

**WEIHONG[®] ENGRAVING MACHINE'S MOTION
CONTROL SYSTEM**

Ncstudio[™]-V8

(For PCIMC-6A Double Z-axes)

User's Manual

Thank you for choosing our product.

This manual helps you acquaint the product of our company and understand the information of system constitution and configuration. It introduces the process of system installing and various functions. Before using this system, please read this manual in detail. It is beneficial for you to use it.

Owing to continuous update of hardware and software, it is possible that the software and the hardware you have received differ from the statement of this manual. We apologize for any inconvenience caused.

We list company address, contact person, telephone number and Web site hereby for your convenience and you are always welcome to contact us.

Company Name: Shanghai WeiHong Electronic Technology Co., Ltd.

Contact Person: Song Xiulong, Zhao Dongjing

Address: Building No.29, lane 2338, Duhui Road, Shanghai, China

Zip Code: 201108

Phone No.: 0086-21-33587550

Fax: 0086-21-33587550-880

Web site: www.weihong.com.cn

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

Pay attention to the followings:

- 1) Please 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) In order to guarantee safety and eliminate interference, the shell of computer and engraving machine must be grounded.
- 4) Please disconnect power supply if the machine is not in use for a long time.
- 5) Please pull out the BNC connector 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.
- 7) The cutter is very sharp. To prevent involvement which can result in injury and equipment damages, don't touch with hand, handkerchief, or silk kerchief while rotating.

Contents

1. GENERAL	1
1.1. SYSTEM CHARACTERISTIC	1
1.2. IMPROVEMENTS AND NEW FUNCTIONS	3
2. SYSTEM INSTALLATION AND CONNECTION	4
2.1. BASIC CONFIGURATIONS OF NcSTUDIO™	4
2.2. INSTALLATION OF NcSTUDIO™	4
2.3. UNINSTALL NcSTUDIO™ SYSTEM	9
2.4. CONNECTION BETWEEN NcSTUDIO™ MOTION CONTROL CARD AND ITS DRIVER SYSTEM	9
3. BASIC CONCEPTS OF NCSTUDIO™	10
3.1. OPERATION MODE AND STATE.....	10
3.2. MACHINE’S COORDINATE SYSTEM.....	11
4. OPERATION INTERFACE OF NCSTUDIO™	13
4.1. TITLE BAR	13
4.2. MENU BAR.....	14
4.3. TOOL BAR	16
4.4. NC INFORMATION BAR	16
4.5. STATUS BAR.....	16
4.6. NC STATE WINDOW	17
4.7. AUTOMATIC OPERATION WINDOW	22
4.8. MANUAL OPERATION WINDOW	24
4.9. CALIBRATION OPERATION WINDOW	26
4.10. MACHINING TRACE WINDOW / PROCESSING TRACK WINDOW	31
4.11. SYSTEM LOG WINDOW	37

4.12.	PROCEDURE MANAGEMENT WINDOW	40
4.13.	PROCEDURE FILE EDIT WINDOW	44
4.14.	STATE OF INPUT OR OUTPUT (I/O) WINDOW	45
5.	NCSTUDIO™ MENU SYSTEM.....	46
5.1.	"FILE" MENU	46
5.2.	"EDIT" MENU	48
5.3.	"VIEW" MENU.....	52
5.4.	"OPERATION" MENU	56
5.5.	"MACHINE" MENU.....	68
5.6.	"WINDOW" MENU	71
5.7.	"HELP" MENU	71
6.	PARAMETER CONFIGURATION.....	74
6.1.	RIGHT OF PARAMETER MODIFICATION	74
6.2.	PARAMETER MODIFICATION METHOD.....	74
6.3.	OPERATOR AUTHORIZATION PARAMETERS.....	75
7.	OPERATION INSTRUCTION FOR CUSTOMIZED SMALL KEYBOARD	99
8.	OPERATION STEPS	101
8.1.	POWER ON.....	101
8.2.	MACHINE RESET (OPTIONAL)	101
8.3.	LOADING THE PROCESSING PROCEDURE.....	101
8.4.	MANUAL OPERATION	101
8.5.	SET WORKPIECE ORIGIN	102
8.6.	EXECUTE AUTO MACHINING.....	103
9.	PRECAUTIONS IN OPERATION	104
9.1.	PRECAUTIONS FOR MULTI-TASK EXECUTING.....	104

9.2.	PRECAUTIONS FOR BACKING TO MACHINE REFERENCE POINT	104
10.	SOFTWARE PERMISSION AGREEMENT	105
11.	APPENDIX: SHORTCUT KEYS LIST	107
11.1.	GENERAL SHORTCUT KEY	107
11.2.	SHORTCUT KEY OF MANUAL WINDOW	109
11.3.	SHORTCUT KEY IN TRACK WINDOW	109

1. General

NcStudio™ software is an engraving machine's motion control system developed by Shanghai Weihong Electronic technology Co. LTD. and also belonging to the company's copyright. NcStudio™ supports codes of G and PLT format which are produced by CAD/CAM applications such as UG, MASTERCAM, CASMATE, Art CAM, AUTOCAD and COREDRAW etc.

NcStudio™ is based on the operation system of Microsoft Windows and takes the great advantage of 32-bits computing and multi-tasks. At the same time, its standard style of Windows interface is simple and dependable, and easy to learn.

In addition to functions of Manual, Stepping and Automatic backing to machine reference point, the CNC system of NcStudio™ also has functions of simulation, dynamic track display, and automatic calibration of Z-axis, breakpoint resume (program jump execution), and machining with rotary-axis, and so on.

NcStudio™ can be used together with all kinds of three-dimensional engraving machine and three-dimensional milling machine, and is applicable to all kinds of complicated mold machining, advertisement and incision fields.

1.1. System Characteristic

NcStudio™ software has the following functions:

- 1) Basic configuration: four motion axes; and can be further extended by switch value input points and switch value input points; and an analog output. These configurations may be different with different manufacturer's choice.
- 2) Support CNC rotary- turret (There may be difference for different version)
- 3) Automatic process; completely supports G order, HP PLT format, DXF format and JDPaint ENG format; compatible with CAD/CAM software, such as UG, Pro/E, MasterCAM, Cimatron, CASMate, ArtCAM, etc.
- 4) Manual function, including control modes of serial, step, and handwheel; user is allowed to control the machine tool not only by input equipment of machine tool but also by input equipment of computer, such as keyboard and mouse.
- 5) Machining function of array; execute a processing procedure repeatedly according to user's appointed numbers of row and column of the array.
- 6) Rotary-image machining function; the function takes the origin of workpiece to be the center and carries out machining of rotary image.
- 7) Stepping function; it is convenient for user to set accurate feed length; the step-size can be agilely adjusted.
- 8) Function of manual data input (MDI); user can directly input G code by online and immediately execute it.

- 9) Advanced processing instruction; input several simple parameters, and you can complete functions of milling bottom or marking edge.
- 10) Single block mode; Customer can set the machining task to be executed as single block mode, thereby, it will provide a good support for fault diagnosis and malfunction restoring.
- 11) Advanced automatic functions of breakpoint resume and program jump execution.
- 12) The function of feeding axis backs to the machine reference point accurately.
- 13) Auto calibration; provides user with the functions of fixed calibration, floated calibration, automatic midsplit, etc.
- 14) Function of preserving/resuming the machining state; the system has well taken into consideration the power breaking problem which commonly occurred to users in southern China; system will not cause any damage to the file although power breaking; functions of breakpoint resume and backing to the machine reference point can fully guarantee the dependable instauration after restart.
- 15) Feedrate online adjustment; user can adjust the feedrate at any time in machining process from 0 %(the smallest) to 120 %(the biggest).
- 16) Add new function for high-smooth speed connection algorithm. In a usual NC system, connection speed between two G codes commonly is a fixed value (e.g., it may be 0 or a certain small value). However, in new version of NC system, it adopts a particular machining speed adaptive prediction algorithm. In accordance with speed value, direction, 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%), improved the processing capability, and eliminated the speed chatter marks left on the processing surface.
- 17) Three-dimensional simulation display; with simple operation, you can observe the machining result of three-dimension, which helps you know the machining result more accurately and more intuitively.
- 18) Function of simulation; this function can simulate machining process and will finish in a short time. Meanwhile, it can check if there is any mistake with the program and if the processing result can meet user's satisfaction. Moreover, it can count precisely the actual processing time.
- 19) Strong and agile keyboard operation. To satisfy user's need, we offer a strong support to the keyboard operation.
- 20) Log function; system has a strong Log function which is very helpful for user to view the detailed processing information and system diagnoses.
- 21) Enclosed files manager; what the user needs to do is to save the processing file into a specified directory; NcStudio™ will administrate these files in the enclosed file manager.
- 22) Enclosed files editor; user can load the processing procedure to the editor to edit and modify.
- 23) File machining information; with simulate or actual machining, window for file machining information can help user to count the file running time and the range of machining, etc.
- 24) PCI BUS motion control card.

