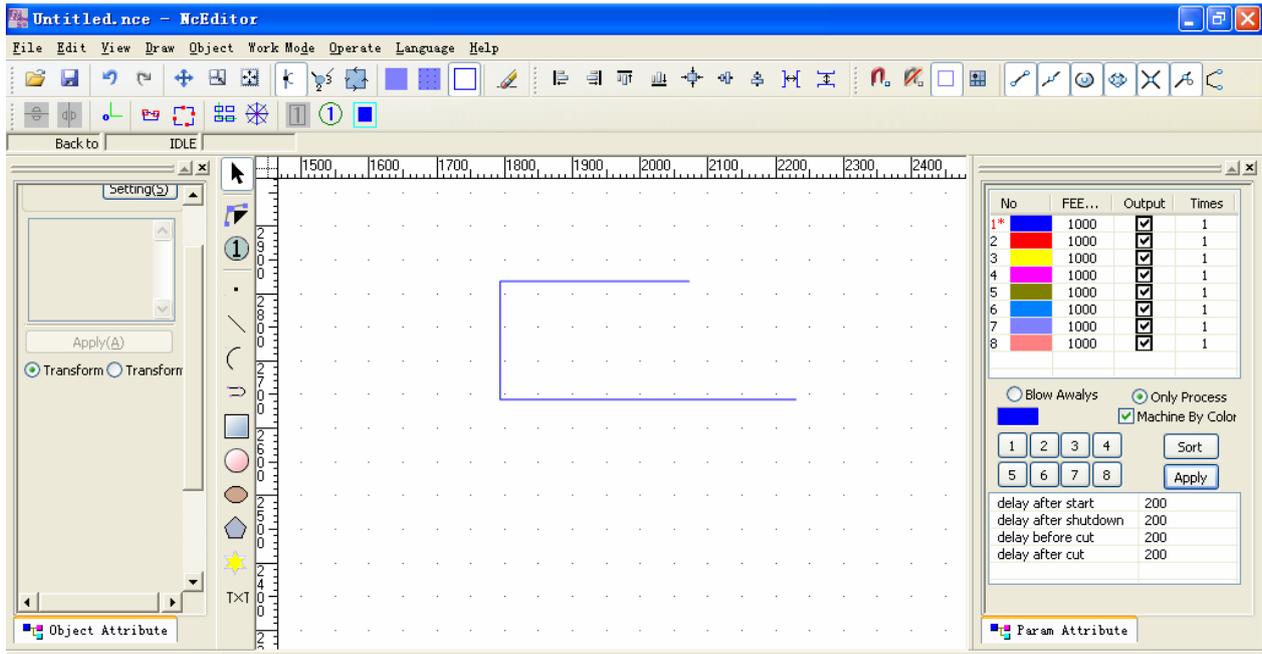


Fig. 4-53 After [Vertical Mirror]

◆ [Reverse Color]

This function inverts the color of objects.

Note: to validate [Reverse Color] function, save objects as monochrome bitmap format first.

◆ [Delete Same Object]

This item is used for deleting objects with the same shape, position and size. The concrete step is as following:

- Select the target object / objects.
- Choose the option of [Delete same object] under [Object] or under right-click menu.

◆ [Add Bridge]

The bridge function simplifies the processed parts management. After processing, the parts will not fall off and remain in the original position.

[Add bridge] is divided into [auto-add-bridge mode] and [manually-add-bridge mode], and [Add bridge] is available for a single object, but unavailable for groups.

Currently, [Add bridge] is available for a circle, an arc, a rectangle, a straight line and a polyline; [Add bridge] is unavailable for a point, an ellipse, and a group.

➤ [Auto-add-bridge mode]

Draw a rectangle and select it as following:

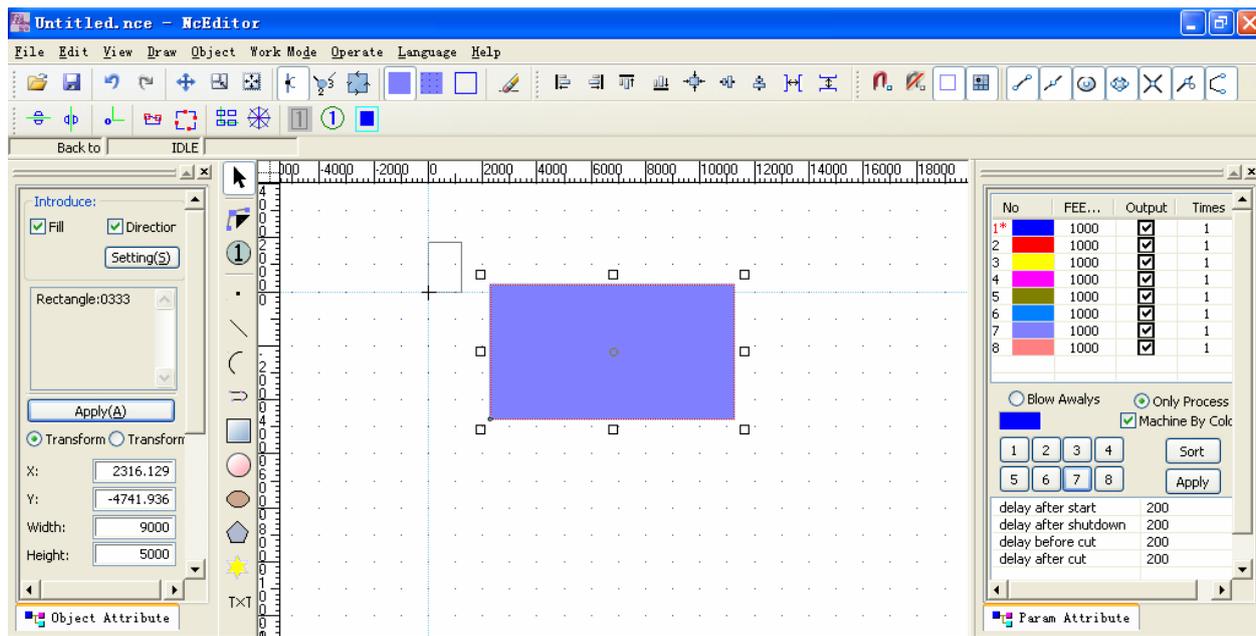


Fig. 4-54 Draw a selected rectangle

Right-click this object → choose [Add bridge] under right-click menu → choose “Auto-add-bridge mode” as following:

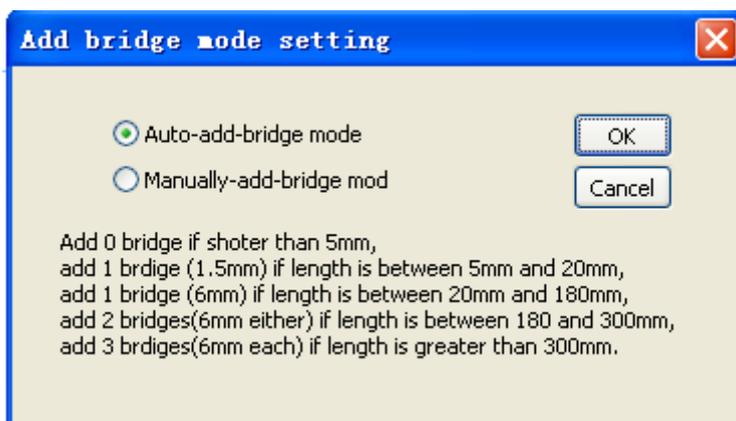


Fig. 4-55 [Auto add bridge mode] dialogue

Click “OK”, the output of rectangle as following.

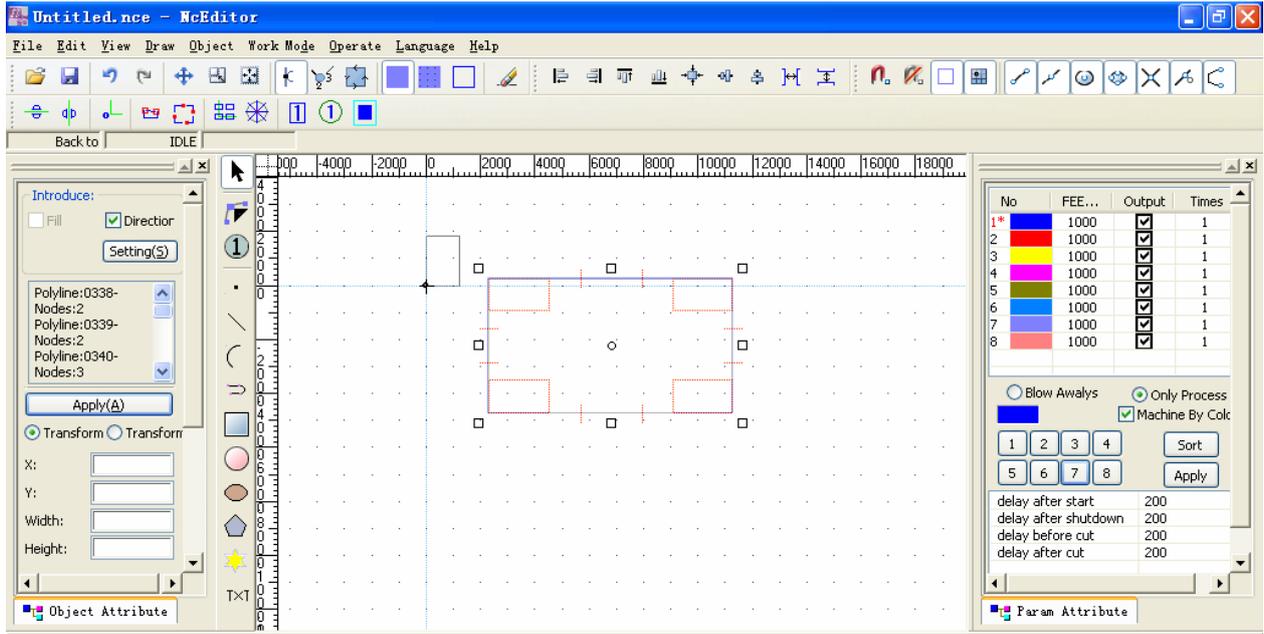


Fig. 4-56 After [Auto add bridge]

The gaps in the above picture indicate the bridge location. And during machining, they will be skipped to keep the rectangle on the workpiece.

➤ **[Manually-add-bridge mode]**

Draw a circle and a rectangle without selecting as following:

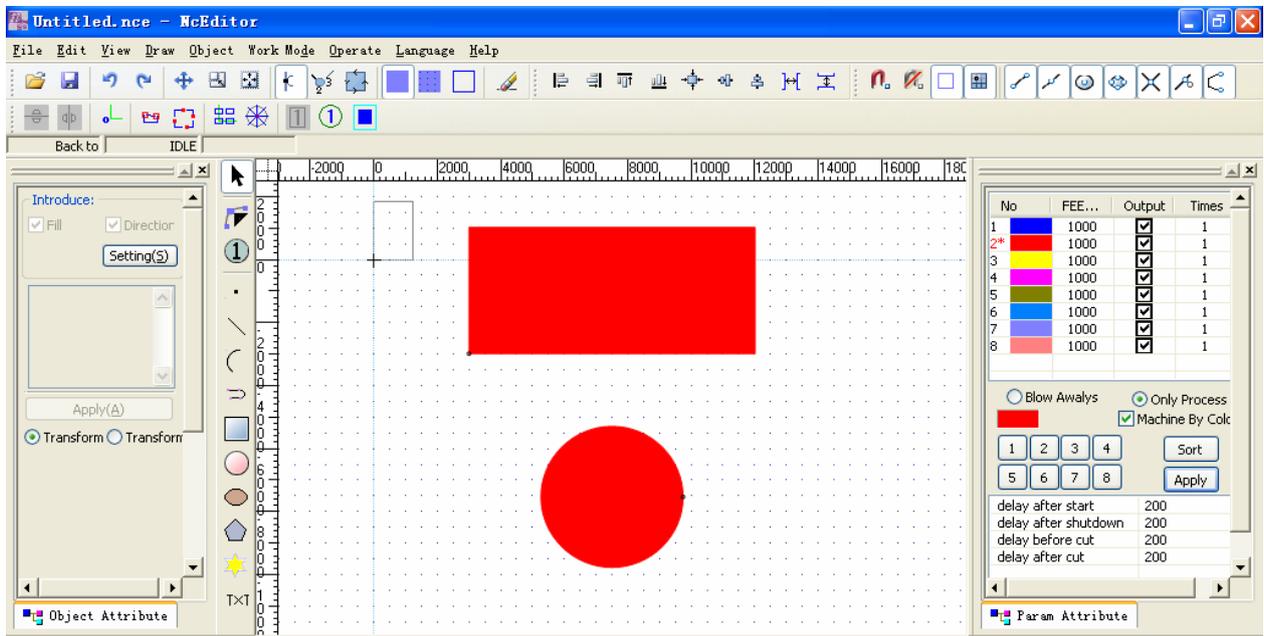


Fig. 4-57 Example of [manually add bridge] mode

Select [Add bridge] under right-click menu or under [Object] menu → choose [manually-add-bridge mode] → input the following data and press “OK”

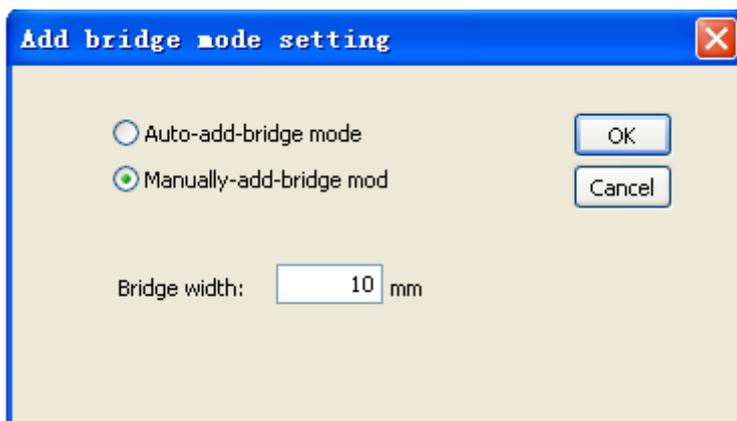


Fig. 4-58 [Manually add bridge] mode dialogue

After back to main interface, single click on the frames of the circle and rectangle to add bridge separately; the output is as shown in Fig. 4-59.

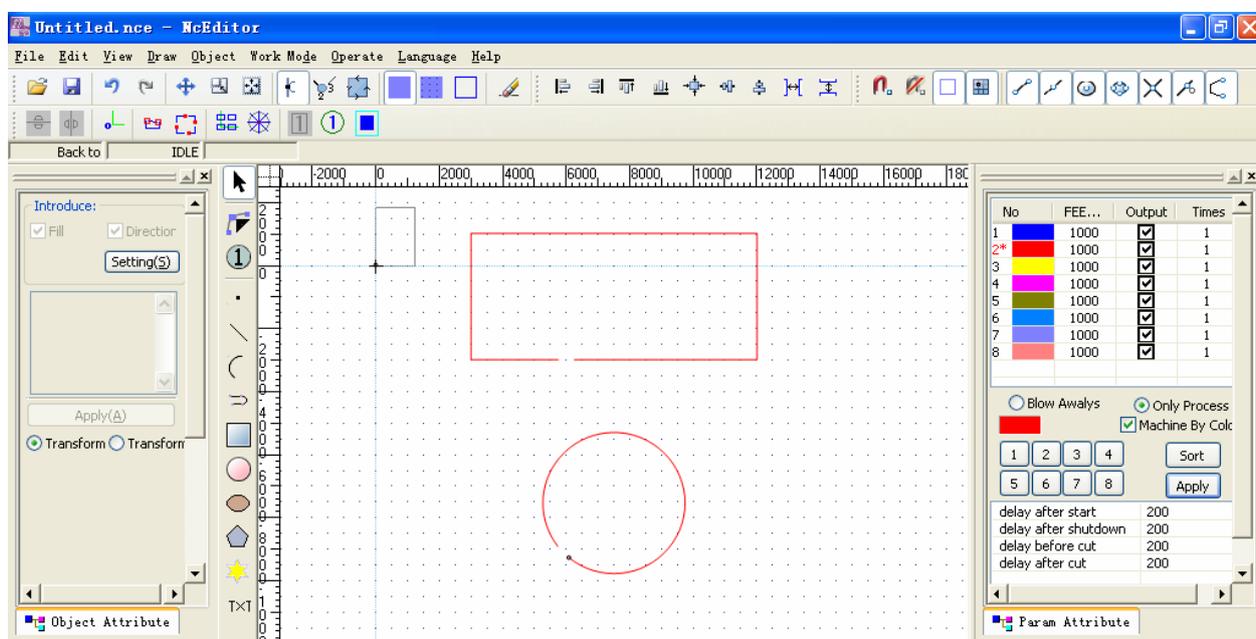


Fig. 4-59 After [manually add bridge]

◆ [Chamfer]

The software supports fillet and chamfer. As following, draw a selected rectangle.

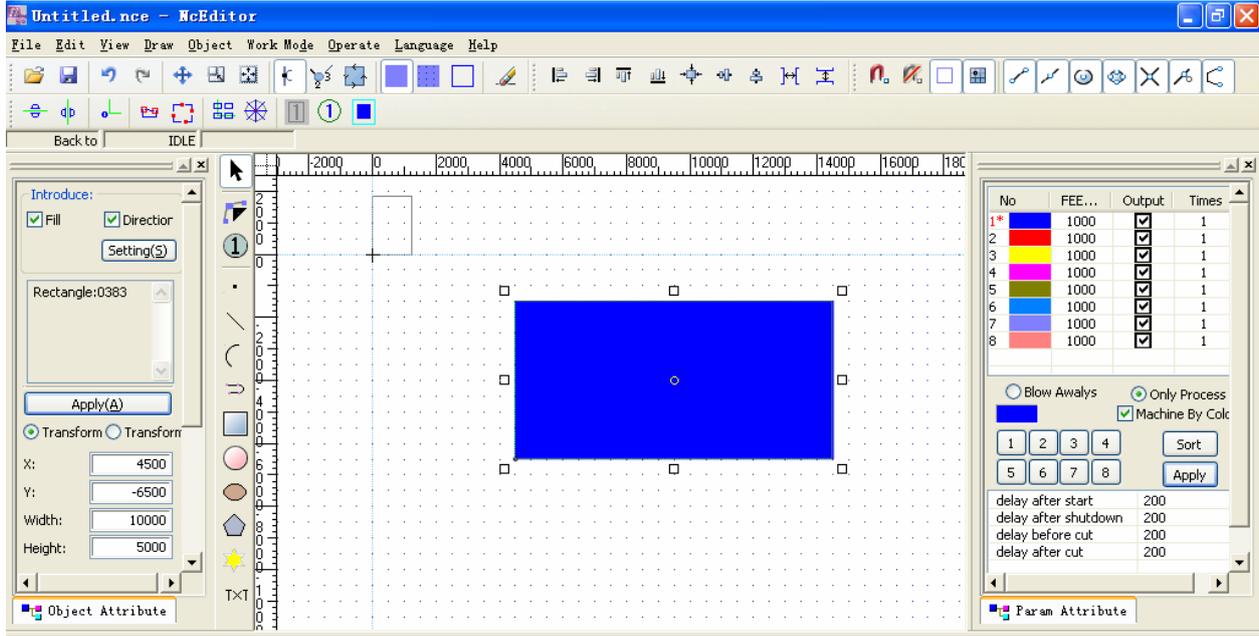


Fig. 4-60 Draw a selected rectangle

Choose [Chamfer] under [Object] menu to eject the following dialog.