1.2. Improvements and New Functions

The following improvements and new functions are applied for version 8.50 or above:

- 1) Count the number of workpieces; once machining task is finished, this function will count the number of workpieces automatically; user can clear the figure at any time.
- 2) When machining task has been finished, buzzer will give out a sound to prompt user the complement of machining. At the same time, the red lamp will glitter in compliance with the user's setting.
- 3) New instructions: G34, G35, G36, G37; for detailed information please turn to programming manual.
- 4) If current coordinate system is not G54 when machining, the CNc State View Window will present light green to prompt user.
- 5) New instructions G923; can directly set the tools offset; for detailed information please turn to programming manual.
- 6) Extended function of instruction G906; can inspect the specified port whether it is overtime; for detailed information please turns to programming manual.
- 7) New function: name the subprogramme; such as: O"SubProgName"; for detailed information please turn to programming manual.
- 8) New instruction: M903; used to change the current tool number; for detailed information please turn to programming manual.
- 9) New function: tool change; applied for round cutter head, and inline tool magazine.
- 10) New function: user can decide whether the spindle moves to the fixed workpiece coordinates when pause.
- 11) Improved function: when the polarity of the port is changed, there will be record in log automatically.
- 12) New function: the software is furnished with warming inspection function for lubricant oil level and spindle fault.
- 13) Interface of "modify the polarity of the current point" is more humanization.
- 14) Machine tool lighting control function increased newly.

2. System Installation and Connection

2.1. Basic Configurations of NcStudio™

Host Computer

CPU:	Pentium (586) or above.
EMS memory:	32M or above;
Hard disk:	2 G or above;
Display adaptor:	support 800*600 at least, high color mode;
Display:	14" VGA or above;
CD-ROM driver:	4X or higher;
Main board expanding slot:	more than one slot of PCI.

Operation System

Microsoft Windows 2000 Professional in Chinese version operation system or

Microsoft Windows XP Professional in Chinese version operation system

2.2. Installation of NcStudio™

Before installing the new version of NcStudio™, please firstly delete the old one. Regarding how to delete the old version, please turn to chapter 2.3 of uninstall procedure.

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

Please install the motion control card before install software. Simply speaking, there are the following steps:

- 1) Shut down the computer; install the motion control card.
- 2) Restart the computer; enter Windows operate system, Put the installation CD in; choose auto update package and double click it; the nstallation will begin and the computer will be restarted.

3) Run NcStudio™.

Detailed information about the key steps will be introduced below.

2.2.1. Install NcStudio™ Motion Control Card

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

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

2.2.2. Install NcStudio™ Software

Please install the software according to the following steps:


- 1) Turn on the computer's power supply, start the computer, and system enters into the Windows Operation System Automatically. Please setup the operation system firstly if you don't have. (Version 8.3 matches with WINXP/2000 only).
- 2) Don't forget to exit other applications that are running after starting Windows operation system.
- 3) Put the setup CD-ROM of NcStudio™ system into the CD-ROM driver.
- 4) Double click "My computer" icon; double click the CD-ROM icon. Find out the auto update package icon (), and double click it. At this time, a setup interface will be displayed in Fig.2-1.



Fig. 2-1 Software Installation Interface 1

Select "English", and enter into next step.

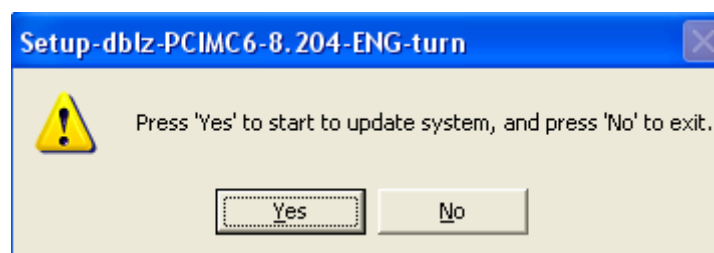


Fig. 2-2 Software Installation Interface 2

Select "Yes" and enter into next step.

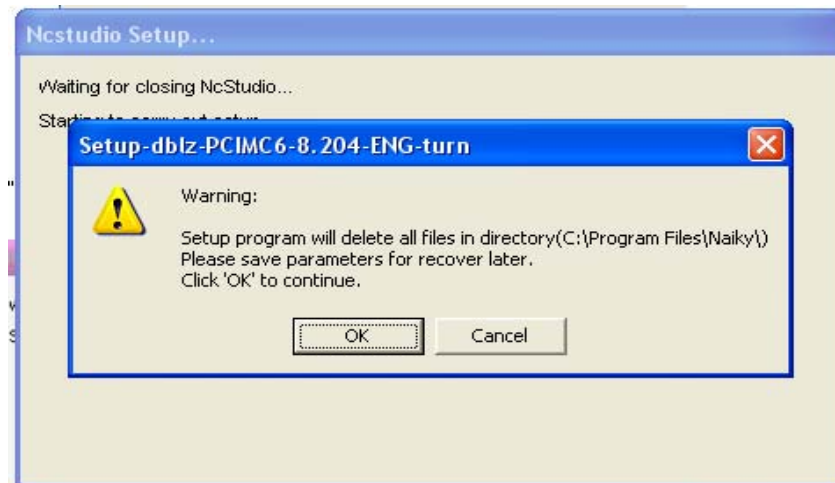


Fig. 2-3 Software Installation Interface 3

Click "OK ", and enter into next step.

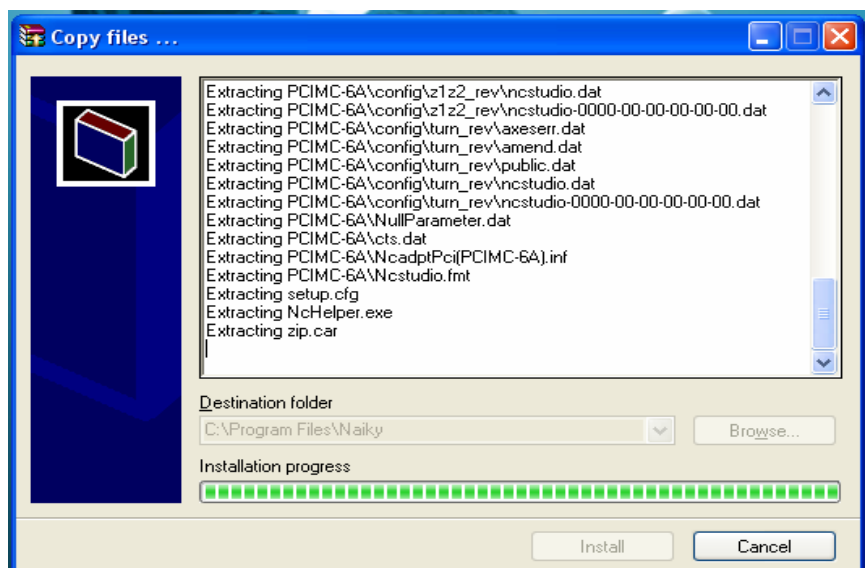


Fig. 2-4 Software Installation Interface 4

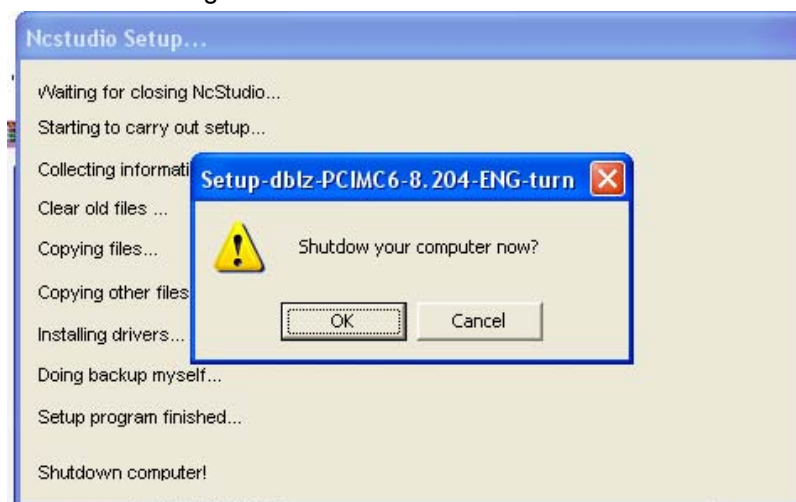


Fig. 2-5 Software Installation Interface 5

- 5) Click "OK", to turn off the computer.
- 6) Restart computer and the following dialog box will popup:



Fig. 2-6 Hardware Installation Interface 1

- 7) Select "NO, not this time (T)";
- 8) Click "Next", and the system will prompt you to install the software of Weihong NC adjuster card.



Fig. 2-7 Hardware Installation Interface 2

- 9) Select "Automatic Installation Software (Recommended)(I)";
- 10) Click "Next", and the system will start to install the software under system disk (default as C disk):

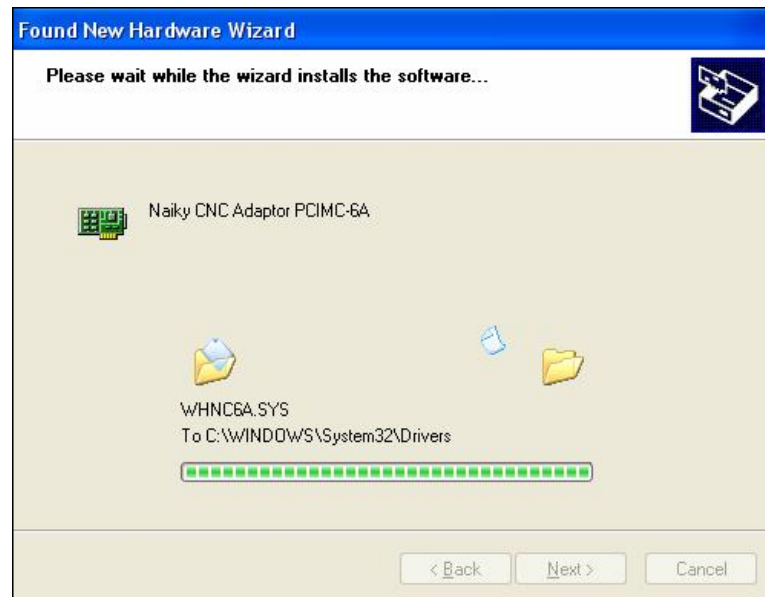



Fig. 2-8 Hardware Installation Interface 3

- 11) Click "Next", the system will prompt that the installation is finished:



Fig. 2-9 Hardware Installation Interface 3

- 12) Click "Finish".
- 13) After the installation of whole system is finished, double click the icon  on desk to enter into the NC system.

2.2.3. Install Special Small keyboard for PS2 Interface

NcStudio™, version 5.4.53 or above, supports special small keyboard, which is used for operating the machine tool conveniently.

Before Installing Ncstudio™ special small keyboard, please turn off the computer; Pull out the keyboard's attachment plug from the computer, and insert it into the small keyboard's socket which has one single strand, then insert the small keyboard's attachment socket which has two strands to the socket of the computer's keyboard. Make sure a good connection between plug and socket. After finishing the installation of the keyboard, you can start the computer, enter into Windows, and check keyboard and mouse for working well; execute NcStudio™ if necessary.

Caution:

Above Installation interfaces may vary in different versions; what we have provided here is only for your reference.

2.3. Uninstall NcStudio™ System

NcStudio™ is a green software, and has the advantages of easy installation, no need to register; easy to uninstall, and convenient to take with flash disk. To delete the NcStudio™ system, user only needs to delete the document folder: Program Files\Naiky.

When installing new version of NcStudio™ system, update will proceed automatically and will overlay the files of old edition.

2.4. Connection between NcStudio™ Motion Control Card and its Driver System

With the motion control card which is inserted in the PCI, mechanical motion control signal of NcStudio™ could realize the communication between NcStudio™ system and the stepping motor driven system that is installed in machine electric box.

Before joining NcStudio™ motion control card to the motor driven system, you should complete the installation between the machine and the electric box firstly; then connect the jack of motion control card to the jack of electric box with special cable. Here we finish all the connection between NcStudio™ system and the stepping motor driven system.

Note:

Regarding the connection between motion control card and electrical system, please refer to the introduction for specific type of card.

Different type of card may have different work mode and different function.

3. Basic Concepts of NcStudio™

3.1. Operation Mode and State

3.1.1. Operation Mode

As the operation of machine tool for user shall always under one of the following operation mode, it is necessary for user to understand the operation mode to ensure a correct operation.


Auto Mode

Under automatic operation mode, machine tool generates motions through the procedure loaded in advance. So under Auto mode, system must have procedure loaded firstly.

Manual Mode

Manual mode consists of continuous mode and stepping mode.

Under continuous mode, user can directly control the motion of the machining tool via manual operation equipments, such as computer keyboard, handset box, or MPG (manual pulse generator). When user sends out motion signals through these equipments, for

example when user presses down  button on software interface, the machine tool will moves continuously until the signal disappears, such as when user releases the button.

Under stepping mode, user also directly controls the motion of the machining tool via manual input devices such as computer keyboard, handset box and manual pulse generator. Its difference from continuous mode is that each time user presses and loses the button can only cause the machine tool to move a specified distance. That is to say, user can control the displacement of the machine tool accurately.

3.1.2. Operation State

Under each operation mode, there are several operation states. It is operation mode and operation state that decide together the full state of the machine tool.

IDLE

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

ESTOP

This is an abnormal state. When there is a hardware fault or when user presses emergence-stop button, system will enter ESTOP state and implement the predetermined protection actions, such as turning off spindle motor and cool pump. In this state the machine tool is locked and cannot carry out any new motion. After hardware fault or ESTOP state is obviated, system will automatically implement [Reset] action to make the machine tool return to IDLE state.

RUNNING

When the machine is implementing any action, system enters into Running State.

PAUSE

While the machine tool is running, if user implement “Operate | Pause” order, or system parses a M01 command (Wait Command), system will enter PAUSE state and wait for user’s further instruction. User can implement “Operate | Start” order to continue the running, or, perform [Stop] or [Reset] button to stop current operation and make system enter IDLE state.

Lock

Lock state is an inner state, which usually appears when system switches from one state to another state. Usually, user can’t get to it.

3.2. Machine’s Coordinate System

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

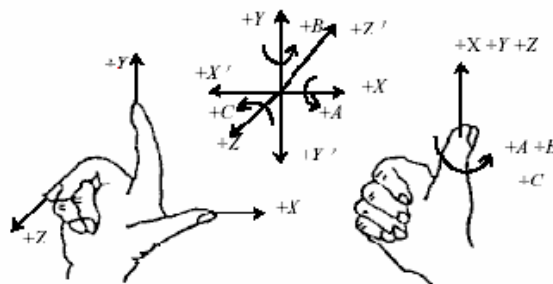


Fig. 3-1 Demonstration for Right Hand Principle

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

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

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

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

3.2.1. Machine Coordinate System

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

In order to full support Machine Coordinate System, the machine must possess the capability of returning to the mechanical machine reference point. Otherwise, the concept of the Machine Coordinate System will be only incarnated in software.

3.2.2. Workpiece Coordinate System

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

4. Operation Interface of NcStudio™

Install NcStudio™ according to the steps introduced previously, and double click the icon of NcStudio™ on the desk to run it. Its main operation interface is shown as below.

NcStudio™ interface consists of title bar, menu bar, tool bar, state bar and some functional window.

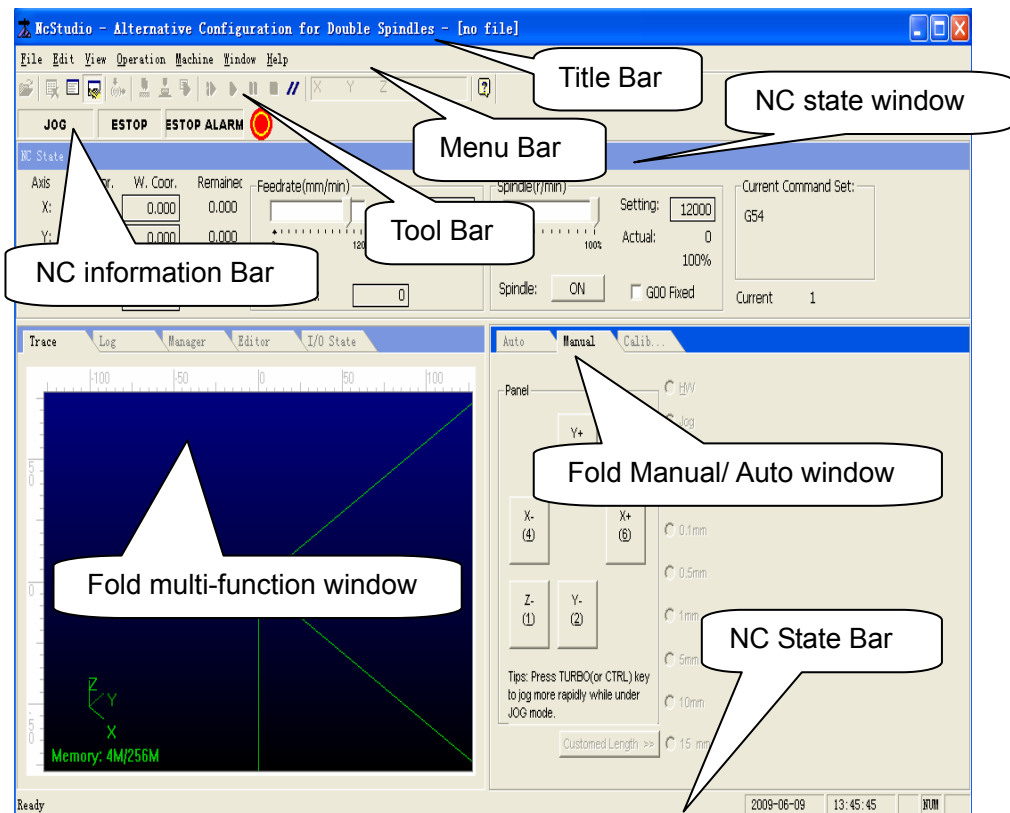


Fig. 4-1 Operation Interface of NcStudio™

The function windows are divided into three zones which is shifted by pressing ESC key, including:

The first zone: NC state window.