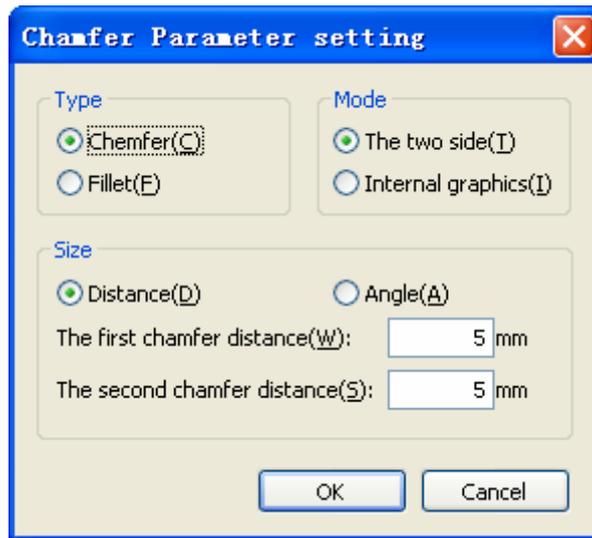


Fig. 4-61 Setting chamfer parameters

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. 4-61, under chamfer mode, there are two modes for setting size: [Distance] and [Angle]. Under [Distance] mode, input the distance of two sides, as Fig. 4-61, for instance, input 5mm into the input frame → press “OK”, cursor reshaping. A tip is displayed in the bottom of NcEditor. See Fig. 4-62.

Please select the first side.The two side must be two lines not parallel with each other or two lines in a Multi-Line not coincide

Fig. 4-62 First edge selecting tip

Select the first side, the tip prompting as Fig. 4-63. Then select the second side adjacent to the

first side. Then the rectangle in Fig. 4-60 will change into as Fig. 4-64.

Please select the second side. The two side must be two lines not parallel with each other or two lines in a Multi-Line not coin

Fig. 4-63 Second edge selecting tip

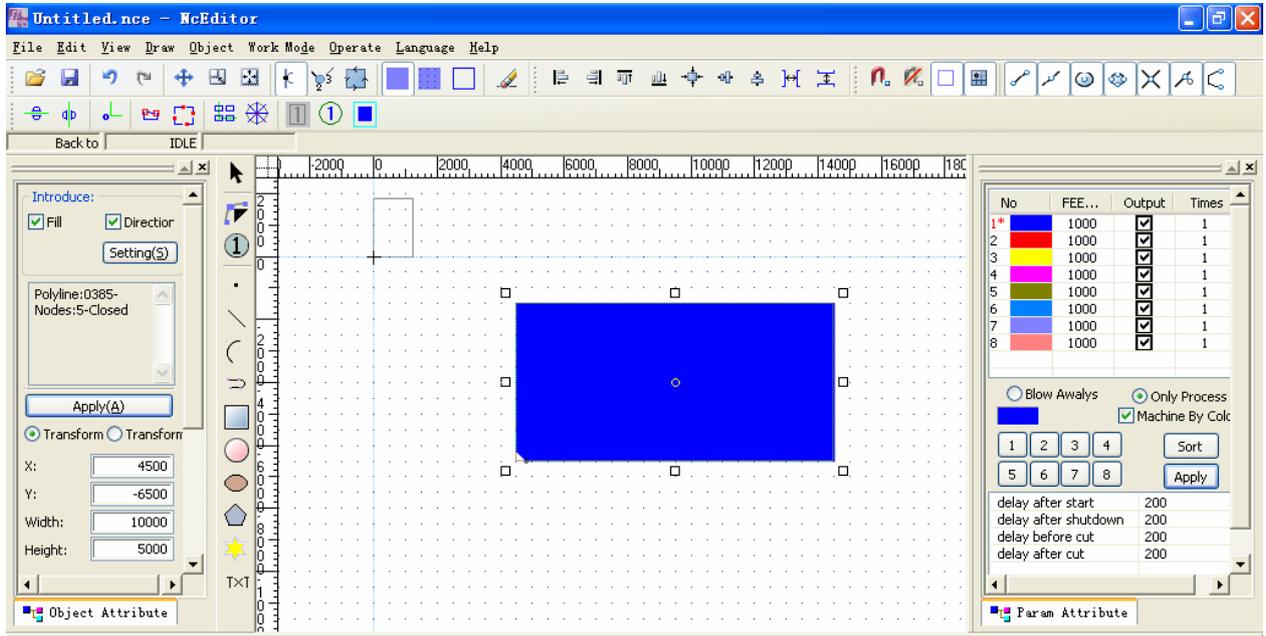


Fig. 4-64 Output after executing [Chamfer]

Input the data under [Angle] mode of [chamfer], as following.

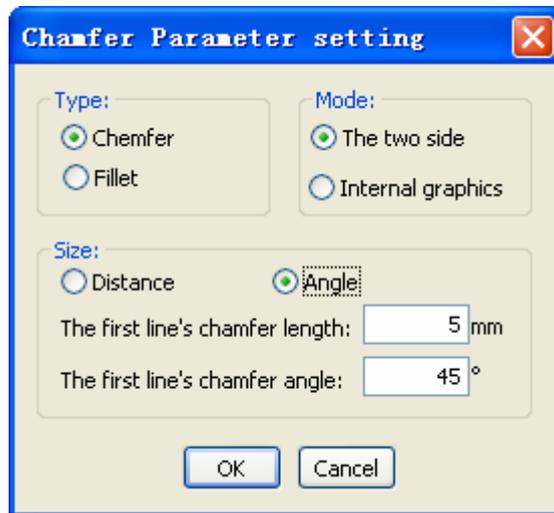


Fig. 4-65 [Angle] mode under [Chamfer] dialogue

After pressing [OK], please select the two adjacent sides; the manipulation is the same as that under [Distance] mode. The output is the same as Fig. 4-64.

[Two sides] of [Fillet] is used to set the round angle of two adjacent sides. The only way for setting size is [Radius]. Input the data as Fig. 4-66. Then press “OK” and select two adjacent sides; the output is as following in Fig. 4-67.

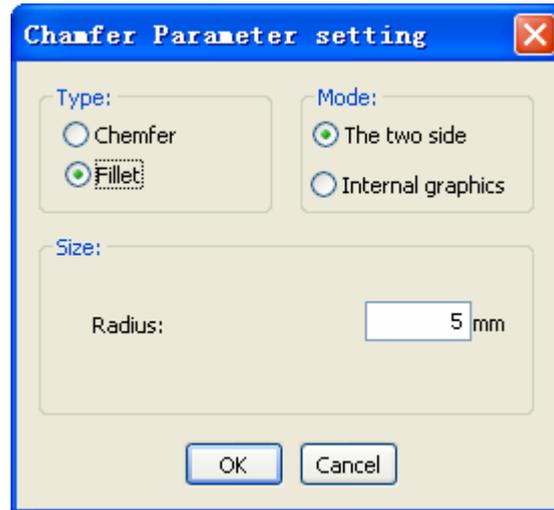


Fig. 4-66 Setting chamfer parameters

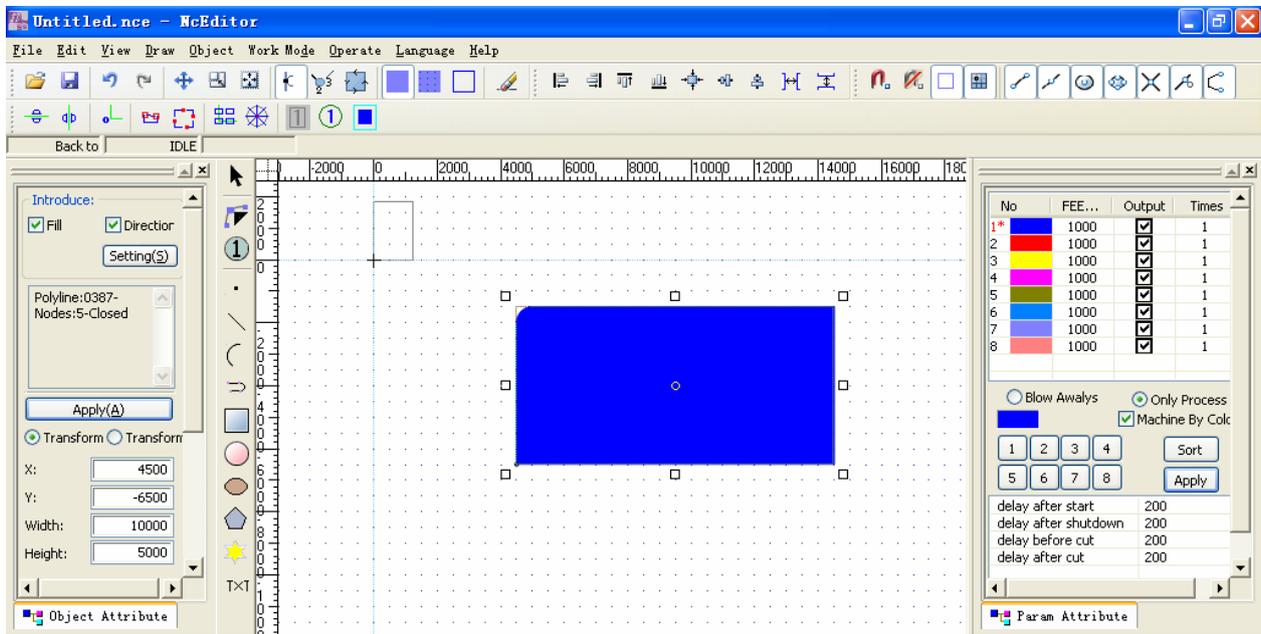


Fig. 4-67 Output after executing [Fillet]

➤ [Internal Graphics]

[Internal graphics] is ensuring all the qualified chamfers inside the graphics. Select [Internal graphics] mode, input data as Fig. 4-68. After pressing “OK” and selecting any edge of the object, the system will automatically process all the qualified included angles, and the output is as Fig. 4-69. Under [Internal graphics], there are also two methods: [Distance] and [Angle], which are identical to the above and will be omitted.

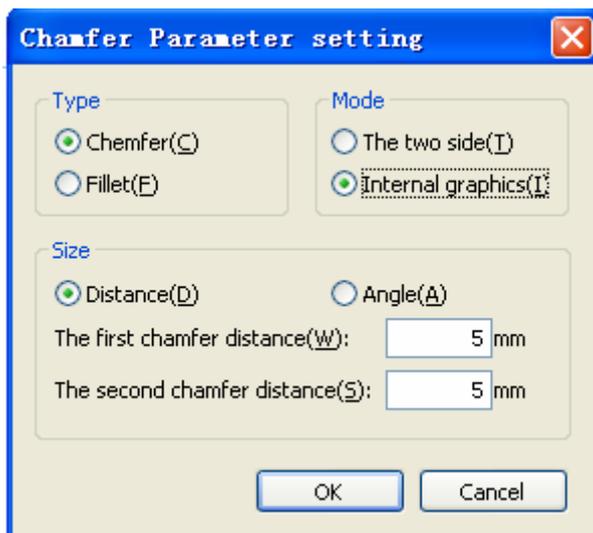


Fig. 4-68 [Internal Graphics] Mode of [Chamfer]

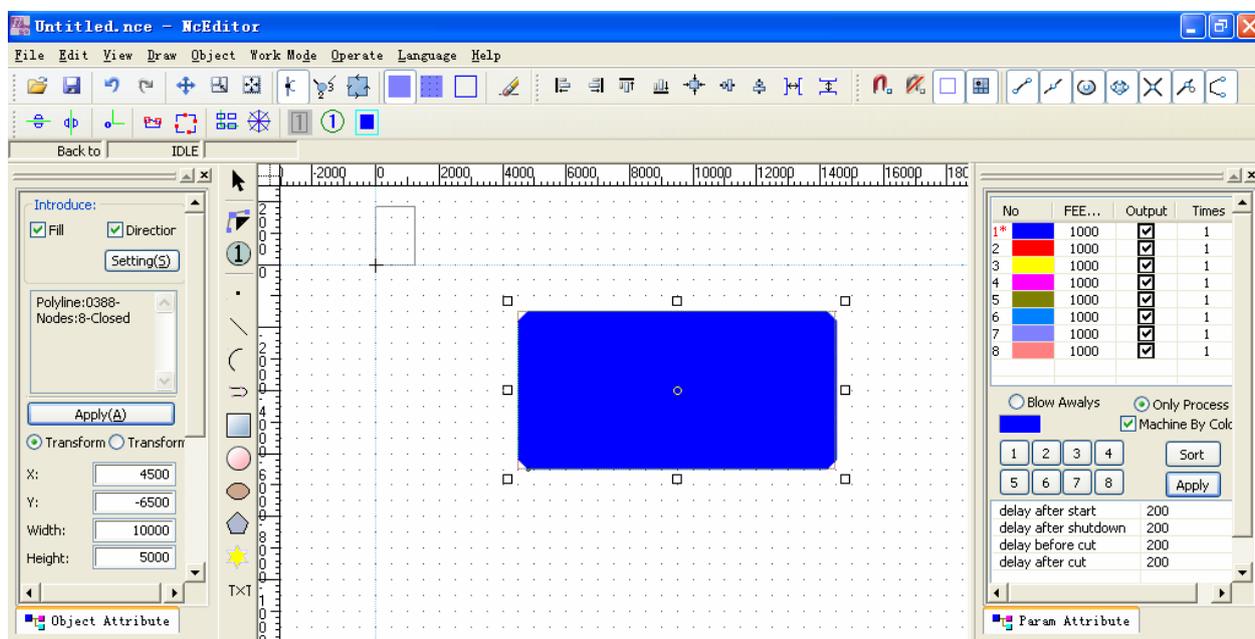


Fig. 4-69 Output after [Internal Graphics]

Under [Internal graphics] mode of [Fillet], the system will automatically process all the qualified included angles; input radius of 5 mm into the input frame and the output is as Fig. 4-70.

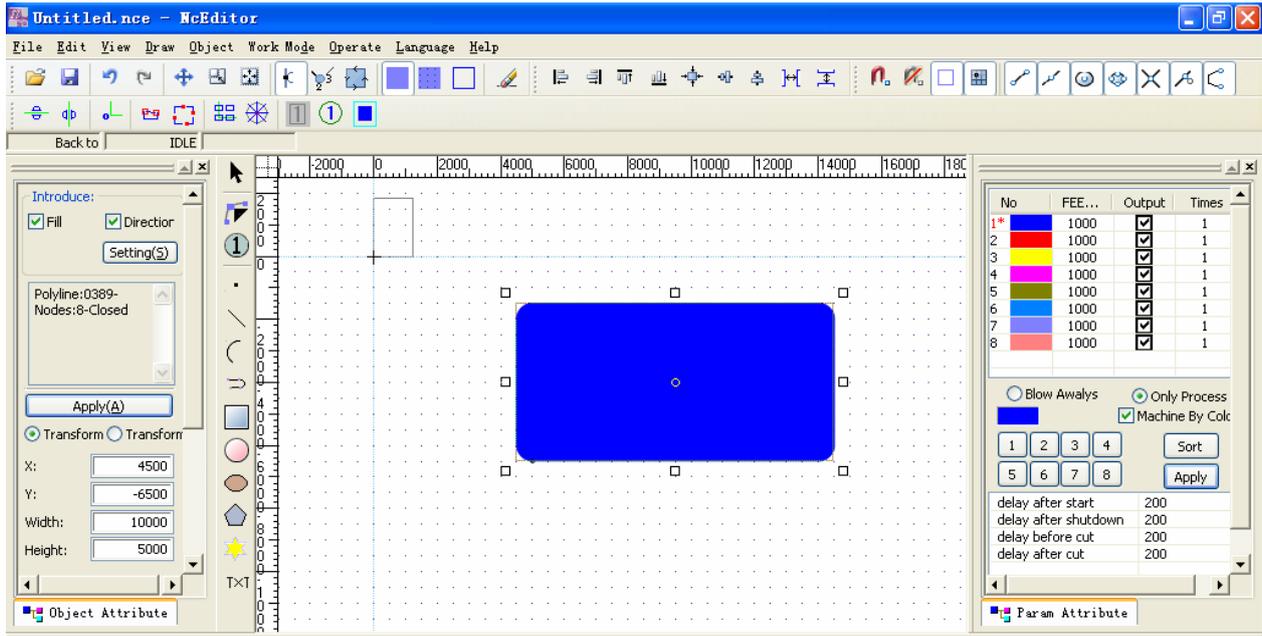


Fig. 4-70 Output after [Internal Graphics] mode of [Fillet]

◆ [Set Tool Path]

Slot (kerf), which is the loss part in cutting, occurs in laser cutting, so deviation exists in actually finished part dimension compared with theoretical part dimension (half of slot width in unilateral cutting and slot width in bilateral cutting). Outer shape of the actual part dimension becomes smaller, while inner shape becomes bigger due to the deviation. Physical dimension compensation is offered for this deviation in our system by [set tool path] function.

The operation steps are as following:

- Select the target object;
- Choose [Set Tool Path] under right-click menu to eject the following dialog;



Fig. 4-71 Kerf compensation setting dialog

- Set "Trans Type", "Compensation", "Tool (kerf width)" in the dialogue. If the outer part is to be reserved, please select compensation as "Shrink", and "Expand" to keep the inner part. Several

compensation samples of Line [Trans Type] are as following.