The second zone: track window, log window, procedure manager window, editor window and I/O state window.

The third zone: auto window, manual window and calibration window.

4.1. Title Bar

The first column of NcStudio operation interface is title bar shown in Fig.4-2. It is used to

show the name of the application and the name of the loaded processing file. The color of title bar is used to indicate whether the window is activated or not.



Fig. 4-2 Title Bar

Note:

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

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

The icon in the left of the title bar is a system menu box. It is used to open the window control menu. Click the icon or press "Alt + spacebar", and a system menu will pop up.

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

In addition, each sub-window also has its title bar. The active sub-window and the inactive sub-window can be distinguished from the color of the title bar. Please refer to the describing of chapters below.

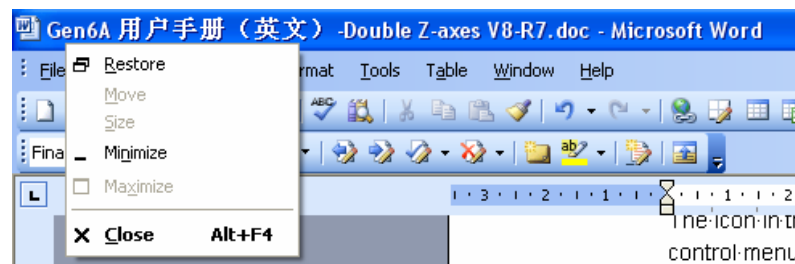


Fig. 4-3 System Menu

4.2. Menu Bar

Underneath the title bar is a menu bar; it includes many concealed menus, as shown in Fig. 4-4 Menu Bar.

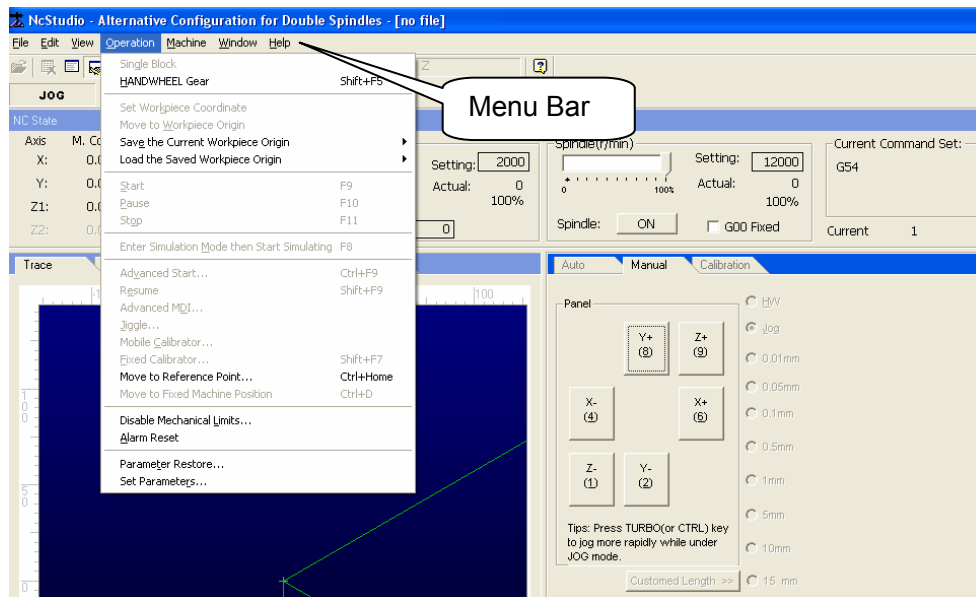


Fig. 4-4 Menu Bar

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

Mouse Operation

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

Keyboard Operation

Press down simultaneously 'Alt' key and hot letter key. For example, there is a line under 'F' in the "File" menu, so the combination key is "ALT+F". After that, a pull-down menu will pop up.

Shortcut Key Operation

In pull-down submenu, some items have shortcut keys in their right. For example, F9 is the hot key of "Start" in the menu of "Operation". That is to say, user can directly select the target item by direct shortcut key, which can avoid the trouble to enter into multilayer menus.

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

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

4.3. Tool Bar

Underneath the menu bar is a toolbar. Toolbar is composed of many operation buttons, which correspond to some menu items or functions, Click the button to realize a specified function.

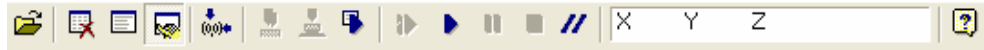


Fig. 4-5 Tool Bar

Toolbar has greatly simplified user's operation process, and makes the whole operation process visual instead of the tedious order list.

Direct Positioning Function

Under IDLE state, user can input the workpiece coordinates into "Direct Appoint Position" dialog, press "Enter" key, and system will move the butter to the appointed position quickly.

means that the input coordinate is workpiece coordinate;

which is increased with a star mark "*", means that the input coordinate is machine coordinate.

All the other buttons on the toolbar are related to the corresponding menu items. When the mouse is pointed at a certain tool, a functional prompt box will be popped up. Please refer to the following chapters.

4.4. NC Information Bar

Underneath the toolbar is a NC Information Bar; it is used to show the current states and some alarm information.

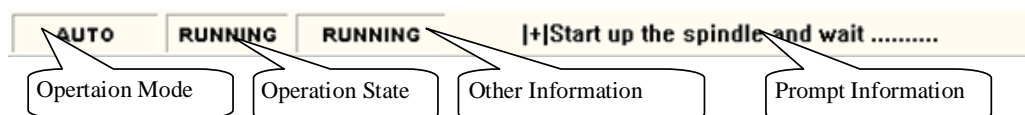


Fig. 4-6 NC Information Bar

4.5. Status Bar

At the bottom of the screen is status bar, as shown in fig. 4-7:

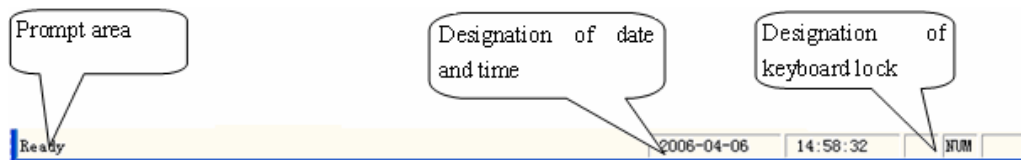


Fig. 4-7 Status Bar

Prompt Area:

It gives the prompt information of current operation or about the selected instruction.

Designation of Date and Time:

Show the current information of date and the time.

Designation of Keyboard Lock:

Show the current state of the keyboard capital lock, the num lock and the scroll lock.

4.6. NC State Window

NC state window is underneath the state bar; it can be divided into four areas according to the function: “current position of the spindle” area, “feed speed” area, “spindle speed” area and “instruction of current interpolation” area.

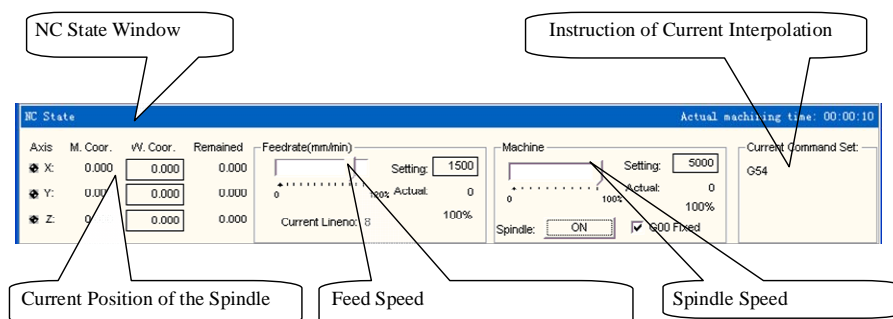


Fig. 4-8 NC State Window

Current Position Zone of the Spindle

Show the current position of the spindle, including workpiece coordinate, machine coordinate and residual distance in the area, moreover, it can set the current point as workpiece origin at any time.

Residual Distance

It is the distance between the position where the tool is designated to arrive under current instruction and the instantaneous position of the tool at executing current instruction.

Feed Speed Zone

Set feed speed, adjust feedrate, and display the actual value of feedrate and feed speed in the area.

Spindle Speed Zone

Set spindle speed, adjust ratio of the spindle speed, show the actual value of the ratio and the spindle speed, and also can manipulate the revolution of spindle.

Instruction Zone of Current Interpolation

Show the state of current instruction being executed, such as: G54, mode/modeless, G01, G17, G18, G19, etc.

Time Information

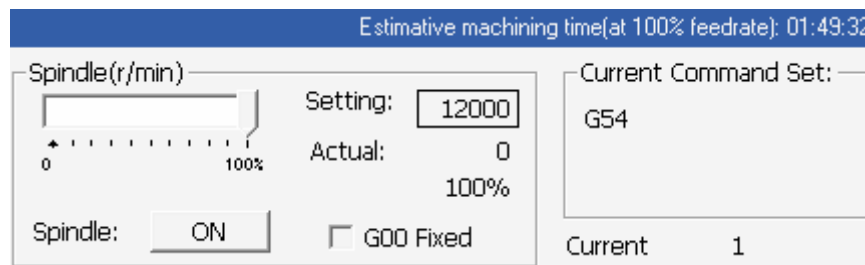


Fig. 4-9 Time Information

At the right of the title bar, the time displayed under simulation state is the predicted executing time (with 100% feedrate). During actual executing process, it will display the actual executing time.

Current Position

In order to describe every position expediently, NcStudio™ shows two sets of coordinate systems: mechanical coordinate system and workpiece coordinate system. NcStudio™ provides many functions to support these two sets of coordinate systems; user can set the relative offset between these two coordinate systems.

After executing “back to the machine reference point”, in front of each axis, there will be an icon “availability of machine coordinate system”, as shown in below.

NC State			
Axis	M. Coord.	W. Coord.	Remained
X:	0.000	0.000	0.000
Y:	0.000	0.000	0.000
Z1:	0.000	0.000	0.000
Z2:	0.000	0.000	0.000

Fig. 4-10 Current Position

System provides a convenient method to set and revise the position of workpiece origin: to set the current point as workpiece origin, you only need to move the cursor to the display-area of the axis coordinate, click workpiece coordinate, and a dialog box will be popup as below:

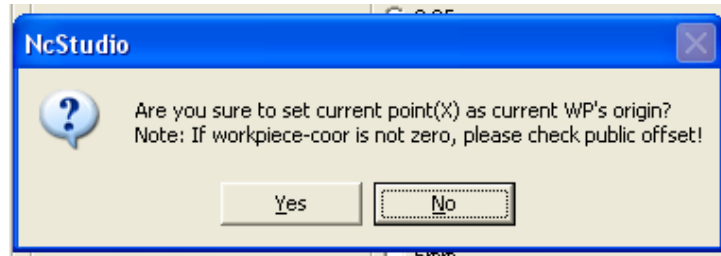


Fig. 4-11

Click “Yes”, and you will see the coordinate of that axis become zero. If you wish the current position of all 3 axes would be 0, please click all coordinate display-area of the 3 axes.

Prompt:

Please check common offset value if the current workpiece coordinate value after setting is not zero.

Feed Speed

In the zone of feed speed, it shows the setting speed, actual speed, speed feedrate, etc. It can also modify the setting value of speed and the feedrate.

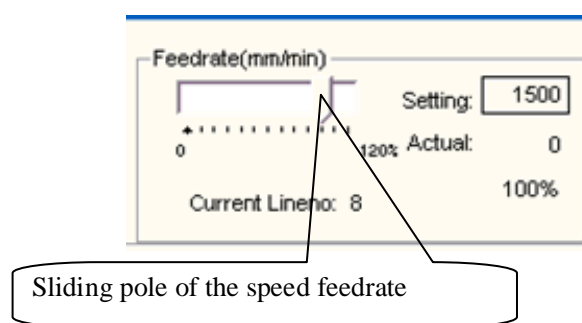


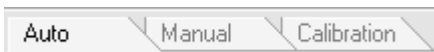
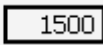
Fig. 4-12 Feed Speed Zone

Sliding Pole of the Speed feedrate:

Pulling the sliding pole can regulate the current moving speed within the scope of 0~120%. Speed feedrate is shown in the type of percentage.

Setting Value:

The setting value is located under auto mode,

namely  , click the figure after setting value
Setting:  , a dialog box will popup as below:

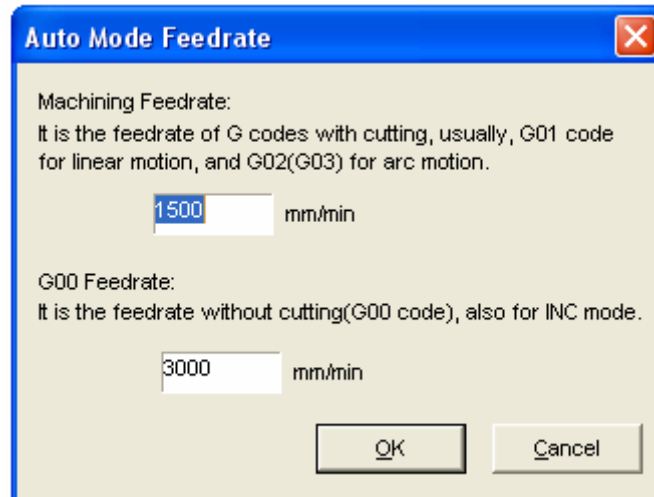
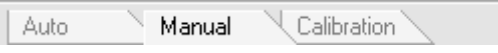
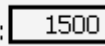


Fig. 4-13 Setting Auto Speed Dialog Box

It is used to adjust machining speed and idle stroke.

The setting value is located under Manual Mode,

namely  , click the figure after setting
value Setting:  , a dialog box will popup as below:

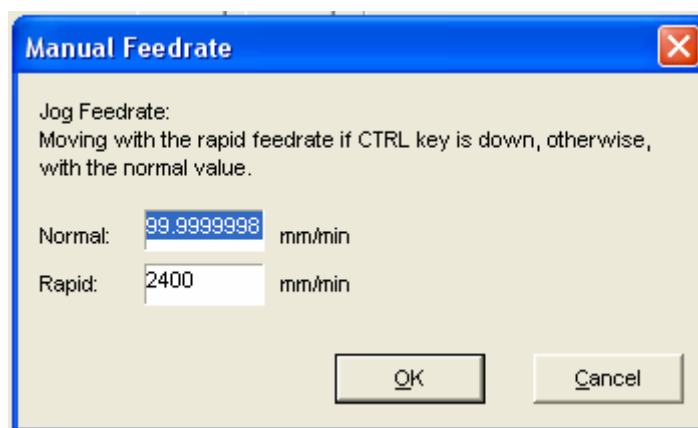
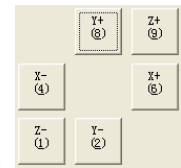


Fig. 4-14 Setting Manual Speed Dialog Box

It is used to adjust manual low speed and manual high speed.

Prompt:

How to switch between manual low speed and manual high speed under manual state:



Press Ctrl and direction key on Num key simultaneously (shown as the figure in), which will switch to manual high speed;

Press Num key directly, which will switch to manual low speed.

What needs to be pointed out is that the setting here is the same as the setting in "Operation Parameter" of parameter window.

Actual Value:

It is the instantaneous speed of the feed speed, and it varies with the modification of setting value, the current acceleration or deceleration, and the feedrate.

Record Number of Workpieces Machined:

It is used to record the number of workpiece machined. Click the number area and the number will become zero.

Spindle speed

Spindle speed zone shows setting speed, actual speed, feedrate percentage, etc; it can also modify the setting speed and feedrate percentage.

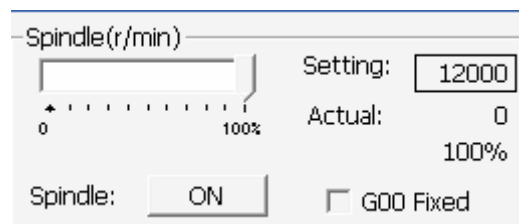


Fig. 4-15 Spindle Revolution Zone

Spindle Feedrate Block:

Pull the block to adjust current spindle speed in the scope of 0~100%, and the speed will show in the manner of percentage: $\text{Actual Value} = \text{Setting Value} \times \text{Current Spindle Speed Rate}$

Spindle Revolution Button:

This button is the same as the on-off of the spindle.