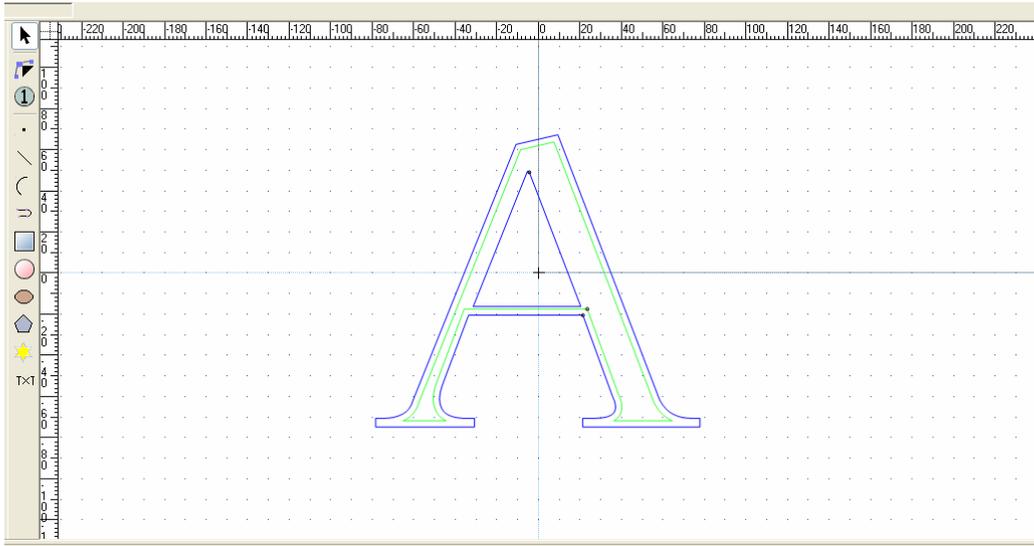


Fig. 4-72 “Shrink” Compensation

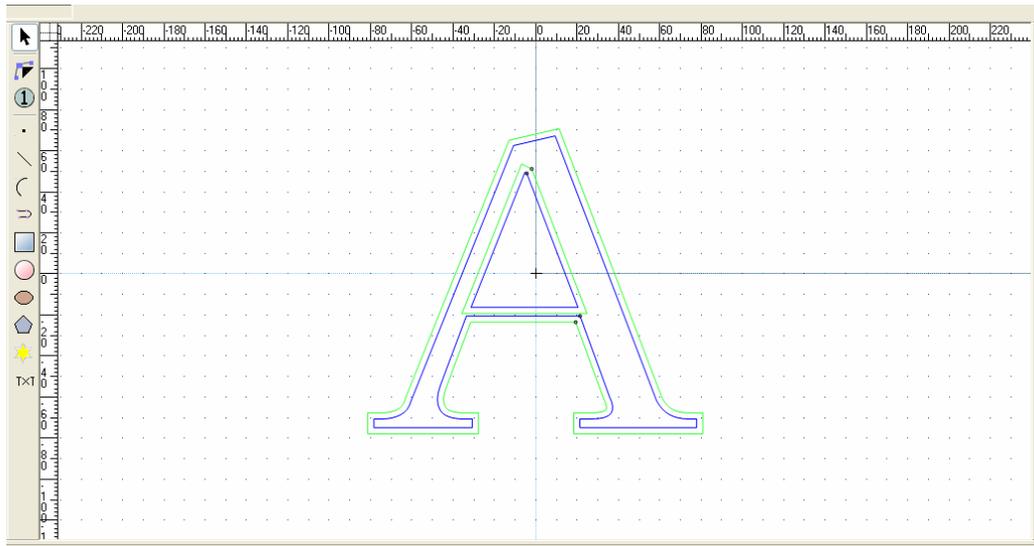


Fig. 4-73 “Expand” Compensation

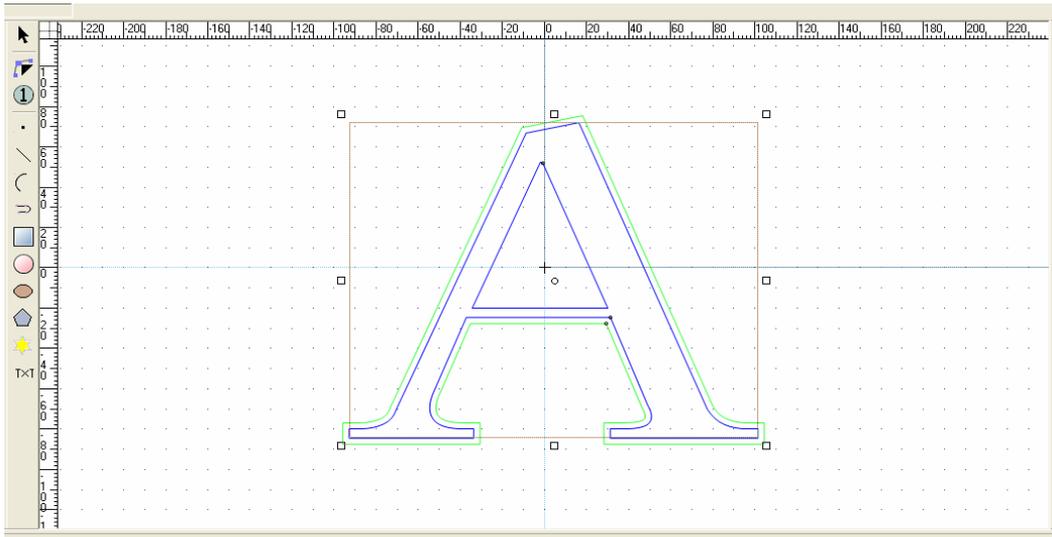


Fig. 4-74 “Shrink or Expand” compensation

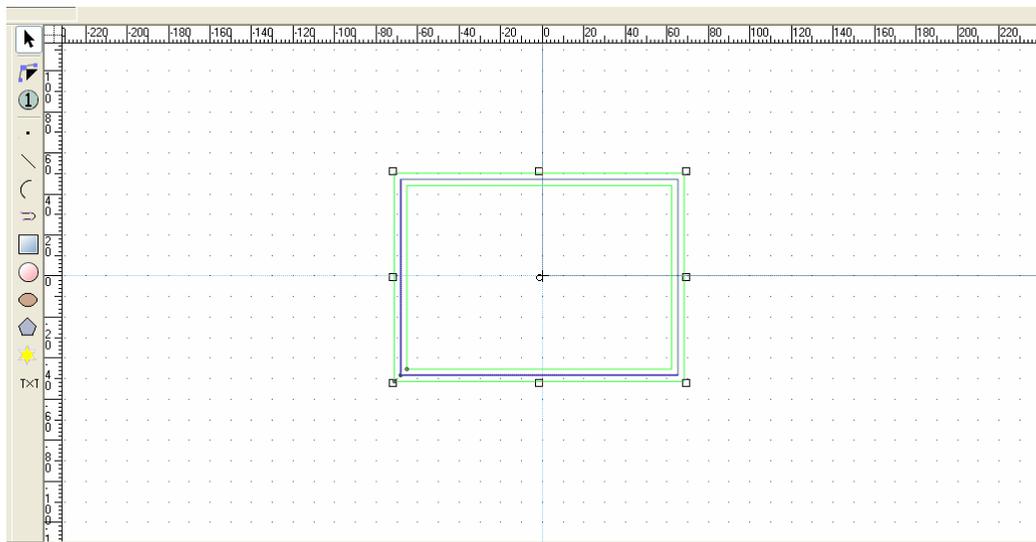


Fig. 4-75 “Shrink and Expand” compensation

◆ **[Delete Tool Path]**

This option is used for deleting kerf compensation setting.

◆ **[Single Block]**

The menu item is only available under AUTO mode. The user can set the selected program to be executed as [Single Block], which is a good way for false diagnosis and fault recovery action. Under the [Single Block] mode, processing stops when the resultant velocity of each axis is zero.

◆ **[Mach. Selected Objects]**

This item is applied to processing the selected objects without processing other unselected objects, which is available for a single object or multiple objects. The machining will stop after finishing machining of the selected objects.

There are two methods for enabling this function, as following:

- Click [Mach. Selected Objects] under [Object] menu;
- Choose [Mach. Selected Objects] under right-click menu

◆ [Clear Process Trace]

When an object is being processed, the red tracks on the object equal to “processed”. After machining is finished, the object will turn into red. Actually, the color is the processing track, rather than the color of the object. If necessary, select [Clear Process Trace] to clear the track.

After machining is finished, there are several methods to fulfill [Clear Process Trace] as following:

- Choose the option of [Clear Process Trace] under [Object].
- Choose [Clear Process Trace] under right-click menu.
- Click the icon  on toolbar.

Note: [Clear Process Trace] is unavailable during machining.

◆ [Text Translate to Graphic]

Like the method mentioned in drawing a text, click the  in [Object editor space], then change “TEXT” to desired words, like “WEIHONG”, and adjust the font and size.

Right click on “WEIHONG”, select “Ellipse translate to polyline” under right-click menu; the output is as shown in Fig. 4-76.

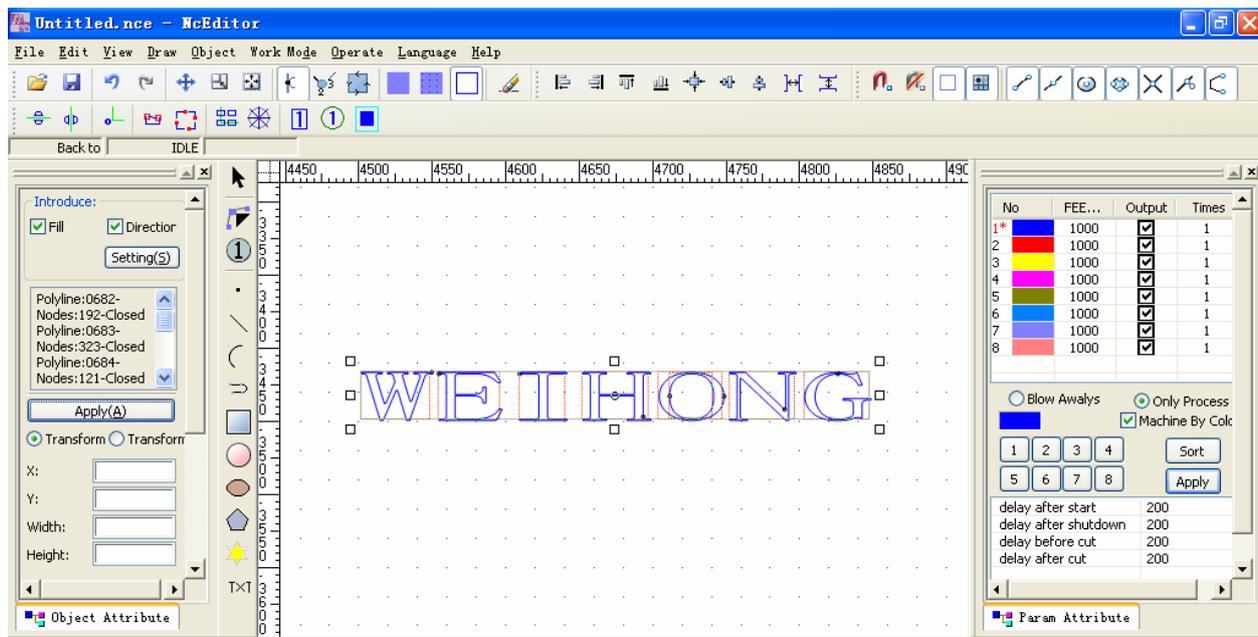


Fig. 4-76 Example of [Text Translate to Polyline]

◆ [Dispart CadPath]

This function is mainly used with add bridge function set lead-in/out line function. After [Dispart CadPath], bridge contacts and lead-in/out line, etc, can be set.

◆ [Ellipse Translate to Polyline]

Draw an ellipse in object editor space, as shown in Fig. 4-77.

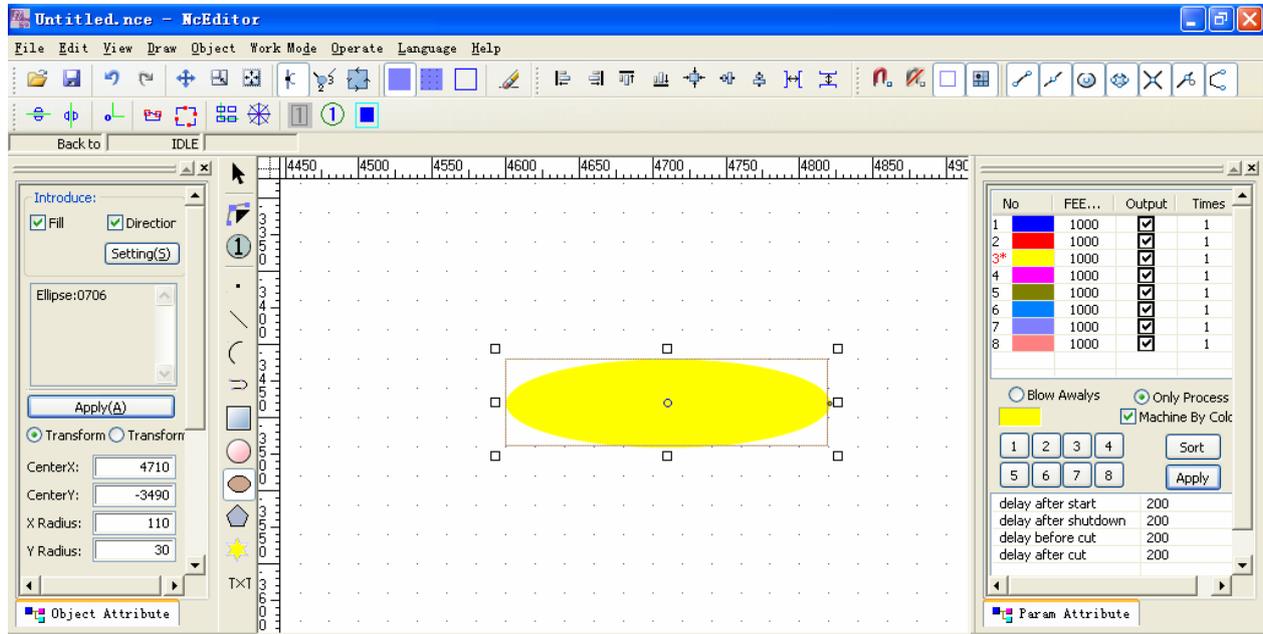


Fig. 4-77 Example of [Ellipse Translate to Polyline]

Select this ellipse, and then select [Ellipse translate to polyline] under [Object] menu or under right-click menu; the output is as shown in Fig. 4-78.

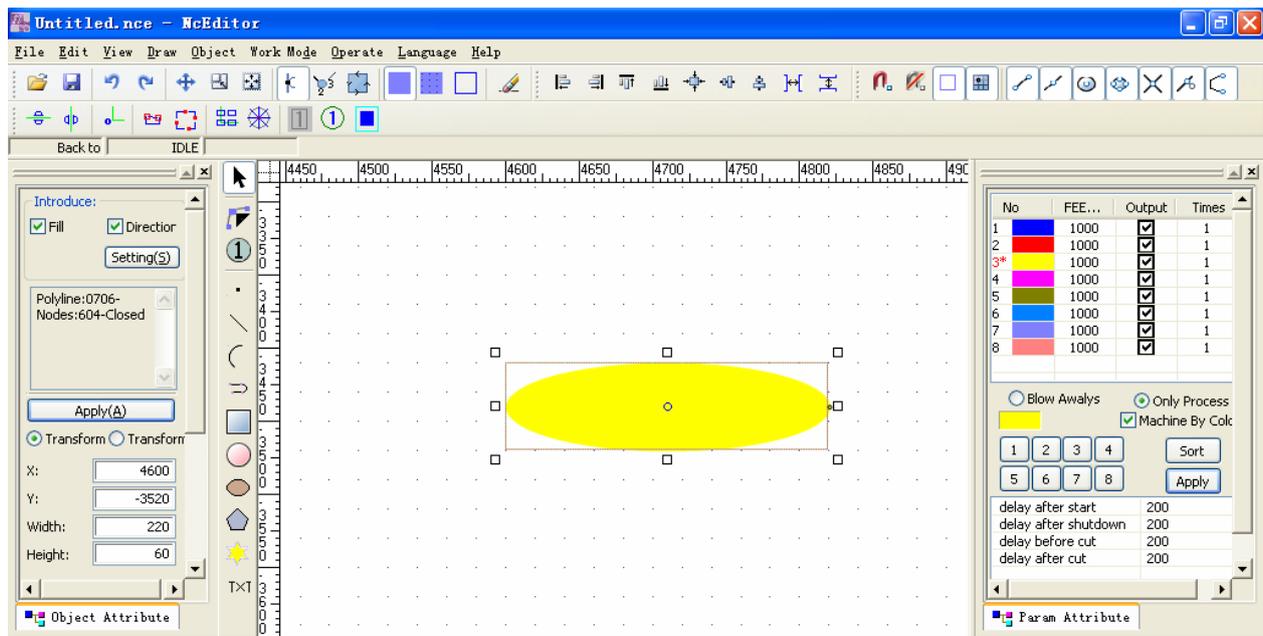


Fig. 4-78 Output after [Ellipse translate to polyline]

◆ [Gallery]

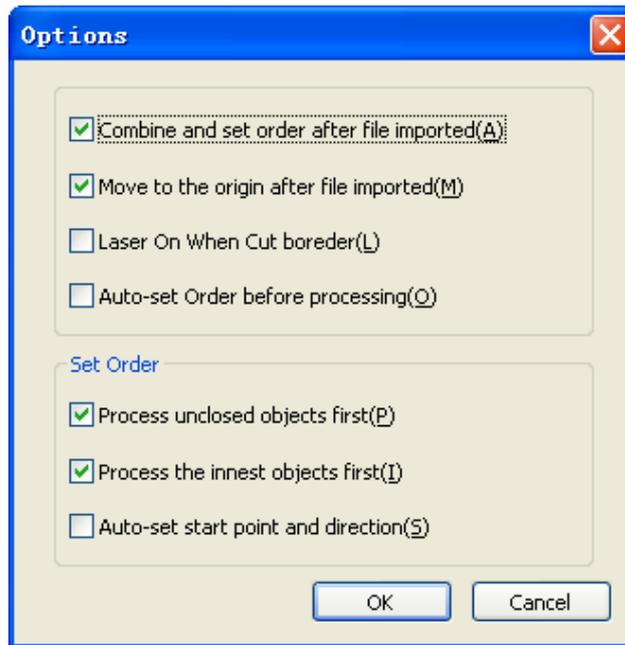


Fig. 4-80 Options

This function is mainly for parameter settings related to “load files”, “frame check”, “sort”, etc.

4.7.6. Work Mode Menu

Work Mode menu includes Auto Mode, Manu Mode, Back to Reference Point Menu, etc, corresponding to Auto, Manual and BKRef buttons.

4.7.7. Operate Menu

Operate Menu includes various commands of machining operation, such as “Single Block”, “Handwheel Guide”, AutoBack”, “Set Workpiece Origin”.