Setting Speed of the Spindle:

Clicking this button “Setting Value” will flip out a dialog box of “Spindle Revolution”; user can modify the speed of the spindle on the dialog, and the spindle revolution box is as shown in fig. 4-16:

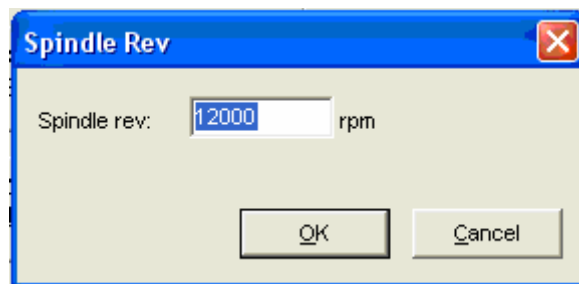


Fig. 4-16 Spindle Revolution Dialogue

Fixing Speed of G00

Select the item, and when running the idle stroke, the running speed will be fixed at hundred-percent of setting value of idle stroke speed without affected by feedrate.

Current Interpolation Instruction and Current Axis No.

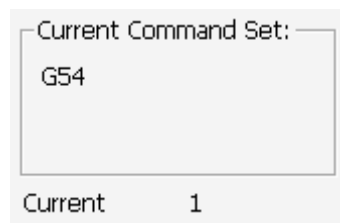


Fig. 4-17

Current interpolation instruction zone is used to display the state of current operated instruction; for example, G54, mode/ modeless, G01, G17, G18, G19, and so on.

The axis No. zone is used to display the current usable axis in Z1 and Z2. The axis can be single axis and double axis and “1” means Z1.

4.7. Automatic Operation Window

It shows the current processing file. Formats that NcStudio™ supports now include G instruction of ISO standard, HP PLT format, DXF format and ENG format and the NCE format which is a special format developed by our company. User can look into the current procedure from this window.

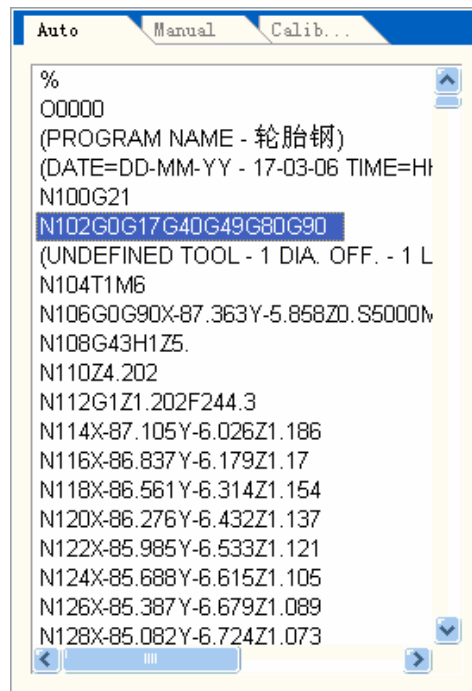


Fig. 4-18 Automatic Operation Window

Right click in this window; system will flip out a context menu, as shown in Fig. 4-19:

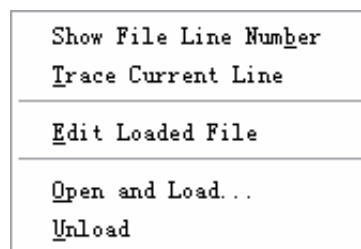


Fig. 4-19 Context Menu of Auto Operation Window

All of those menu items can be found in standard menu, among which, "Show File Line Number" and "Trace Current Line" can be found in "View" menu; three residual items can be found in "File" menu. Referring the explanation for them please turn to chapter 5.1 and 5.3.

The three windows for auto mode, manual mode, and calibration mode can be switch from each other.

Method for Switching to Auto Window:

1. Menu method: Choose "window | Show Auto Window".
2. Shortcut Key Method: Press "Ctrl+1" key to activate Auto window.
3. Single click: click the title "automatic" directly in the window.

Prompt:

Processing program shown in this window is only for your inspection; it can't be edited or

modified. If you need to edit the program, please select “File| Edit Processing File”; if you only want to edit a procedure or a file, please select click “file| open and edit...”

4.8. Manual Operation Window

Manual window provides user with an interactive operation environment to operate the machine tool with manual manner.

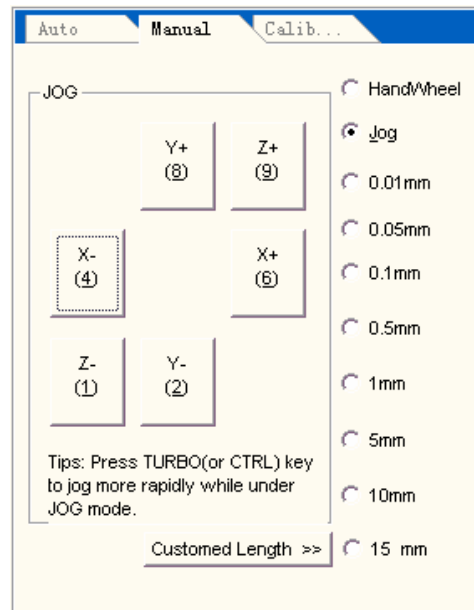






Fig. 4-20 Manual Operation Window



Methods for switching to manual window:

1. Menu Method: Choose "window | Show Manual Window".
2. Hotkey method: Press keys "Ctrl+2" to activate the window.

Direction Button

 and  buttons represent the positive or negative direction of X axis;


 and  buttons represent the positive or negative direction of Y axis;

 and  buttons represent the positive or negative direction of Z axis;

Feeding method selection

There are two feeding methods to operate the machine tool manually: Serial point-move mode, stepping mode, illustrated as below:

Serial Feed Operation

Select item  , to choose serial mode.

Press the direction Num. key on small keyboard, the machine tool will move, and lose the key, the machine tool will stop.

Caution:

When manual window is active, number lock is not taken into consideration.

Stepping Mode

Select any one in the following items to represent stepping mode.

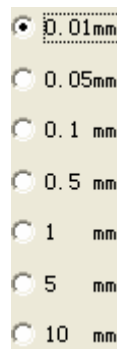
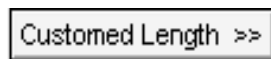


Fig. 4-21

When implementing stepping mode, in the processing track window the machining track will be shown in the color of G01 instruction.

The user can implement stepping mode via mouse, keyboard, and operation panel. The corresponding axis will move with a specified stepping length after every triggering of direction button.

Customized Step-size



Click “Customer setting step-size”, and the following dialog box will popup:

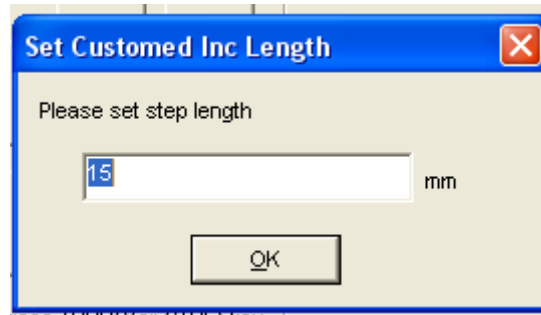


Fig. 4-22

Input proper step-size value, and click "Yes".

Caution:

You must avoid setting the customer step-size of Z-axis too large, or you will damage the machine tool with misoperation.

Since the system needs a certain time to carry out the task of each serial point-move, if you press the button too frequently, system will give you mistaking information: "system is busy now and the operation is invalid."

4.9. Calibration Operation Window

The calibration operation window is used the execute calibration operation.

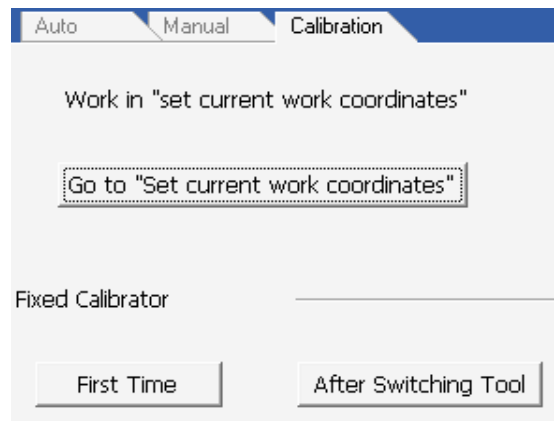


Fig. 4-23 Calibration Operation Window

4.9.1. Click Button "Switch to coordinate management and calibration"

The following dialog box will be popped up:

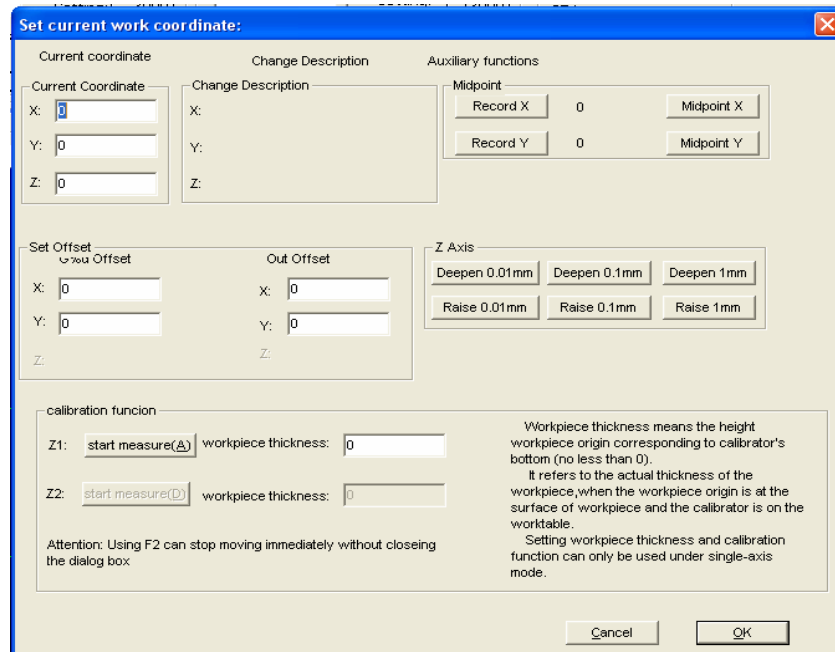


Fig. 4-24 Coordination Management and Calibration Operation Window
Current Workpiece Coordination Setting

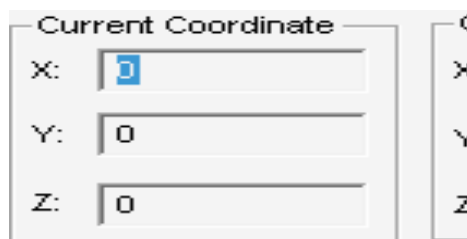


Fig. 4-25

The current number in current workpiece coordination shown can be modified directly and the modification will be prompted after corresponding axis, as shown in the following fig.

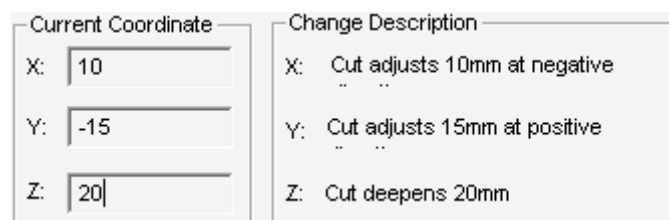
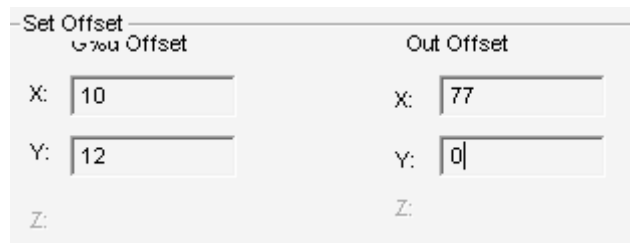


Fig. 4-26 Offset Setting



The 'Set Offset' dialog box is divided into two sections: 'In Offset' and 'Out Offset'. Each section contains input fields for X, Y, and Z coordinates. In the 'In Offset' section, X is 10 and Y is 12. In the 'Out Offset' section, X is 77 and Y is 0. The Z fields are empty in both sections.

Fig. 4-27

Workpiece offset is also titled as workpiece biasing: workpiece offset for X and Y axis is the offset from workpiece origin to corresponding mechanical origin, that is to say, it is the mechanical coordinate of X and Y axis of the system when the tool point moving to workpiece origin. The offset is for G54 coordinate system only.

Outer offset is also titled as common biasing: it is used to record the temporary adjusting value of workpiece origin. It can only be modified manually as there is no automatic function for the modification. It means that when having fixing calibration or floating calibration, the value will no be maintained also. If the front outer offset of calibration is not zero, the workpiece coordination value will not be zero also after the completion of calibration. For example, if the front outer offset of calibration value is "10", the outer offset value after the completion of calibration will be "10" also and the system workpiece coordination will be "-10".

Auxiliary Function



The 'Auxiliary functions' dialog box features a 'Midpoint' section with four buttons: 'Record X', 'Record Y', 'Midpoint X', and 'Midpoint Y'. The 'Record X' and 'Record Y' buttons are currently active, showing a value of 0 next to them.

Fig. 4-28

Manual Midsplit

It is used to get the center point between two points. The procedure: move the tool to the first point, and click button "record X" and then the system will record the X axis mechanical coordinate for the point; then move the tool to the second point, select the order "Midsplit X" and the system will work out the center point between the two points of X axis mechanical coordinate. The usage of button "record Y" and "midsplit Y" which are aim at Y axis mechanical coordinate is similar with the procedure.

Adjust Z Axis

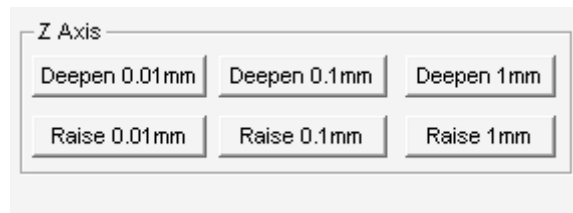


Fig. 4-29

Press the button to move the workpiece origin of Z axis up or down with a recommended distance, and form a new workpiece coordinate system; Click the button continuously for several times to move the distance with accumulated values of several times.

Calibration Function

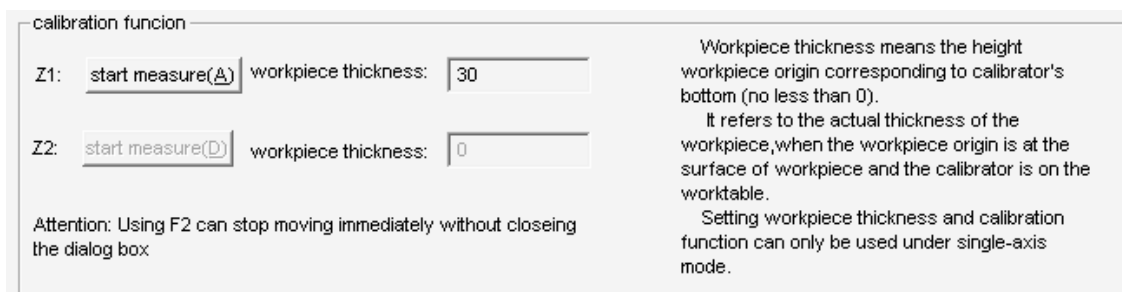


Fig. 4-30

Workpiece Thickness: the surface thickness (Namely, the origin position of Z axis workpiece coordinate) is corresponded to the height of workpiece surface (generally, it should be no less than Zero).

Input a figure as , click the button , and wait for calibration signal to finish calibration operation.

Caution:

The abovementioned calibration belongs to floated calibration process. Making use of the floated calibration function, the user can position workpiece origin for Z axis conveniently. The floated calibration procedures are shown as below: position the cutter onto the surface of workpiece, run the machine tool and move Z axis above the cutter, and then click the button “Start Measurement”, by this way, the system will execute the calibration process automatically to ensure the Z axis coordinate of workpiece coordinate.

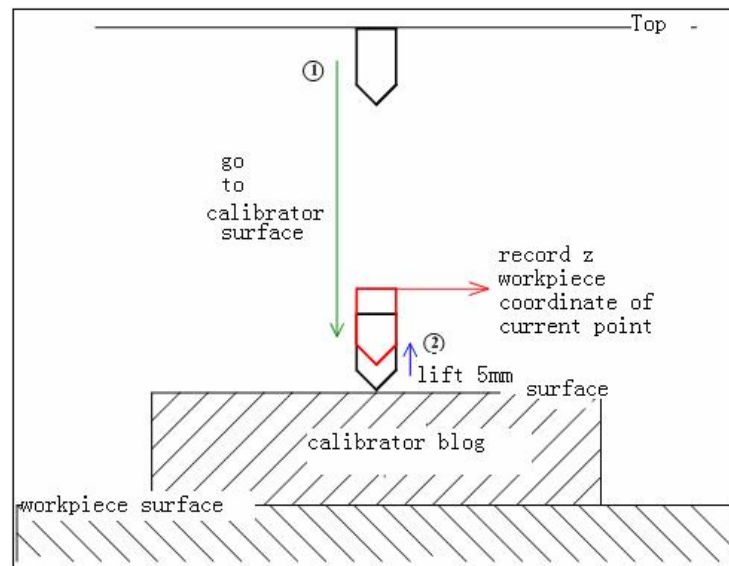


Fig. 4-31 Floated Calibration Diagram

4.9.2. Fixed Calibration

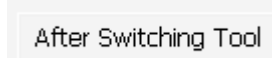


Fig. 4-32 Fixing Calibration

The function is used to recalibrate Z axis coordinate of workpiece origin after cutter replaced.