◆ [Single Block]

Refer to Chapter 4.7.5 for [Single Block].

◆ [Handwheel Guide]

Connect a handwheel if it is to be used to control machining. Then check this option. After machining starts, the system will execute the program with clockwise turning of handwheel at a speed controlled by handwheel turning speed, and stop execution with the stop of handwheel turning.

◆ [AutoBack]

This is a newly added function of V9 version, only valid under Auto mode, used for reversing during machining. To move against the original path to back to a previous position, select this option.

◆ [Set Workpiece Origin]

This menu item is used to set current position as workpiece origin, which won't cause any motion of actual position. The user can also set workpiece origin in [WorkCoor] and [Offset] screens of [OFFSET] section in NcStudio.

◆ **[Save Workpiece Origin]**

This menu item is used to save current workpiece origin into the system machining file. Regarding workpiece origins frequently used, the user can save them for future use. At most 10 groups of workpiece origin can be saved. After setting a workpiece origin for the first time, by saving this workpiece origin, the user can locate it swiftly.

It is not recommended to modify a file with saved coordinate origin by edit function in our software or other edit tools.

◆ **[Load Workpiece Origin]**

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

◆ **[Start], [Pause], [Stop]**

They correspond to [Start], [Pause] and [Stop] buttons in Machine Control Bar, only valid under Auto mode, also applicable to simulation function.

◆ **[Simulation Mode]**

It corresponds to the button [Start Simulate] in Machine Control Bar. If a machining procedure has already been loaded, and the system is in "IDLE" state, choosing the menu item, the system will carry out rapid simulation automatically from the first section. Simulation function provides the user with a fast and vivid simulating machining environment.

To run a machining file under simulation mode, the system will not drive the machine tool to do the relative actions, but only show the cutter trace with high speed in the processing trace window. By simulation, the user can know the cutter's movement in advance to avoid damage to the machine tool which is caused by a mistake (e.g. stroke over-travel) in writing the procedure, and also get other additional information.

Once simulation begins, this menu changes to "Withdraw the simulation mode". And clicking it will exit from this mode. During simulation, [Start], [Pause], [Stop] and [Continue] will change to [Simulation Start], [Simulation Pause], [Simulation Stop] and [Continue] respectively.

◆ **[Select Processing Block]**

Only available to a G code file.

Its counterpart is the button [SelWork] in [POS] function section of Auto mode, only valid under Auto mode and simulation mode, with shortcut key as Ctrl + F9. This function realizes block skip function. After selecting this menu item, a dialog will pop up, as shown in Fig. 4-81. Operate in terms of prompts.



Fig. 4-81 (Advanced Options) dialog

◆ [Breakpoint Resume]

The item corresponds to button [continue] in Machine Control Bar. It is only available under AUTO mode and the shortcut key is Shift + F9. When this item is selected, the system will resume machining from the breaking point of last time. It is also applied to simulation mode.

◆ [Back to Workpiece Origin]

Workpiece origin is the zero point of workpiece coordinates system and workpiece dimension programming; 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 is decided by [Set Workpiece Origin].

After the menu item is selected, the spindle will go back to workpiece origin automatically from the current position.

[Back to Workpiece Origin] is corresponding to the button [BackWorkOrg] under [Auto] & [Manual] mode.

◆ **[Back to Fixed Point]**

A fixed point can be set in advance; when there is laser head damage or laser head change is needed, selecting this menu item, the system will uplift the spindle to back to the fixed point for laser head change.

◆ **[All Back to Reference Point]**

This menu item is used to call all the axes to return to the reference point. We strongly recommend using this function before machining or after E-stop.

◆ **[Reset]**

The menu item corresponds to [Reset Pause] in Machine Control Bar. Selecting this menu item, the machine tool will stop current machining task (If it is machining) immediately, and restore from warning state (If warning occurs) to IDLE state.

Using [Reset] button to stop is different from using [Stop] button. [Reset] button terminates machining with E-stop, but spindle's state and cooling will not be changed, and there is no deceleration and cutter uplift.

◆ **[Mach. Selected Objects]**

Refer to Chapter 4.7.5 for [Mach. Selected objects].

◆ **[Mach. by Color]**

Different colors can be selected for each object. After sorting the sequence of colors, machining can be performed following the order of colors, for instance, first red, then blue.

Color sorting zone is at the lower part of color window of machining parameters, which can also be used to set a color for machining objects, as shown in Fig. 4-82. After selecting a machining object, click a number to set the corresponding color as the object color. Then check the option "Machine by color" in this zone, and then click Start. The machining effect is the same as using this menu item.

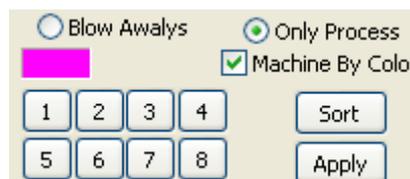


Fig. 4-82 Color sorting dialogue

◆ **[Mach. Current Group]**

If only a group is needed for machining, this function can be selected. After this function selected,

only the currently selected group will be processed regardless of other objects. It is unavailable for a single object.

◆ [Mach. from Selected Objects]

There is an order for machining objects to be machined; after selecting [Mach. from selected objects], only the selected object and its backward objects will be machined while the forward ones will not be machined. For instance, there are 8 objects to be machined, if the sequence of the currently selected object is “3”, then the No. 3 to 8 objects will be machined, but No.1 and No.2 won't.

4.7.8. Language Menu

Currently, there are Chinese and English under Language menu for option. The user can select an appropriate one. The two languages can be switched directly in the system without restart.

4.7.9. Help Menu

◆ [About]

Select the menu item to eject a dialog “About NcStudio”. See Fig. 4-83.

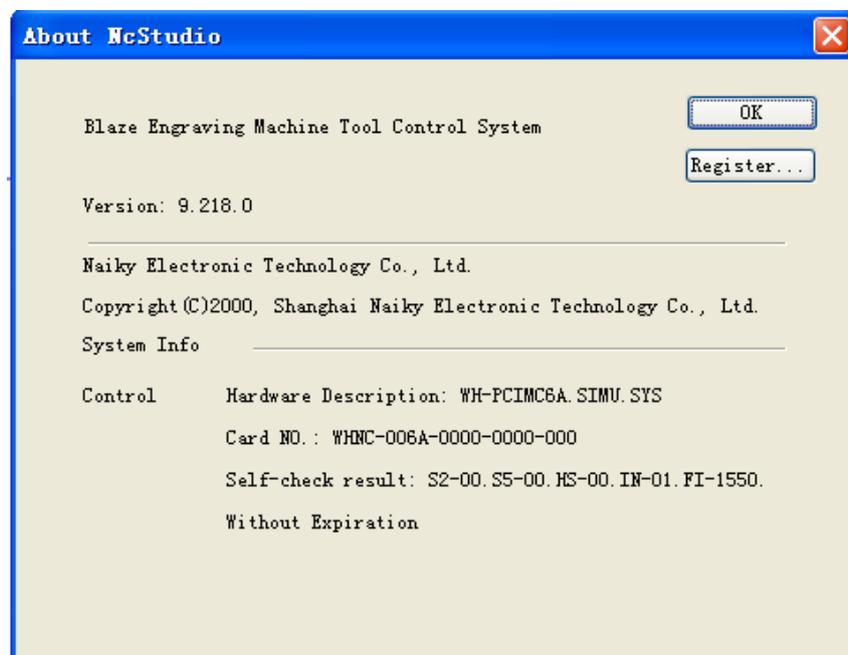


Fig. 4-83 “About NcStudio” dialogue

In the above figure, the user can check the information of version, motion card, and manufacturer, etc.

Register function is used for limiting the system usage time. After the time is run out, send the Card No. to the machine tool manufacturer to get registration code. Then click the button [Register...], then input the received code into the following dialog, and then click [OK] to finish registration.

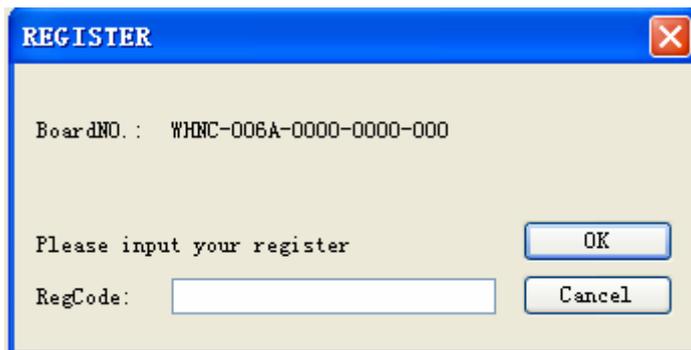


Fig. 4-84 REGISTER Dialogue

Note: the Card ID changes with registered times, reflected in the last three numbers of Card No.. When the registered times is “0”, the last three numbers are “000”; when “1”, the last numbers will become “001”.

4.8. Machine Control Section

4.8.1. Auto Mode

The overall machining window under Auto mode is as shown in Fig. 4-85.

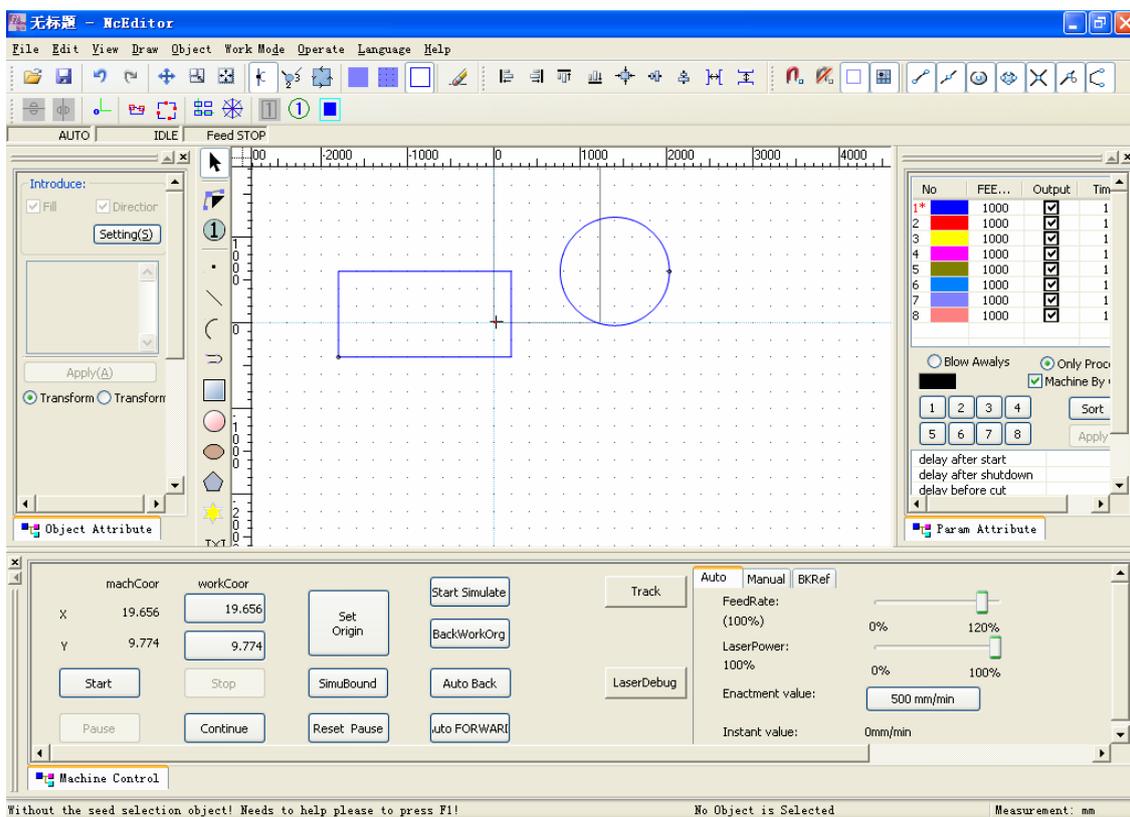


Fig. 4-85 [Automatic Mode]

On the left of the [Machine Control] section, there are coordinate display zone and several machining related buttons, including [start], [stop], [pause], [continue], [ser origin], [auto back], [reset

pause], [simubound], [backWorkOrg], [laser debug]; machine coordinate and workpiece coordinate are shown in coordinate display zone.

On the right of the [Machine Control] section is mode button section, in which the buttons vary with different modes.

The overall machine tool control window is as shown in Fig. 4-86, under automatic mode.

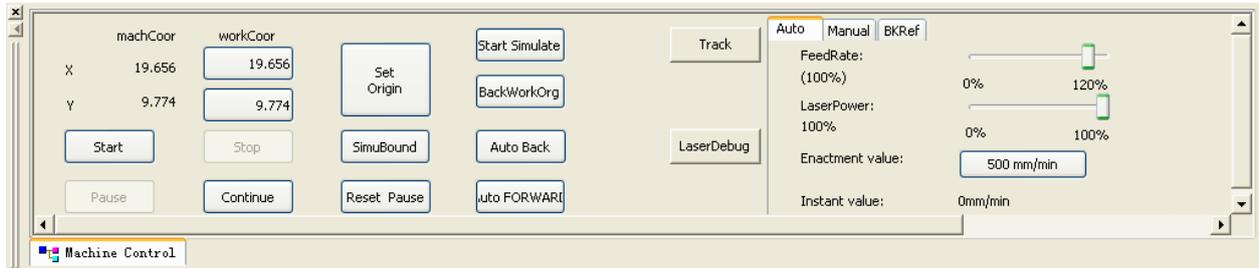


Fig. 4-86 [Machine Tool Control] section under [Automatic] mode

By default, “Back to Machine Origin” must be executed before machining, however, [Set Directly] can also be adopted when the machine origin position is definitely assured; or, when lower precision is enough, the user can click “Start” to begin machining directly after setting the parameter N74001 “NEED _ REFPT _ BEFORE _ MACHINING” as “False”.

◆ [Start]

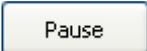
See: , Hotkey: F9, only available under AUTO mode.

- If there is no loaded processing file, the button is disabled.
- After a processing file is loaded and operation of backing to the reference point is finished, the button is enabled. Pressing the button, the system will start machining automatically from the first line.
- If a processing file is loaded for the first time but operation of backing to reference point (“directly setting” or “back to machine origin”) is not performed, the system will prompt the user to back to the reference point after this button is clicked.

The button has two functions:

- If a certain processing procedure has been loaded and “back to reference point” has been finished, and the system is under “idle” state currently, pressing the button, the machine tool will carry out the procedure automatically from the first section. Once machining starts, the system will enter into “Auto | Running” state. If the system is in the state of simulation, the system will start simulation machining.
- If the system is under “Auto | Pause” state, pressing the button, the system will resume machining from the suspended position and enter into “Auto | Running” state. If the system is under simulation state, the system will continue simulation machining.

◆ [Pause]

See: 

[Pause] button is valid under auto processing. Clicking the button, the machine tool will pause and lift the cutter and then enter into “Auto | Pause” state. To resume machining, press the button [Start] or choose the relative menu item.

If the system is now under simulation state, pressing the button, simulation will pause and the system will enter into “Auto | Pause” state. To resume simulating, press the button [Start] or choose other relative menu item.

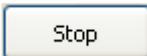
Hotkey: F10, only available under AUTO mode.

Note:

The system has two ways to enter into [Pause] state:

1. When the system is executing [single block] function;
2. The user selects [Pause] function during machining process.

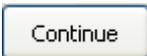
◆ [Stop]

See: 

This function is enabled under auto machining, or when G code is input and executed. At this time, pressing the button, the machine tool will stop and uplift the cutter to terminate all the machining tasks, then the system enters into “Auto | Idle” state. This function is to break off the machining procedure normally during machining.

Hotkey: F11, only available under AUTO mode state or when input G code is being executed.

◆ [Continue]

This button  equals to [Breakpoint Resume], which is available only under AUTO mode.

When power-off or E-stop occurs, or when [Stop] button is clicked during machining process, if the accuracy of workpiece coordinate can be ensured, the user can directly press this button to continue machining. If not, go back to the reference point firstly and then resume the machining. This function can also be applied to simulation mode.

◆ [Set Origin]

After the button  is clicked, the system will alter the offset to ensure X and Y

workpiece coordinates of current point are both zero.

◆ [SimuBound]

Pressing the button  after loading a machining file, the system will walk around along the circumscribed rectangle of machine path to determine machining stroke. This function is optional.

◆ [Reset Pause]

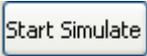
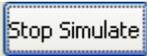
See: 

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

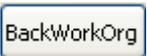
Using [Reset Pause] button to stop is different from using [Stop] button. [Reset] button terminates machining with E-stop, the state of air pump will not change, and there is no deceleration and cutter uplift.

Hotkey: [PauseBreak], available under any mode.

◆ [Start Simulate]

After loading a machining file, pressing the button , the system will enter into simulation mode. Clicking the button  will exit from simulation mode.

◆ [BackWorkOrg]

Pressing the button , the system will return to workpiece origin automatically. Please note the difference among workpiece origin, fixed point and machine origin (the reference point).

◆ [Auto Back, Auto Forward]

If the user wants to return to a certain previous machining position, he needs to select [Auto Back] to make the system move against the original machining path. This function is scarcely used, so the user just needs to know it.

[Automatic Mode] section includes buttons frequently used in auto machining, which offers the user an interactive environment for operating a machine tool automatically.

◆ [FeedRate]

The user can control feed speed by controlling current feedrate override. The formulation is as following:

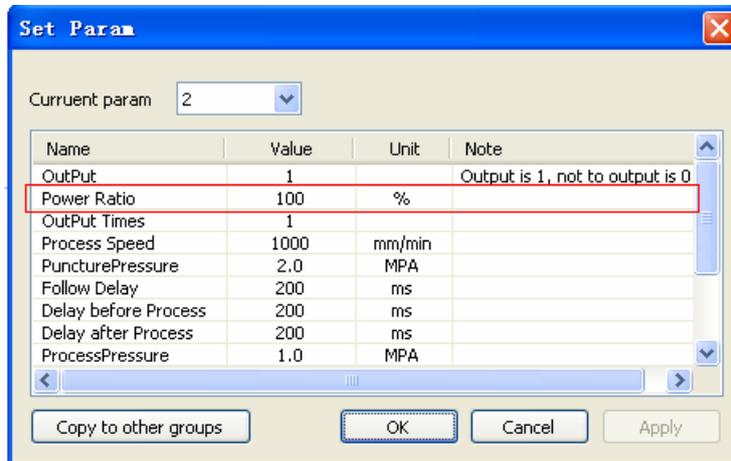
Current feed speed = rated feed value * current feedrate override.

Current feedrate override is displayed beneath the word “FeedRate”. Three ways can be used to revise feedrate override. Click on the sliding axle; Drag the slide block; Click the slide block to choose it, and then press [PageUp], [PageDown] key; after [PageUp] or [PageDown] key is pressed each time, the feedrate override increases (decreases) 10%. The range of Feedrate override is 0% ~ 120%.

◆ [LaserPower]

Adjust the laser power by dragging the scroll bar; machining power is determined by laser power and layer power simultaneously.

Layer power: double click on a color bar to eject a dialog which includes power ratio, as following:



Laser power in machining = layer power * laser power.

Note that laser power is adjustable, while layer power is nonadjustable during machining.

◆ [Enactment Value]

The dialogue, as Fig. 4-87, will pop up after this button is pressed, including settings of machining speed and G00 speed under [Auto Mode].

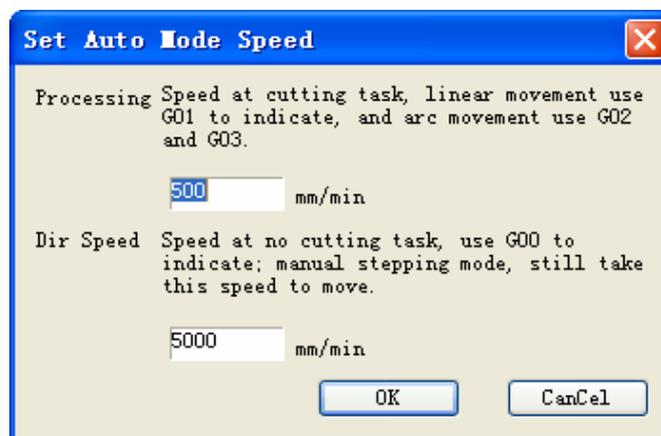


Fig. 4-87 [Set Auto Mode Speed]

Processing speed refers to the moving speed under G01, G02 and G03 command. Its range is 1~180000. If out of range, the system will give a prompt.

G00 speed refers to the running speed under G00 command, and its range is 1~99999.999. If out of range, the system will give a prompt.

◆ [Instant Value]

This function is used to display the actual machining speed currently.

4.8.2. Manual Mode

The overall machining window under Manual mode is as Fig. 4-88.

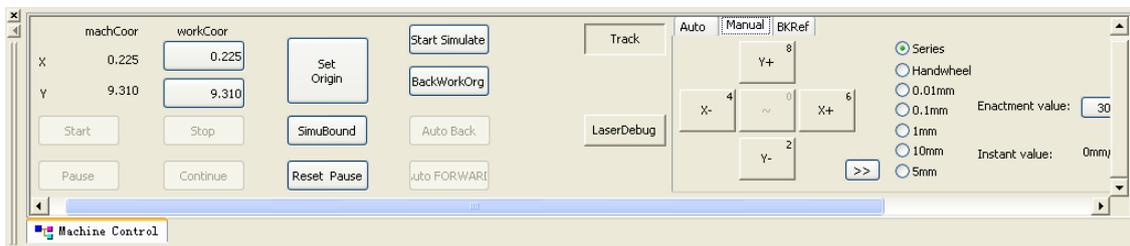


Fig. 4-88 [Machine Tool Control] under [Manual Mode]

The left part of [machine control] window is the same as that under [Auto Mode].

However, the following buttons are disabled under [Manual Mode], for instance, [start], [pause], [continue], [stop], etc.

The following buttons are included on the right part of [machine control].

◆ [X+], [X-] manual button, shown as:



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

◆ [Y+], [Y-] manual button, shown as:



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

◆ [~] manual button, shown as:



Acceleration button, only available under [Series (Jog)] mode; it corresponds to the number key “0” in mini-keyboard and can be used together with one of X, Y manual buttons to accelerate the

speed of tool; Combination key: “0 and a manual button”, for instance, “0” and “8” (acceleration towards “+Y”).

◆ **[Series], [Handwheel], [0.01], [0.1], [1], [10], [>>] manual buttons, as Fig. 4-89.**



Fig. 4-89 Buttons on manual operation panel

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

Jog manner: after pressing down the button, the user can manually make a machine tool move continuously. Under MANUAL mode (the function screen is current active window), clicking a direction key on the panel or pressing down the corresponding shortcut key on mini-keypad will make the machine tool move continuously. Loosening it will stop the machine tool.

Note:

Manual screen must be active, which can be distinguished by 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 a handwheel (MPG—Manual Pulse Generator), in V9 NcStudio, we add the button [Handwheel]. Firstly, the user should install a handwheel and then press down the button, and then he can manipulate the machine tool by handwheel. At this time, the X, Y manual buttons are invalid.

Stepping (increment) manner (0.01, 0.1, 1, 10, >>): it is also called stepping-manner or increment-manner. This manner can control the machine’s feed distance more accurately compared with jog-manner. The user can click a button on interactive panel to adjust the increment; every time the user triggers one of the X, Y manual buttons, the machine will move a specified step-size along the corresponding direction. When one of the buttons [0.01], [0.1], [1], [10], [>>] is pressed, the increment stepping mode is activated. And increment stepping length equals to the selected step-length. For instance, if [0.1] button is selected, increment stepping length is 0.1 mm. If the button [>>] is selected, a dialogue will pop up; after the user inputs the specific step-size, this value will be displayed near the button [>>]. For instance, after the user inputs “500.0” into the dialogue, the step-size will be adjusted to “500.0”; and once the button [500.0] pressed, each time clicking one of the X and Y manual buttons will make the machine move 500.0mm along the corresponding direction.

◆ **[Enactment Value]**

A dialogue, as Fig. 4-90, will pop up after this button is pressed, including setting of normal jog and rapid jog under [Manual Mode].

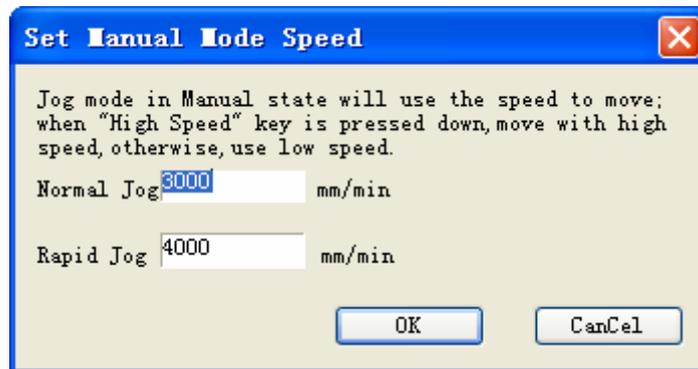


Fig. 4-90 Setting [Manual Mode] speed

Normal jog speed refers to the manual moving speed when the manual high-speed key is not pressed, with setting range as 1 mm/min ~ rapid jog speed;

Rapid jog speed is the manual running speed when the manual high-speed key is pressed, with setting range as normal jog speed ~ G00 speed.

4.8.3. BKRef Mode

The overall machining window under Reference point mode is as Fig. 4-91.

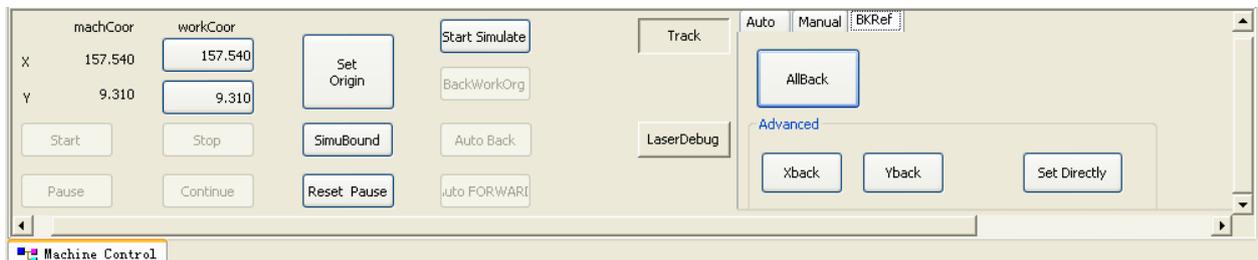


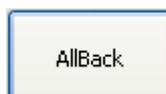
Fig. 4-91 [Machine Tool Control] window under [Reference Point] mode

The left part of [machine control] window is the same as that under [Auto Mode].

However, some buttons are disabled under [Reference Point Mode], e.g., [start], [pause], [continue], [stop], [BackWorkOrg], etc.

The following buttons are included on the right part of [machine control].

➤ [All Back]



The function of this button is to make X, Y axis back to machine origin simultaneously.

Machine origin (home, reference point) is the zero point of machine coordinates system, fixed on a machine tool, which is positioned by both mechanical switch and electrical system. Regarding the

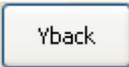
machine coordinate system, please refer to chapter 3.2. Operation of “back to reference point” requires the support of machine hardware, so some type of machine doesn’t have this function. Reference point is the benchmark of the whole machine, so it is used to calibrate the coordinates of the current point.

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

➤ [X Back]

Pressing the button , the system will control the X-axis to go back to the machine origin. When the operation is finished, in the mini-coordinate display zone of operation panel and in [Pos] function screen, there will be a mark (*) of completing “back to machine origin” after X-axis.

➤ [Y Back]

Just like [X Back] button, pressing the button , the system will control the Y-axis to go back to the machine origin. When the operation is finished, in the mini-coordinate display zone of operation panel and in [Pos] function screen, there will be a mark (*) of completing “back to machine origin” after Y-axis.

➤ [Set Directly]

If the user can make sure the current position is consistent with its machine coordinates and there is no abnormal state, such as “E-stop”, etc, he can click the button  to directly set the current machine coordinate as the accurate machine coordinate. When the operation is finished, in the mini-coordinate display zone of operation panel and in the [Pos] function screen, there will be a mark (*) of completing the operation of backing to machine origin after both X and Y axes.

5. Parameters of NcStudio in Laser Cutting

With abundant machining 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: general parameters and laser cutting parameters. General parameters can be subdivided into operators' parameters, manufacturers' parameters and developers' parameters based on different permissions; laser cutting parameters are specially configured for different cutting machine tools. To look over or modify some types of parameters, the user must have enough jurisdictions.

5.1. Methods for Parameters Modification

Firstly, use the mouse, or press 'Tab' or 'Shift + Tab' key to move the focus onto parameter list box, and then press a direction key (up and down direction keys) to choose the parameter to be modified. Finally, press 'Enter' key and input value into the parameters input box, or double click the parameter line and input value into the parameters input box. Regarding parameters whose value is "true" or "false", the user can directly input "true" or "false", or directly input "1" equalling to "True", and "0" equalling to "False".

Note:

"Tab" key can be used to move the focus on the interface, while 'Tab + Shift' will move the focus reversely.

In the parameter list box, only part of parameter information can be displayed. Following two methods will describe how to look over the complete content of a parameter: look at the lower part of the parameter interface, this window will show the complete information of the parameter selected (recommended); press direction key (left or right direction key) in the parameter list box to look over the whole information about the parameter selected.

Modification to all parameters must be done under IDLE state.

5.2. Parameter List of Manufactures (Including Operators)

The operators' parameters are as shown in Fig. 5-1.

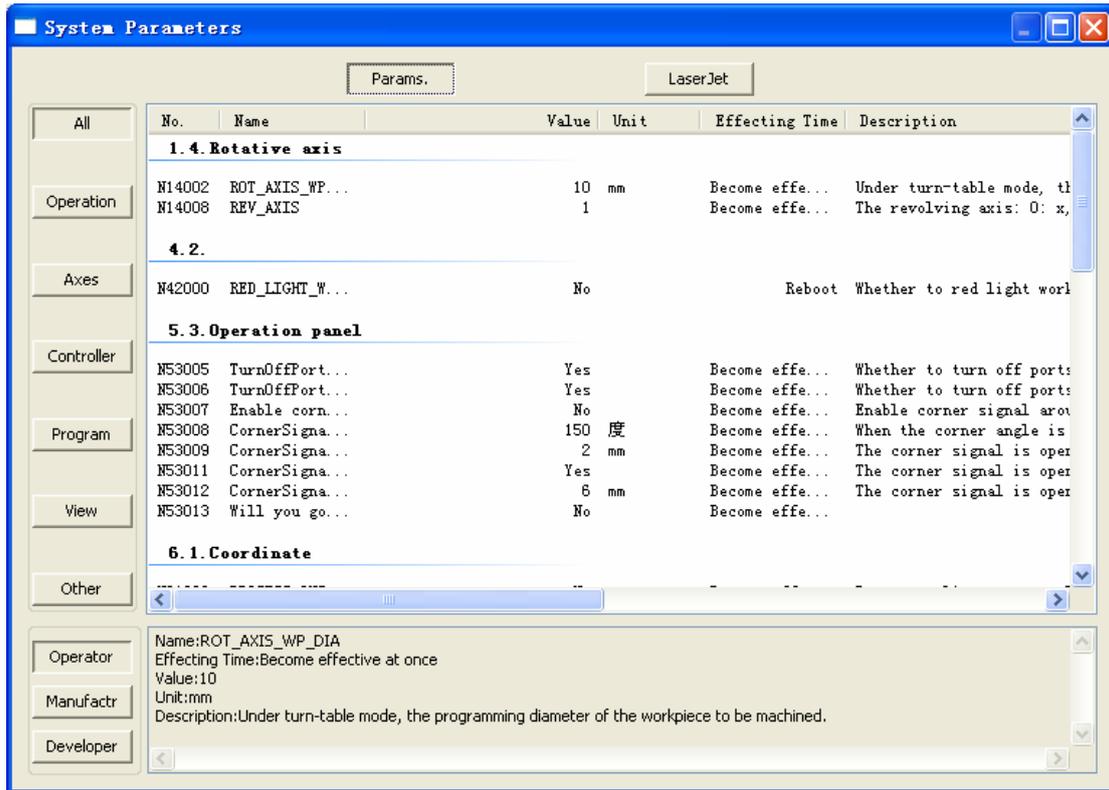


Fig. 5-1 Operators' parameters

Manufacturers' parameters mainly consist of operators' parameters and ex-factory parameters, as shown in Fig. 5-2.

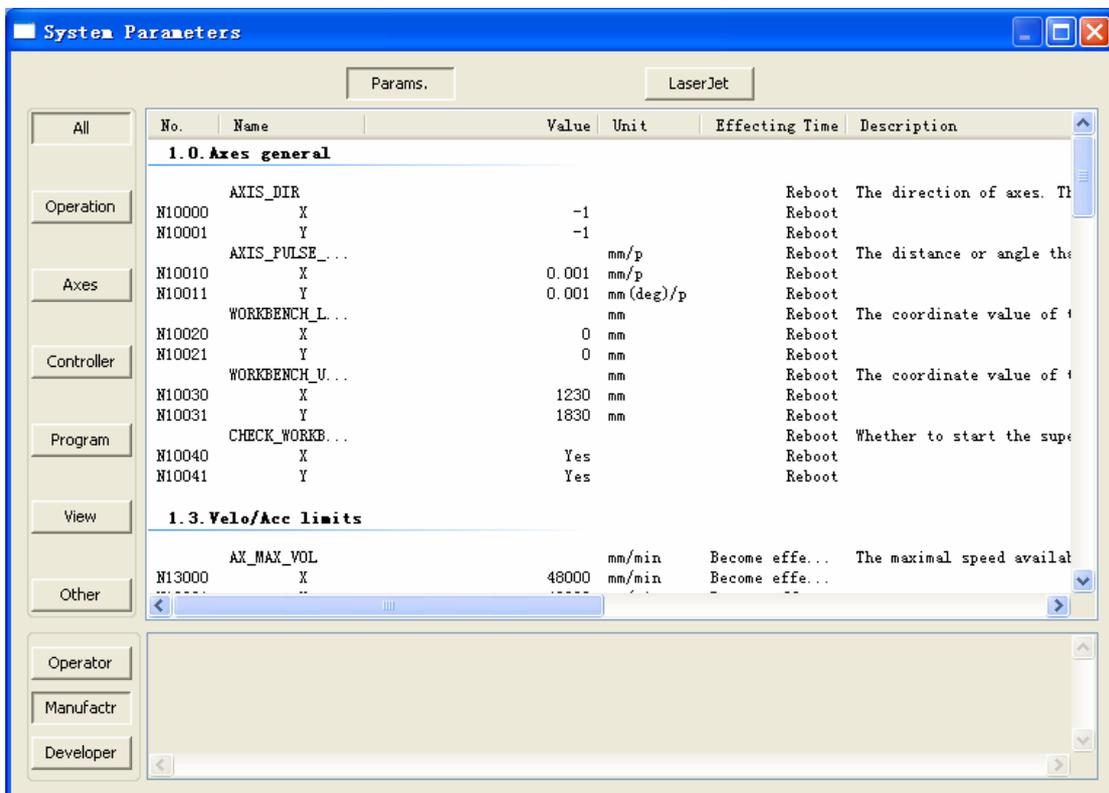


Fig. 5-2 Manufacturers' parameters

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

The user can review parameters in terms of parameter No., according to which all the following parameters of operators and manufacturers are listed.

5.2.1. General Parameters

1.0. Axes general

◆ N10000~N10001 Axis_Dir (X|Y)

Type:	INT
Unit:	None
Range:	1: Positive Direction -1: Negative Direction
Default:	-1
Valid time:	After restart
Instruction:	It specifies the incremental direction of mechanical coordinates of an axis: "1" indicates "positive increment", and "-1" indicates "negative increment".

◆ N10010~N10011 Axis_Pulse_Factor (X|Y)

Type:	Float
Unit:	mm/p (millimeter/pulse)
Range:	0.0000009~999.0
Default:	0.001
Valid time:	After restart
Instruction:	It refers to the displacement or angle on the related axis generated 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. According to the transmission connection, the angle displacement of

	the stepping pulse will be converted into linear value.
--	---

◆ N10020~N10021 Workbench_Lower_Limit (X\Y)

Type:	Float
Unit:	mm (millimeter)
Range:	-99999.0~99999.0
Default:	X:0.0 Y:0.0
Valid time:	After restart
Instruction:	It refers to the lower limit of the worktable stroke on the direction of each axis.

◆ N10030~N10031 Workbench_Upper_Limit (X\Y)

Type:	Float
Unit:	mm (millimeter)
Range:	Workbench lower limit(machine coordinate)~99999.0
Default:	X: 1230.0 Y: 1830.0
Valid time:	After restart
Instruction:	It refers to the upper limit of the worktable stroke on the direction of each axis.

◆ N10040~N10041 Check_Workbench_Range (X\Y)

Type:	BOOL
Unit:	None
Range:	0 (False): Disabled. 1 (True): Enabled.
Default:	1 (True)
Valid time:	After restart

Instruction:	This parameter checks if current object is out of machining stroke, if it is set as "True", when the general circumscribed rectangle of objects is out of machining stroke, after clicking "Start", machining will be forbidden, and alarm occurs. If it is set as "False", this checking will be avoided.
---------------------	--

1.3. Velo/ Acc limits

◆ N13000~N13001 AX_MAX_VOL (X|Y)

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.01~100000.0
Default:	48000.0
Valid time:	Immediately
Instruction:	It specifies the maximum allowable velocity of each axis.

1.4. Rotative Axis (Rotary Table)

◆ N14006 ROT_AXIS_WP_DIA

Type:	Float
Unit:	mm (millimeter)
Range:	1.0 ~ 1300.0
Default:	2.5
Valid time:	Immediately
Instruction:	It specifies the external programming radius of the cylinder to be processed under rotary table mode.

◆ N14007 Tool Height

Type:	Float
Unit:	mm (millimeter)

Range:	0.0 ~ 200.0
Default:	2.5
Valid time:	Immediately
Instruction:	It is the tool height of the cylinder under rotary table mode.

4.2.

◆ N42000 Red_Light_When_Finish

Type:	BOOL
Unit:	None
Range:	0 (False): No prompt 1 (True): Prompt
Default:	0 (False)
Valid time:	After restart
Instruction:	It sets whether red light prompt is adopted in ending processing until external input, generated via mouse or keypad, is received.

5.3. Operation Panel

◆ N53005 Turn off Ports When Tracking Back

Type:	BOOL
Unit:	None
Range:	0 (False): Remain unchanged 1 (True): Shut down open ports
Default:	1 (True)
Valid time:	Immediately
Instruction:	During processing, if "Stop" and "Auto back" are clicked sequentially, this parameter sets whether to change status of some ports under [auto back]

	mode; "True" indicates shutting down open ports, and "False" equals to remaining unchanged. The variety of "Some ports" will vary with different needs.
--	---

◆ **N53006 Turn off Ports When Tracking Forward**

Type:	BOOL
Unit:	None
Range:	0 (False): Remain unchanged 1 (True): Shut down open ports
Default:	1 (True)
Valid time:	Immediately
Instruction:	Its function is similar to "Turn Off Ports When Tracking Back".

◆ **N53007 Enable Corner Signal around Corner**

Type:	BOOL
Unit:	None
Range:	0 (False): Don't open corner signal around corner 1 (True): Open corner signal around corner
Default:	0 (False)
Valid time:	Immediately
Instruction:	In cutting objects, deceleration is needed when the corner is encountered, so it is necessary to deal with object corner specially. As a result, the software should offer a signal to mark the corner of current machining object and be used as additional control, e.g. decreasing laser tube power output.

◆ **N53008 Corner Signal Angle**

Type:	Float
--------------	-------

Unit:	deg (degree)
Range:	0 ~180
Default:	150
Valid time:	Immediately
Instruction:	This parameter is subject to “Enable Corner Signal around Corner”; to enable N53007, make sure what is corner first, like how much degree the corner angle will be. Or is it a corner angle when it is not a straight line? Of course not, corner angle, defined in this parameter, is when the included angle of two lines is less than this set value.

◆ **N53009 Corner Signal Distance**

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~100.0
Default:	2
Valid time:	Immediately
Instruction:	It is also subject to “Enable Corner Signal around Corner”; first fix the corner angle degree according to parameter “Corner Signal Angle”. And then this parameter determines the distance away from corner angle for opening corner signal.

◆ **N53011 Corner Signal Distance**

Type:	BOOL
Unit:	None
Range:	0 (False): Don't continue retracting in breakpoint 1 (True): Continue retracting in breakpoint

Default:	1 (True)
Valid time:	Immediately
Instruction:	If [Stop] and [Continue] are clicked sequentially during machining, the stop position may not be cut thoroughly since it takes time to increase laser tube power if a machine tool continues machining from this position. To avoid this kind of problem, the machine tool will retract a certain distance from the breakpoint and then continue machining.

◆ N53012 Corner Signal Distance

Type:	Float
Unit:	mm (millimeter)
Range:	0.0~100.0
Default:	6
Valid time:	Immediately
Instruction:	This parameter is subject to N53011 "Continue retracting in breakpoint". If N53011 is set as "True", the retracting distance is fixed by this parameter.

◆ N53013 Go Forward /Backward Fixed Distance

Type:	BOOL
Unit:	None
Range:	0 (False): Don't go forward/backward fixed distance 1 (True): Go forward/backward fixed distance
Default:	0 (False)
Valid time:	Immediately
Instruction:	Whether to go specified distance in breakpoint resuming

6.1. Coordinate

◆ N61200 Process_Once_When_Repeated

Type:	BOOL
Unit:	None
Range:	0 (False): Process more than once 1 (True): Process only once
Default:	0 (False)
Valid time:	Immediately
Instruction:	When there is repetition part in two objects, this parameter sets whether the repetition side will be processed only once. Note that repeated cutting can lead to widening slot, affecting cutting effect.

6.4. Speed/ Acc

◆ N64000 AX_Startup_Vol

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.0 ~ 100000.0
Default:	0
Valid time:	Immediately
Instruction:	The parameter sets the minimum feed rate during machining.

◆ N64020 Rapid_Travel_Feedrate

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	Within the max. allowable speed of hardware
Default:	5000.0
Valid time:	Immediately

Instruction:	It sets default running speed (not processing speed) of a machine tool during positioning (under G00 command).
---------------------	--

◆ N64040 Default_Feedrate

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.0~100000.0
Default:	500
Valid time:	Immediately
Instruction:	It is the default processing speed (not the positioning speed) of a machine tool.

◆ N64100 AX_Linear_Mach_Acc

Type:	Float
Unit:	mm/s ² (millimeter/ second ²)
Range:	0.001~100000.0
Default:	300.0
Valid time:	Immediately
Instruction:	It describes the acceleration deceleration capacity of a single axis, depending on the physical features of a machine tool. The larger the value is, the higher the efficiency is. But if the value is too large, abnormal situation may occur.

◆ N64101 AX_Linear_Post_Acc.

Type:	Float
Unit:	mm/s ² (millimeter/ second ²)
Range:	0.001~100000.0
Default:	300.0

Valid time:	Immediately
Instruction:	It specifies the max. acceleration of a single axis in the process of machine positioning.

◆ N64102 Z_LINEAR_ACC

Type:	Float
Unit:	mm/s ² (millimeter/ second ²)
Range:	0.001~100000.0
Default:	400
Valid time:	Immediately
Instruction:	It specifies the max. acceleration of Z axis.

◆ N64120 AX_Con_Acc

Type:	Float
Unit:	mm/s ² (millimeter/ second ²)
Range:	0.001~100000.0
Default:	1000.0
Valid time:	Immediately
Instruction:	It describes the maximum acceleration of the feed motion occurring on adjacent axes. 1~2 times acceleration of a single axis is recommended.

◆ N64150 AX_Acc_Acc

Type:	Float
Unit:	mm/s ³ (millimeter/second ³)
Range:	0.001~100000000000.0
Default:	10000.0
Valid time:	Immediately
Instruction:	It sets growth rate of acceleration (acceleration's

	acceleration) of a single axis.
--	---------------------------------

◆ N64203 INTERP_ALGO

Type:	INT
Unit:	None
Range:	0, Echelon Algorithm 1, Type S Algorithm 2, Acceleration Trapezoid Algorithm
Default:	1
Valid time:	Immediately
Instruction:	To decrease error, please choose the most suitable interpolation algorithm after debugging.

◆ N64208 Ref_Circle_Max_Velo

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.001~100000.0
Default:	1800
Valid time:	Immediately
Instruction:	<p>For arc speed limit. When a circle is processed, vibration occurs due to centripetal force in circular motion. To reduce vibration, machining speed is limited in circle machining in terms of centripetal acceleration. Take default setting as an example, the diameter of reference circle is 10mm, and the max. line speed is 1800mm/min. According to the formula</p> $a = \frac{v^2}{r}$ <p>, $r = (10/2)mm$, $v = 1800mm/min$, the centripetal acceleration a will be calculated. When other arcs are processed, this calculated a will be the max. allowable centripetal acceleration. The</p>

	speed of an arc will be limited when the centripetal acceleration caused by too large arc speed is larger than this value.
--	--

◆ **N64209 Circle_Min_Velo**

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.001~100000.0
Default:	180.0
Valid time:	Immediately
Instruction:	<p>This parameter is also corresponding to arc speed limit. The calculated centripetal acceleration a in terms of the above formula is not the only criterion.</p> $a = \frac{v^2}{r}$ <p>According to the formula $a = \frac{v^2}{r}$, the arc speed limit will result in too small processing speed if an arc is too small, which will be a waste of time. To ensure processing efficiency, when the speed figured out by the system is lower than the setting value of this parameter, the system will take this setting value as the processing speed, no matter how small the radius is.</p>

◆ **N64210 Enable_MCM**

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default:	0 (False)
Valid Time:	Immediately
Instruction:	It specifies whether the system will process steadily.

	When this parameter is set as “Yes”, a machine tool will become more stable during processing, but the processing speed will slow down.
--	---

◆ N64211 Cur_Eff_Scheme_Index

Type:	INT
Unit:	None
Range:	0-4
Default:	0
Valid Time:	Immediately
Instruction:	It specifies the current efficiency strategy group.

6.5. File Translation

N65106 DXF Format Convert Param

DXF file translation parameters are used for the translation of DXF files.

◆ N65106 Force To Recognize Dxf File As Metric Sys

Type:	BOOL
Unit:	None
Range:	0 (False): Don't recognize as metric size 1 (True): Recognize as metric size
Default:	1 (True)
Valid time:	After loading a processing procedure again.
Instruction:	It sets whether to regard DXF files as metric size forcibly.

7.1. Manu

◆ N71000 Jog_Vol.

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

Range:	Within the max. allowable speed of hardware
Default:	3000
Valid time:	Immediately
Instruction:	It sets the default running speed of system when a manual button is pressed under manual mode.

◆ N71001 Rapid_Jog_Vol

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	Within the maximum allowable speed of each axis
Default:	4000
Valid time:	Immediately
Instruction:	It specifies the system default running speed when “a manual button and the acceleration key” are pressed simultaneously under manual mode.

◆ N71020 Lock_Dir_After_Limited

Type:	BOOL
Unit:	None
Range:	0 (False): Not prohibit 1 (True): Prohibit
Default:	1 (True)
Valid time:	After restart
Instruction:	It specifies whether to prohibit the spindle from moving towards limit direction when limit occurs. If limit occurs, a machine tool should be allowed to move towards no limit direction, and it will be damaged if moving towards limit direction during limit.

◆ N71021 Jiggle_Vol

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	Less than the maximum allowed speed of hardware
Default:	1200
Valid time:	Immediately
Instruction:	It sets the jiggle velocity under auto machining mode.

7.2. Auto

◆ N72001 Speed_Assign_Type

Type:	INT
Unit:	None
Range:	0: use the file speed 1: use the default speed 2: specify the speed according to ratio
Default:	0
valid time:	Immediately
Instruction:	It sets the selection style of feed speed. "0: use the file speed"; "1: use the default speed"; "2: specify the speed according to ratio".

◆ N72010~N72011 Enable_Workarea_Check (X/Y)

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default:	0 (False)
Valid time:	Immediately
Instruction:	It sets whether to check workpiece coordinate range, consisting of upper & lower limit of the coordinate.

◆ N72020~N72021 Workarea_Lower_Limit (X|Y)

Type:	Float
Unit:	mm (millimeter)
Range:	-99999.0~99999.0
Default:	-10000.0
Valid time:	Immediately
Instruction:	It specifies the lower limit of workpiece coordinates on each direction.

◆ N72030~N72031 Workarea_Upper_Limit (X|Y)

Type:	Float
Unit:	mm (millimeter)
Range:	Lower limit of workbench stroke (machine coordinate)~ 99999.0
Default:	10000.0
Valid time:	Immediately
Instruction:	It sets the upper limit of workpiece coordinates on each direction.

7.4. Bkref

◆ N74000 Cancel_REFPT_After_Estop

Type:	BOOL
Unit:	None
Range:	0 (False): Don't cancel 1 (True): Cancel
Default:	0 (False)
Valid time:	Immediately
Instruction:	If it is set as "True", when E-stop occurs, the symbol of backing to the reference point will be cleared.

◆ N74001 Need_REFPT_Before_Machining

Type:	Bool
Unit:	None
Range:	0 (False): No need to back to the reference point 1 (True): Need to back to the reference point
Default:	0 (False)
Valid time:	Immediately
Instruction:	When servo system is equipped with an incremental coding machine tool, the system can only know the relative position with respect to the previous one. In order to fix the coordinate of an arbitrary point on the machine tool, a reference point is needed, which is the machine origin. During processing, if ESTOP or power failure occurs, the displacement may appear in resuming machining. Please use "back to machine origin" to retrieve the WCS so as to ensure the accuracy of continuing processing.

◆ N74010~N74011 REFPT_Coor (X|Y)

Type:	Float
Unit:	mm (millimeter)
Range:	Lower limit of workbench stroke(machine coordinate) ~ Upper limit of stroke (machine coordinate)
Default:	0
Valid time:	After restart
Instruction:	The machine coordinates of machine origin is fixed in terms of setting of [lower limit of workbench stroke] and [upper limit of workbench stroke].

◆ N74091 Double Y Encoder Origin Error

Type:	Float
--------------	-------

Unit:	None
Range:	-10000.0~10000.0
Default:	0
Valid time:	Immediately
Instruction:	

7.9. Operation others

◆ N79110~N79111 FIXPT_Pos (X|Y)

Type:	Float
Unit:	mm (millimeter)
Range:	-99999.0~99999.0
Default:	0
Valid time:	Immediately
Instruction:	It is enabled only when [Auto back to the Fixed point valid after processing] parameter is valid, indicating the machine coordinates of the fixed position.

◆ N79200 Display Simu Out Range

Type:	BOOL
Unit:	None
Range:	0 (False): Don't clew 1 (True):Clew
Default:	1 (True)
Valid time:	Immediately
Instruction:	It specifies whether to clew "out of mechanical travel" in simulation mode.

8.1. Position View

◆ N81000 Auto_Load_Track

Type:	BOOL
Unit:	None
Range:	0 (False): Invalid 1 (True): Valid
Default:	1 (True)
Valid time:	Immediately
Instruction:	After a file is loaded, it sets whether to analyze the processing track automatically to facilitate loading the track in advance.

◆ **N81001 Auto_Load_Track_Limit**

Type:	INT
Unit:	KB
Range:	0~100000
Default:	1000
Valid time:	Immediately
Instruction:	Only when a processing file size is smaller than the value of this parameter, the system will automatically analyze the processing track; when larger, the system will not analyze the processing track.

8.7. Show Interface

◆ **N87000 Show Interface**

Type:	INT
Unit:	None
Range:	0: Solely open NcEditor 1: Solely open NcStudio 2: Open NcEditor and NcStudio simultaneously
Default:	0
Valid time:	After restart

Instruction:	It specifies the displaying form of the interface. And three types are available, "0: Solely open NcEditor"; "1: Solely open NcStudio"; "2: Open NcEditor and NcStudio simultaneously".
---------------------	---

5.2.2. LaserJet Parameters

9.0 Application general

◆ N90010 Delay After Laser On (Pause-Start)

Type:	Float
Unit:	ms (millisecond)
Range:	20 ~600000
Default:	300
Valid time:	Immediately
Instruction:	It sets the delay time after laser on.

◆ N90011 Delay Before Laser Off (Machining-Pause)

Type:	Float
Unit:	ms (millisecond)
Range:	20 ~600000
Default:	300
Valid time:	Immediately
Instruction:	It sets the delay time before laser off.

◆ N90020 Delay Before Cutting

Type:	Float
Unit:	ms (millisecond)
Range:	20 ~600000
Default:	300

Valid time:	Immediately
Instruction:	It sets the delay time before cutting.

◆ N90042 Open_Blow2_Delay_Time

Type:	Float
Unit:	ms (millisecond)
Range:	0 ~600000
Default:	20
Valid time:	Immediately
Instruction:	It sets the delay time to open blow 2.

◆ N90060 Up_Tool_Delay_Time

Type:	Float
Unit:	ms (millisecond)
Range:	0 ~600000
Default:	300
Valid time:	Immediately
Instruction:	It sets the delay time of tool lifting at the time of E-stop and stop.

◆ N90061 Before Close Laser Jet

Type:	Float
Unit:	ms (millisecond)
Range:	0 ~ 600000
Default:	300
Valid time:	Immediately
Instruction:	It sets the delay time before laser off at the time of E-stop and stop.

◆ N90062 After Close Laser Jet

Type:	Float
Unit:	ms (millisecond)
Range:	0 ~600000
Default:	300
Valid time:	Immediately
Instruction:	It sets the delay time after laser off at the time of E-stop and stop.

6. General Overview of Laser Engraving

Laser Engraving refers to a thermal carvings craft, which results in chemical and physical changes on the surface of materials irradiated by laser beam with high-energy-density to obtain the required designs.

The software of NcStudio, motion control system of laser engraver with copyright, is independently developed by Weihong Electronic Technology Co., Ltd. It can directly support G code, DXF format, PLT code format and ENG code of JDPaint processing files that are created by CAD/CAM applications such as UG, MasterCAM, CASMate, ArtCAM, AutoCAD, CorelDraw, and so on.

NcStudio takes great advantage of 32-bit computing and multi-tasks. Meanwhile, its standard interface is dependable and user-friendly.

Apart from functions of automatic mode, manual mode and “back to reference point”, this control system features the functions as following: simulation, processing time forecast, loading processing track in advance, dynamic tracking, breakpoints resuming (program block skip function), etc.

Note:

Regarding system installation, functions & features of NcEditor and parameters of NcStudio of laser engraving, they are almost identical with those of laser cutting, please refer to Chapter 1 ~5 for detail; but some functions are exclusively held in laser engraving software with a different interface, which will be introduced in detail as following.

7. Features of NcEditor in Laser Engraving

7.1. Main Interface

The integral interface of NcEditor includes: [title bar], [menu bar], [toolbar], [drawing toolbar], [status bar], [machining control bar], [object editor space] and [ruler bar]. See Fig. 7-1.

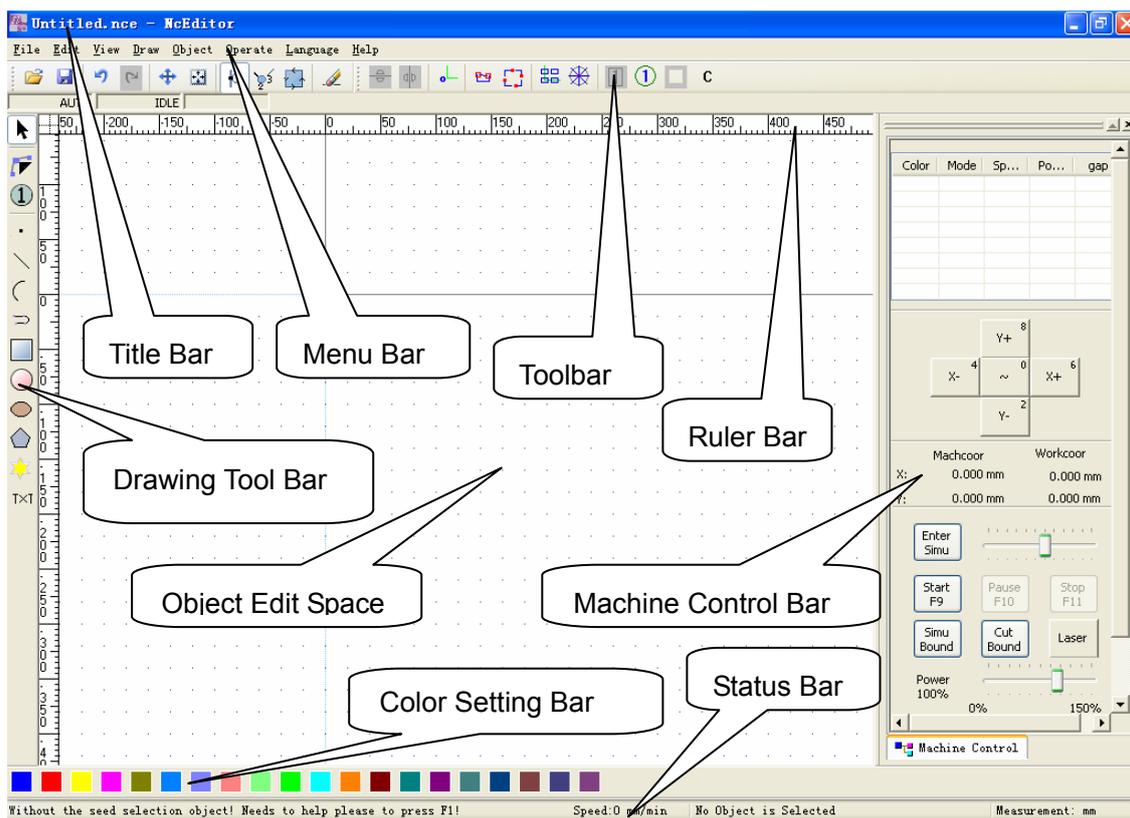


Fig. 7-1 Integral interface

7.2. Machine Control Bar

The machine control bar is on the right side of NcEditor, as shown in Fig. 7-2.

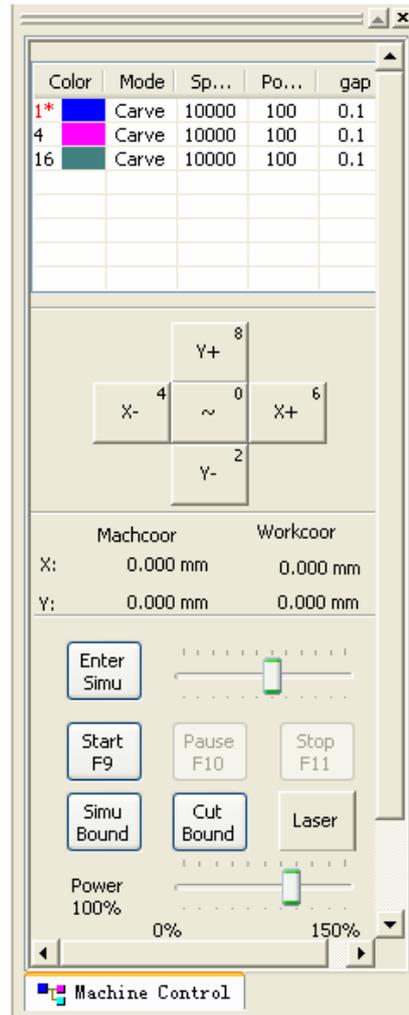


Fig. 7-2 Machine control bar

Gap: it is the line space under carve mode, unavailable under cut mode.

7.2.1. Color Window of Machining Parameters

The upper part of Fig. 7-2 is the color window of machining parameters. In two-dimensional mode of NcEditor, machining objects with the same color are regarded as holding the same value of machining parameters; thus, if the value of a machining parameter of an object in a certain color is altered, the value of this machining parameter of all objects in this color will be changed accordingly.

The parameter attribute window is under Cut mode after software start-up, which is out of regard for compatibility between laser cutting and engraving. Clicking the word “Cut” will switch to Carve mode. In addition, different machining speed and power can be set for laser engraving and cutting, as shown in Fig. 7-3.

Setting the color of a machining object is realized by [Color setting bar] above the status bar. Select a machining object, and click any color of [Color setting bar] (as shown in Fig. 7-4) to change the object color to the selected color.

Color	Mode	Speed	Po...	gaj
1*	Carve	10000	100	0.1
4	Carve	10000	100	0.1
16	Cut	1000	100	

Color setting bar

Fig. 7-3 Color window of machining parameters

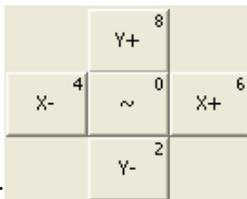


Fig. 7-4 Color setting bar

7.2.2. Machine Control Section

The lower part of Fig. 7-2 is machine control section, including “X\Y manual button”, “X\Y coordinate value”, “power adjusting slider”, and “start”, “pause”, “stop”, “enter simu”, etc.

◆ X\Y manual button



See:

They correspond to positive and negative motion directions of X-, Y-axis, with shortcut keys “6, 8, 4, 2” accordingly on mini-keypad.



Among them, [~] is the acceleration button , it corresponds to the number key “0” in mini-keyboard and can be used together with a manual button to accelerate the speed of tool; Combination key: “0 and a manual button”, for instance, “0” and “8” (acceleration towards “+Y”).



Double clicking  can be used to switch between sequence (jog) mode and incremental stepping mode.

Jog manner: after pressing down the button, the user can manually make a machine tool move continuously. Under MANUAL mode (the function screen is current active window), clicking a direction key on the panel or pressing down the corresponding shortcut key on mini-keypad will make the machine tool move continuously. Loosening it will stop the machine tool.

Note:

Manual screen must be active, which can be distinguished by 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.

Stepping (increment) manner (0.1, 1, 10, >>): it is also called stepping-manner or increment-manner. This manner can control the machine's feed distance more accurately compared with jog-manner. The user can click a button on interactive panel to adjust the increment; every time the user triggers one of the X, Y manual buttons, the machine will move a specified step-size along the corresponding direction. When one of the buttons [0.1], [1], [10], [>>] is pressed, the increment stepping mode is activated. And increment stepping length equals to the selected step-length. For instance, if [0.1] button is selected, increment stepping length is 0.1 mm. If the button [>>] is selected, a dialogue will pop up; after the user inputs the specific step-size, this value will be displayed near the button [>>]. For instance, after the user inputs "500.0" into the dialogue, the step-size will be adjusted to "500.0"; and once the button [500.0] pressed, each time clicking one of the X and Y manual buttons will make the machine move 500.0mm along the corresponding direction.

◆ **X/Y coordinate value**

See:

	Machcoor	Workcoor
X:	56.558 mm	56.558 mm
Y:	0.000 mm	0.000 mm

It displays the machine coordinates and workpiece coordinates of X-, Y-axis.

◆ **[Enter Simu]**

This button will run simulation operation for machining objects; Press [Enter Simu]  to enter simulation mode; at the same time, this menu changes to "Exit Simu". After finishing simulation, a machining file information dialogue will pop up, as shown in Fig. 7-5, displaying information like "Gxx Time", "G00 Time", "Gxx Length", "G00 Length" and "Total Figure" for the user to learn machining information.

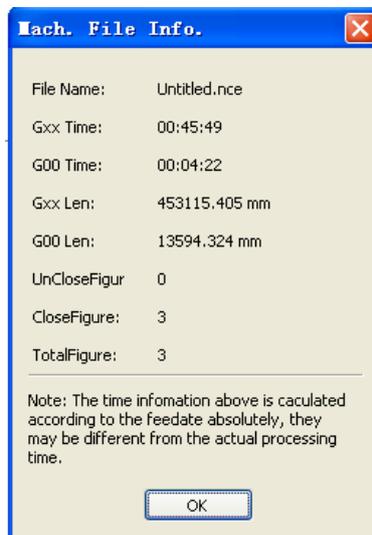


Fig. 7-5 Machining file information

◆ [Start]



Hotkey: F9. If there is no loaded processing file, the button is disabled; when a processing file is loaded, the button is enabled. Pressing down the button, the system will start machining automatically from the first line of file.

The button has two functions:

- If a certain processing procedure has been loaded and “back to reference point” has been finished, and the system is under “idle” state currently, pressing the button, the machine tool will carry out the procedure automatically from the first section. Once machining starts, the system will enter into “Auto | Running” state.
- If the system is under “Auto | Pause” state, pressing the button, the system will resume machining from the suspended position and enter into “Auto | Running” state.

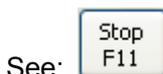
◆ [Pause]



[Pause] button is valid under auto processing. Pressing the button, the machine tool will pause and lift the cutter and then enter into “Auto | Pause” state. To resume machining, press the button [Start] or choose the relative menu item.

Hotkey: F10.

◆ [Stop]



This function is enabled during machining. Pressing the button, the machine tool will stop and uplift the cutter to terminate all the machining tasks, then the system enters into “Auto | Idle” state. This button is to break off machining procedure normally during machining.

Hotkey: F11.

◆ [SimuBound]

Pressing the button  after loading a machining file, the system will move around along the general circumscribed rectangle of machining path to determine machining stroke. This function is optional.

◆ [Cut Bound]



Pressing the button **Cut Bound** after finishing machining, the system will cut off frames of the general circumscribed rectangle of machining path.

◆ **[Laser]**



Press the button **Laser** to open laser jet valve.

Under [Auto Mode], the system will open laser jet valve automatically when machining begins.

◆ **[Power override slider]**



See:

The power is regulated by controlling the current power override.

Three ways can be used to revise power override. Click the sliding axle; Drag the slide block; Click the slide block to choose it, and then press [PageUp], [PageDown] key; After [PageUp] or [PageDown] key is pressed each time, power override increases (decreases) 15%. The range of power override is 0% ~ 150%.

Just like laser cutting, laser engraving machining power is also determined by laser power and layer power simultaneously.

Color	Mode	Speed	Po...	gal
1*	Carve	10000	100	0.1

Laser power in machining = layer power * laser power.

Note that laser power is adjustable, while layer power is nonadjustable during machining.

8. Menu of NcEditor in Laser Engraving

8.1. File Menu

[File menu] includes every operation command related to a machining file, such as [New], [Open], [Import File], etc.

◆ [Import File (B)]

The shortcut key is CTRL+ B; the main function is to load a machining file of “G code/ DXF/ ENG/ PLT/ BMP” format. The method is as below (taking G code as an example):

Open or draw a machining file;

Select [Import file] under [File] menu to eject a following dialogue.

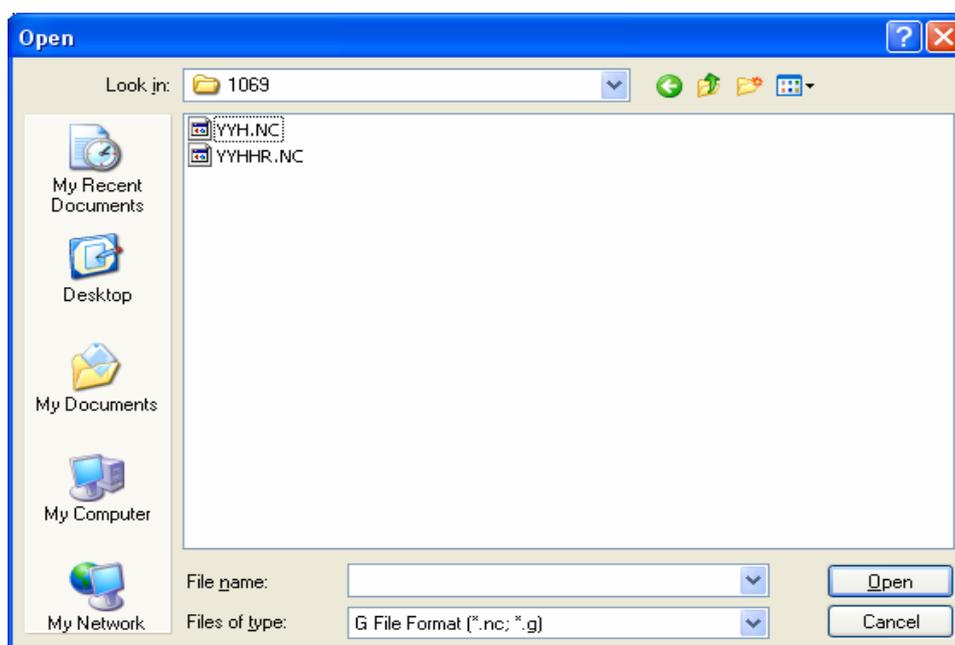


Fig. 8-1 [Import G code files] dialogue

Select the desired file and click [Open].

Note:

Currently, only monochromatic bitmap is available for our system, so the image of BMP format must be processed by Photoshop before imported.

The picture type should be black-and-white, and laser engraving can only support black point (laser on) and white point (laser off), so a grey scale image must be changed into a black and white picture.

First step: open Photoshop and load the figure→ image → mode → grayscale, as Fig. 8-2.



Fig. 8-2 [Grayscale] adjustment

Second step: image → mode → bitmap, as shown in Fig. 8-3; at this time bitmap resolution dialogue will pop up, as shown in Fig. 8-4.



Fig. 8-3 "Bitmap" mode

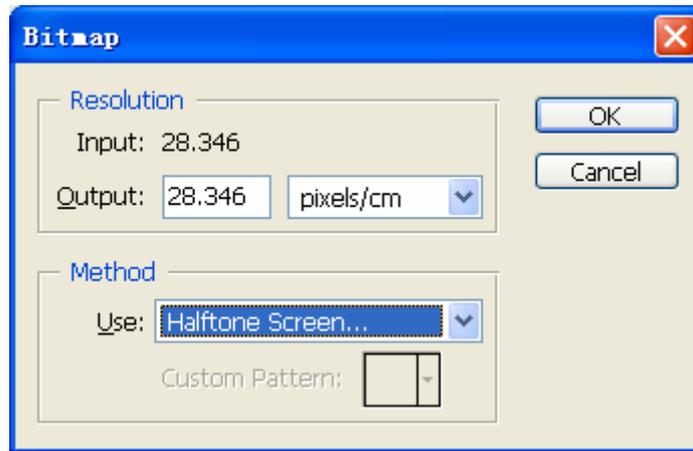


Fig. 8-4 Resolution setting dialogue

Output resolution is recommended to be less than or equal to input resolution; the higher the resolution is, the more the sampling is, the clearer the engraving is. But the resolution can not be too high; some factors must be taken into consideration, such as the size of laser beam and laser open/close frequency.

Output resolution is generally equal to input resolution.

The use method: halftone screen

Click "OK" to enter halftone screen setting dialogue, as shown in Fig. 8-5.

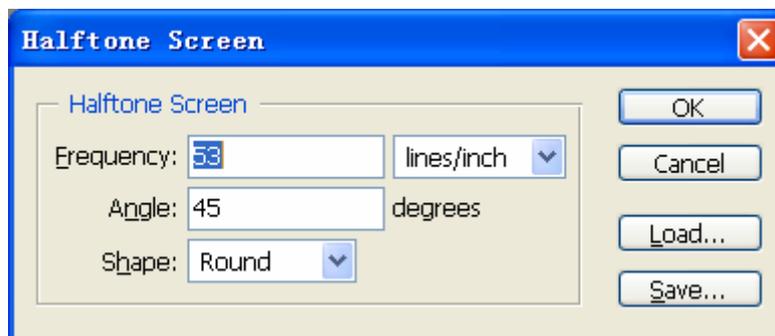


Fig. 8-5 Halftone screen

The frequency is recommended to be set as a comparable value corresponding to input

resolution of bitmap. The angle is 45°, and the shape is round; then click “OK” and save the picture as BMP format.

Temporarily, only NCE format is available for our laser engraving system, and other format files should be transformed to NCE format after loaded (e.g. “Save as” NCE format), or the system will prompt “there is no machining file loaded” after [Start] button is pressed in [machine control] section.

◆ **[Open text file], [Save selected objects as], [Recent loaded text file]**

These functions are unavailable in laser engraving.

8.2. Edit Menu

◆ **[Figure Size]**

This function is used for editing figure size, rotary angle, etc. A dialogue will pop up after this item is selected, as shown in Fig. 8-6. Input the width after edit, height after edit and angle to be rotated. Selecting “keep scale” will proportionally scale the selected figure.



Fig. 8-6 Figure size dialogue

◆ **[Edit Figure]**

This function is used for modifying figure size and position. Following dialogue will pop up after this option is selected, as shown in Fig. 8-7. Input the corresponding value in “Left X”, “Bottom Y”, “Width”, “Height”, and then click “OK” to finish modification.



Fig. 8-7 Edit figure dialogue

◆ **[Insert File]**

This function is used to insert other objects into the currently loaded file.

◆ **[Delete]**

This function is unavailable in laser engraving.

8.3. View Menu

The main options in [View] menu are: [Toolbar], [Statusbar], [Draw Toolbar], [Set Parameter], [View Translate], [Fit to Window], and [Catching Option], etc.

The functions under this menu are identical with those in laser cutting.

8.4. Draw Menu

The functions under this menu are identical with those in laser cutting.

8.5. Object Menu

The [Object] menu includes many sub-menus, such as [Delete Object], [Combine Object], [Combine Object Tolerance Setting], [Align], etc.

◆ [Reverse Direction]

Selecting this item, the machining direction of selected objects will be reversed. First click the icon  to display machining direction, and then this item is enabled.

◆ [Fill Selected]

This item is related to lead-in/out line function. Selecting this item, the type of lead-in/out line is set as external one to reserve the inner part of machining object. If not selected, the type of lead-in/out line is set as internal one to reserve the outer part of machining object with the inner part of object milled.

◆ [Reverse Color]

Choose this item to invert the color of machining objects.

Note: change the picture type to monochrome bitmap format first to achieve this item function.

8.6. Operate Menu

Operate menu includes many options, like “Start”, “Pause”, “Stop”, “Breakpoint Resume”, etc.

◆ [Back to Workpiece Origin]

Workpiece origin is the zero point of workpiece coordinates system and workpiece dimension programming; 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].

After [Back to Workpiece Origin] selected, the spindle will go back to the workpiece origin from the current position automatically.

◆ **[Set Origin]**

[Set Origin] is the same as that under [Draw] menu in laser cutting. If the user wants to set workpiece origin, he should select this menu item to eject a dialogue as Fig. 8-8.

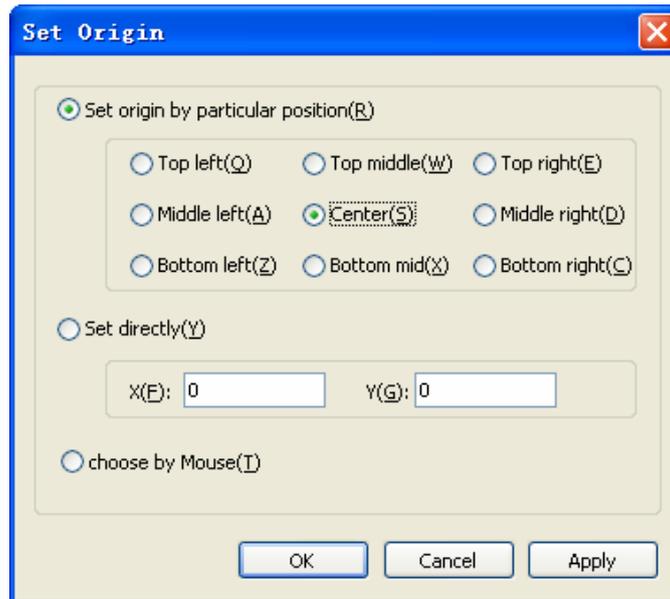


Fig. 8-8 Setting coordinate origin

There are totally three methods for option: setting workpiece origin via character point, via directly inputting coordinates of X, Y origin, or choosing origin by mouse.

8.7. Language Menu

The user can select the required language via this menu, as shown in Fig. 8-9.



Fig. 8-9 Language menu

8.8. Help Menu

The menu is used for displaying system information and registration. See Fig. 8-10.



Fig. 8-10 Help menu

◆ [About]

Selecting the menu item, the system will eject a dialog "About NcStudio". See Fig. 8-11.

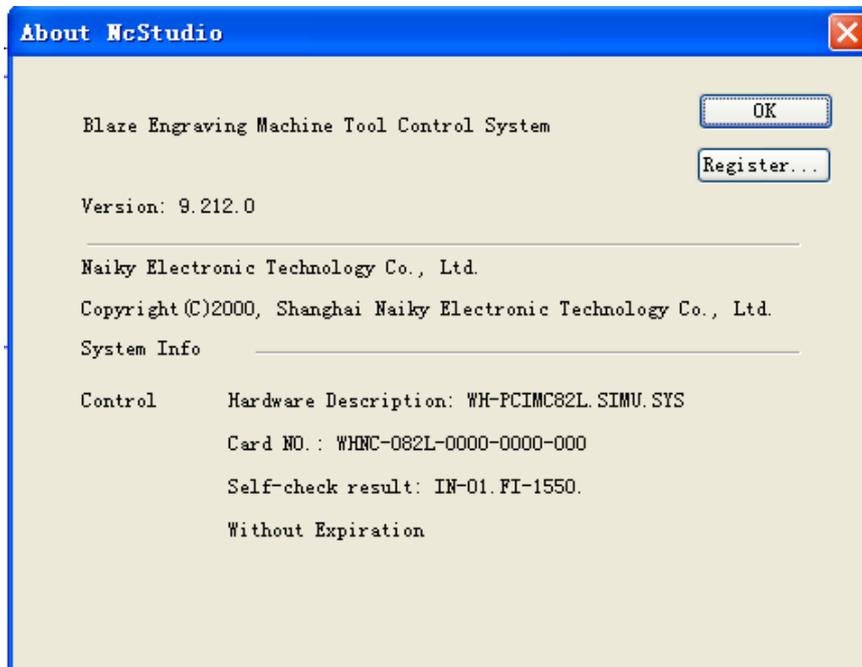


Fig. 8-11 [About NcStudio] dialogue

The user can learn information of version, control card, manufacturer, etc from the above picture.

Register function is used for limiting the system usage time. After the time is run out, send the Card No. to the machine tool manufacturer to get registration code. Then click the button [Register...], then input the received code into the following dialog, and then click [OK] to finish registration.

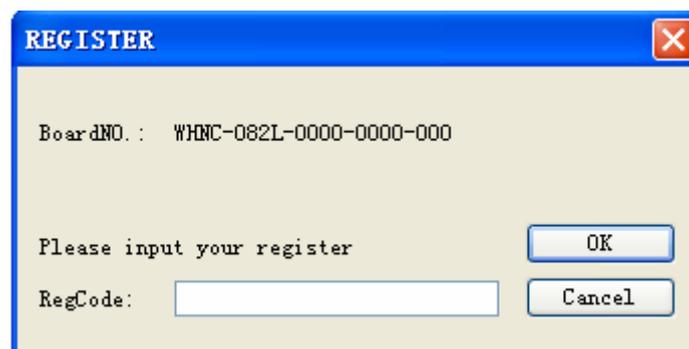


Fig. 8-12 Register dialogue

Note: the Card ID changes with registered times, reflected in the last three numbers of Card No.. When the registered times is "0", the last three numbers are "000"; when "1", the last three numbers will become "001".

9. Parameters of NcEditor in Laser Engraving

With abundant machining parameters, laser engraving system is competent for various machining tasks. This chapter will specialize in introducing the parameters involved in laser engraving system. We divide the parameters into operators' parameters, manufacturers' parameters and developers' parameters in terms of permissions; to look over or modify some types of parameters, the user must have enough jurisdictions.

9.1. Parameters Setting

Click [Set Parameters] under [View] menu to eject a dialog, as shown in Fig. 9-1; directly modify parameters in the dialog.

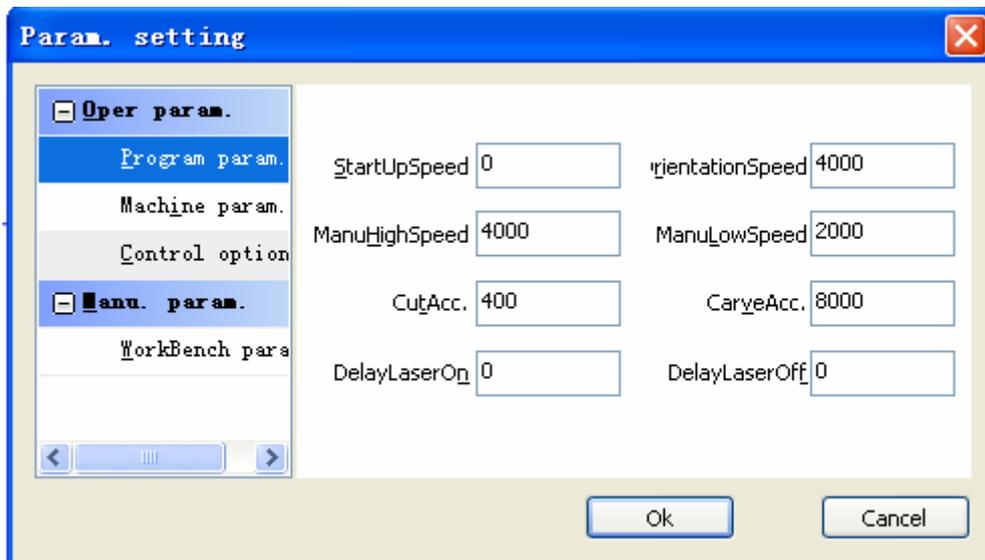


Fig. 9-1 Parameter setting

9.2. Operators' Parameters

9.2.1. Program Parameters

◆ Startup Speed

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

Default:	0
Instruction:	It sets the minimum feed speed in machining.

◆ **Orientation Speed**

Type:	Float
Unit:	mm/min (millimeter / minute)
Range:	0.001 ~ 100000.0
Default:	4000
Instruction:	It sets G00 speed.

◆ **Manu High Speed**

Type:	Float
Unit:	mm/min (millimeter / minute)
Range:	Value of manual low speed ~ Value of G00 speed
Default:	4000
Instruction:	It sets system default running speed under manual mode when a manual button and the acceleration key are pressed simultaneously.

◆ **Manu Low Speed**

Type:	Float
Unit:	mm/min (millimeter / minute)
Range:	0.001 ~ Value of manual high speed
Default:	2000
Instruction:	It sets system default maximum running speed under manual mode when a manual button is pressed.

◆ **Cut Acc.**

Type:	Float
Unit:	mm/s ² (millimeter / second ²)
Range:	0.0 ~ 100000.0
Default:	400
Instruction:	It sets the acceleration in cutting.

◆ **Carve Acc.**

Type:	Float
Unit:	mm/s ² (millimeter / second ²)
Range:	0.0 ~ 100000.0
Default:	8000
Instruction:	It sets the acceleration in carving.

◆ **Delay Laser on**

Type:	Float
Unit:	ms (millisecond)
Range:	0.0 ~ 100000.0
Default:	0
Instruction:	It sets the delay time of opening laser valve before machining for a laser engraving machine.

◆ **Delay Laser off**

Type:	Float
Unit:	ms (millisecond)
Range:	0.0 ~ 100000.0
Default:	0

Instruction:	It sets the delay time after closing laser valve when machining ends for a laser engraving machine.
---------------------	---

9.2.2. Machining Parameters

◆ Carve Mode

Type:	Integer
Unit:	None
Range:	Carve black Carve white
Default:	Carve black
Instruction:	It sets engraving mode, depending on the engraved materials. If the background color of engraved materials is such deep color as black, please choose “carve white”; if the background color of engraved materials is such light color as white, please choose “carve black”

◆ Carve Direction

Type:	Integer
Unit:	None
Range:	Double carve Single carve
Default:	Double carve
Instruction:	It selects engraving direction. Laser-jet will be on in forward engraving, but off in reverse engraving under [Single carve]; It will be on in both forward & reverse carving under [Double carve] mode.

◆ Border Setting

Clicking the button [Border Setting] will eject a following dialogue, as shown in Fig. 9-2.

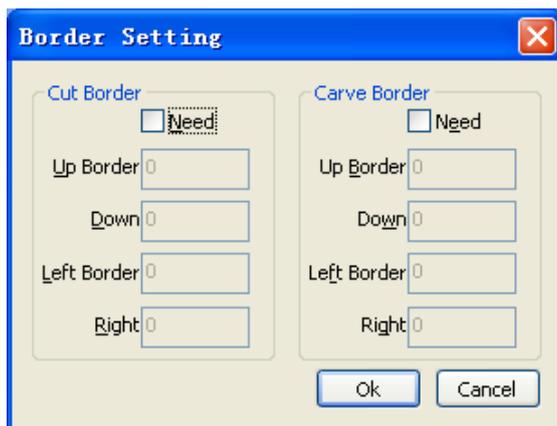


Fig. 9-2 Border setting

The user has two modes for selection, [Cut Border] and [Carve Border], and can also specify the values of up, down, left and right borders.

◆ **Carve Offset**

Type:	Float
Unit:	None
Range:	None
Default:	None
Instruction:	It is used to deal with the error caused by backlash of machine tool under [Double Carve] mode.

9.2.3. Control Option

Clicking the button [Control Option], a dialogue will pop up, as shown in Fig. 9-3.

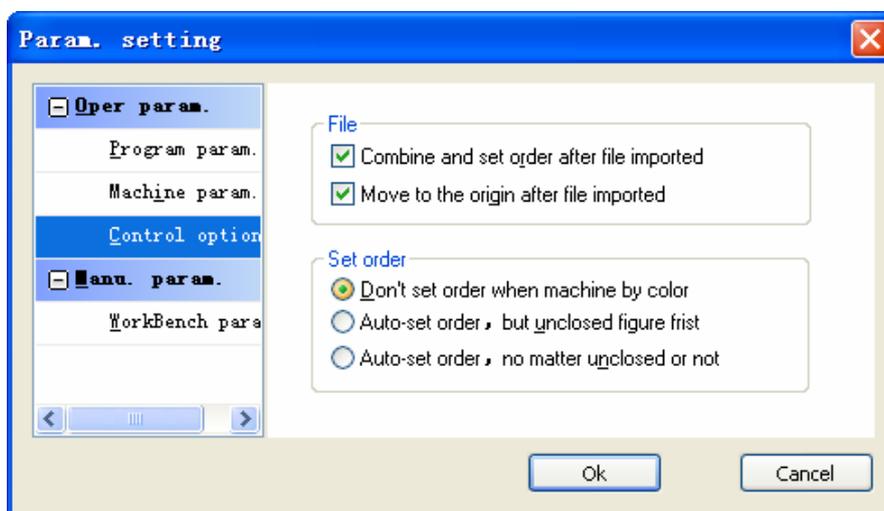


Fig. 9-3 [Control Option]

This function is mainly used for parameter settings related to loading files and sorting, etc.

9.3. Manufacturers' Parameters

When selecting [Manufacturers' Parameters], the user needs to enter the password to view and modify them.

9.3.1. Workbench Parameters

◆ Axis Factor (Pulse Equivalent) (XY)

Type:	Float
Unit:	mm/p (millimeter / pulse)
Range:	0.0000009 ~ 999.0
Default:	0.001
Instruction:	It refers to the displacement or angle on the corresponding axis generated 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. According to the transmission connection, the angle displacement of the stepping pulse will be converted into linear value.

◆ Workbench (Length)

Type:	Float
Unit:	mm (millimeter)
Range:	-99999.0 ~ 99999.0
Default:	800
Instruction:	It specifies the length of machine tool. Set it according to the specific machine tool.

◆ Workbench (Width)

Type:	Float
--------------	-------

Unit:	mm (millimeter)
Range:	-99999.0 ~ 99999.0
Default:	600
Instruction:	It specifies the width of machine tool. Set it according to the specific machine tool.

◆ **Axis Dir. (X \ Y)**

Type:	Integer
Unit:	None
Range:	X: Left / Right Y: Front / Back
Default:	X: Left Y: Back
Instruction:	It specifies the increasing direction of mechanical coordinates of each axis. After making sure the positive direction of each axis based on right-hand rule, manually operate the machine tool to check whether the axial motion is right. If the direction is opposite, modify this parameter.

◆ **Laser Polarity**

Type:	Integer
Unit:	None
Range:	Low-level laser on High-level laser on
Default:	Low-level laser on
Instruction:	It specifies the polarity of laser-jet.

Clicking the button [Machine Orig. Setting], the following dialogue will pop up, as shown in Fig. 9-4, for parameters setting related to homing.

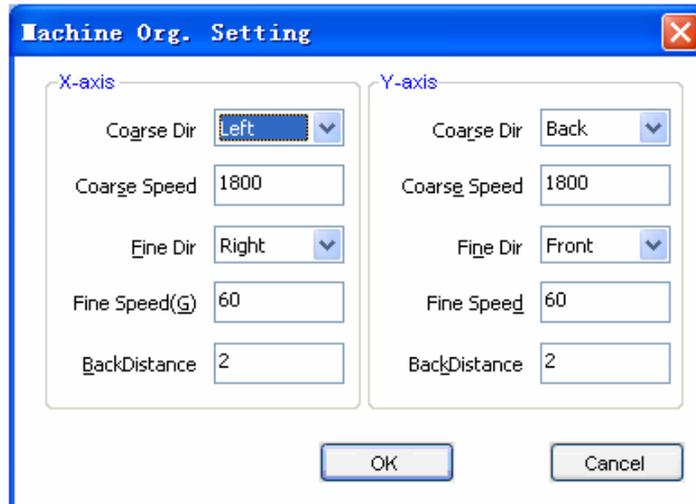


Fig. 9-4 Setting of returning to machine origin

◆ Coarse Dir. (X\Y)

Type:	Integer
Unit:	None
Range:	X: Left / Right Y: Front / Back
Default:	X: Left Y: Back
Instruction:	It specifies the motion direction of coarse positioning stage in the process of returning to machine origin (i.e. the motion direction of machine tool towards origin switch from any point).

◆ Coarse Speed (X\Y)

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.0 ~ 10000.0
Default:	X: 1800 Y: 1800
Instruction:	It specifies the feed speed of coarse positioning in the process of returning to machine origin.

◆ Fine Dir. (X\Y)

Type:	Integer
Unit:	None
Range:	X: Left / Right Y: Front / Back
Default:	X: Right Y: Front
Instruction:	It specifies the motion direction of fine positioning stage in the process of returning to machine origin (i.e. the motion direction of machine tool towards encoder origin from any point).

◆ **Fine Speed (X\Y)**

Type:	Float
Unit:	mm/min (millimeter/minute)
Range:	0.0 ~ 10000.0
Default:	X: 60 Y: 60
Instruction:	It specifies the feed speed of fine positioning in the process of returning to machine origin.

◆ **Back Distance (X\Y)**

Type:	Float
Unit:	mm (millimeter)
Range:	-1000.0 ~ 1000.0
Default:	X: 2.0 Y: 2.0
Instruction:	It refers to the additional displacement after finishing fine positioning stage of returning to machine origin in order to exit from fine positioning signal belt.

10. Appendix

10.1.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 also be provided.

- ◆ **Single number keys (small keyboard is invalid) can be used to choose function section under the current mode (AUTO, MANUAL, REFER). List the keys according to the sequence of the screen from left to right.**

1	position function section
2	offset function section
3	program function section
4	system function section
5	parameter function section
6	diagnosis function section

- ◆ **Single number keys (small keyboard is invalid) can be used to choose mode.**

7	auto mode
8	manual mode
9	reference point mode

- ◆ **Single letter keys can be used to choose function screen under current function section. List the keys according to the sequence of the screen from left to right.**

Q	the first function screen
W	the second function screen
E	the third function screen
R	the fourth function screen
T	the fifth function screen
Y	the sixth function screen

- ◆ **Other single letter keys frequently used to choose buttons 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 in function screen)

★F9 start (valid under auto mode)

★F10 pause (valid under auto mode)

★F11 stop (valid under auto mode and MDI state)

Pause reset (E-stop)

Ctrl + O open and load

Ctrl + U unload

Ctrl + N create a processing procedure

Ctrl + E open and edit

Ctrl + P edit current processing procedure

★Ctrl + F9 select processing block (valid under auto mode)

Shift + F6 set the current point as workpiece origin

Shift + F9 breakpoint resuming

10.2. Software Operation Procedure

Software Operation Procedure