During calibration, check workpiece surface firstly, and set workpiece surface as Z axis

workpiece origin, click button , and then the system will record the workpiece coordinate for current Z axis automatically. If you need to carry out cutter

replacement or encounter cutter broke, click , and then the system will recover the previous workpiece origin of Z axis before cutter replacement or cutter broke automatically.

Caution:

The above calibration will be invalid automatically after NcStudio system closed, and the fixed calibration should be carried out again when restarting NcStudio.

Carry out the operation in accordance with the following fig.:

First Calibration:

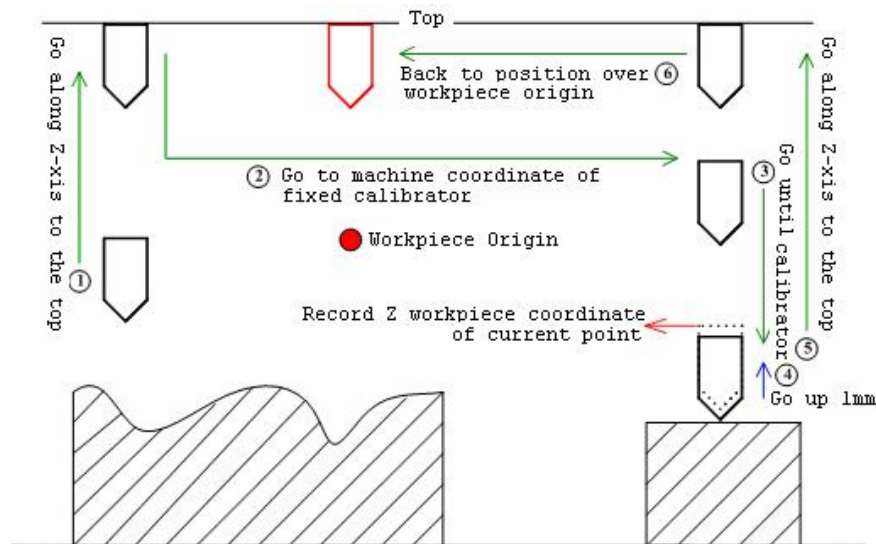


Fig. 4-33 First Calibration Diagram

Calibration after cutter replaced;

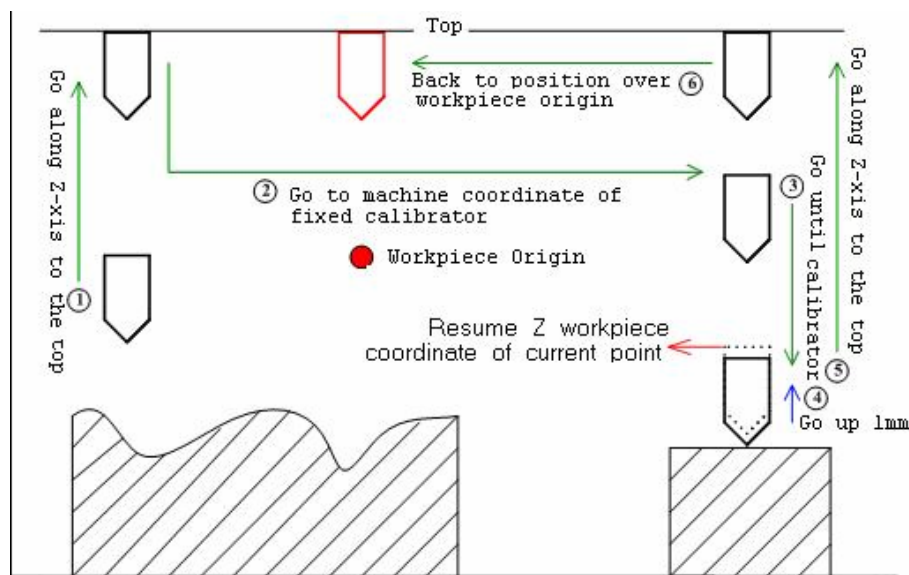


Fig. 4-34 Calibration After Cutter Replaced

4.10.Machining Trace Window / Processing Track Window

When the machine tool is implementing machining procedure or simulation, the processing track window can show the trace that follows the actual cutter path in real-time. The trace shown in real-time 3D helps user see the cutter path more visually, so it can ensure the correctness of machining.

The processing track window adopts three-dimensional view mode. You can select menu "View (V) | Customize..." to set it characteristically. Detailed information please refers to the following part of "Attribute".

Under three-dimensional tracking mode, user is provided with abundant operation to examine the figure from different angles and suitable scaling.

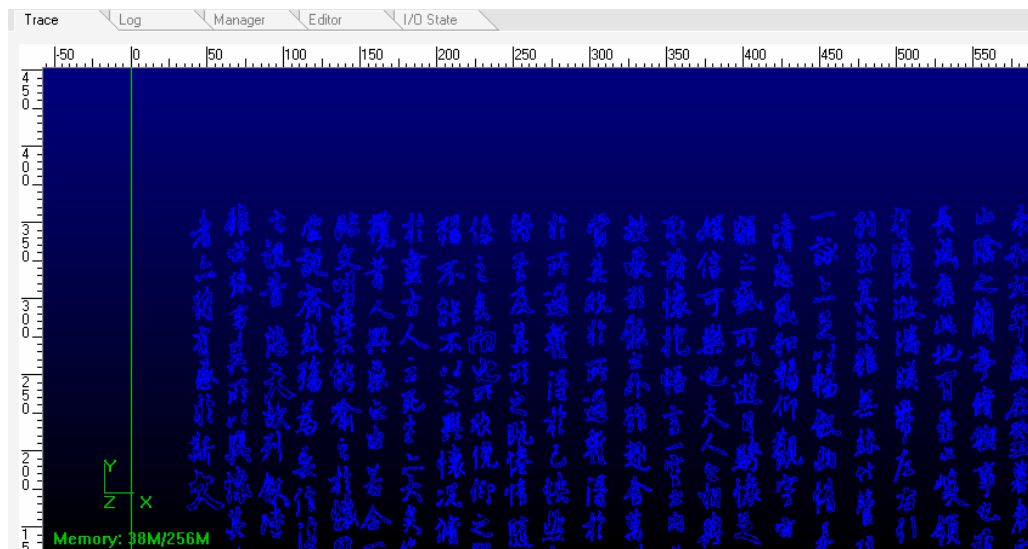


Fig. 4-35 Processing Track Window

Right click in the window, a context menu will popup as shown in the following figure:

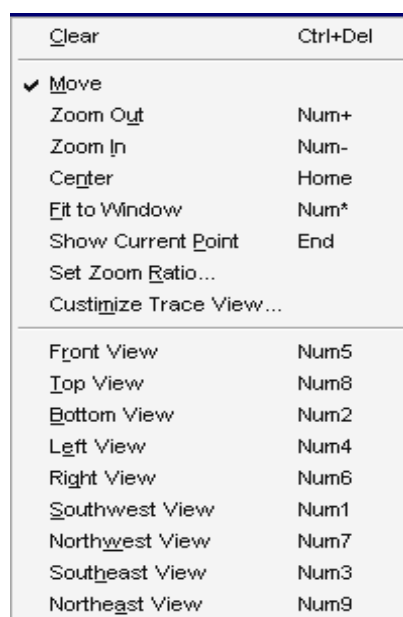



Fig. 4-36 Context Menu of Processing Track Window

Clear Function

After a long time of machining, the simulation figure becomes very complex and the temporary folder used to record processing path becomes more and more bulkiness. It will spend a lot of time at redrawing, moving, or revolving the simulation figure. At this time, user needs to clean the tracking window.

There are many methods to clear the processing track window. You can complete it by menu, toolbar button, shortcut key, and the keyboard.

Menu or Toolbar Method

Choose menu "Edit | Clear View", or choose the button  on the toolbar.

Shortcut Key Method

At any time, pressing "CTRL+ DEL" keys can realize the clearance of processing track window.

Mouse Method



Move the mouse into processing track window, right click the mouse, then a context menu will flip out; chose the option of "Clear".

Keyboard Method

When the processing track window in active, Press "DEL" key.

Moving Function

Mouse Method

Move the mouse into processing track window, right click the mouse, then a context menu will flip out; and select "move" option. At this time, the mouse shows ; press the left key of the mouse, the mouse shows ; drag the mouse slowly, after completing the moving of the whole following figure, loose it.

Keyboard Method

Use the four directions key on keyboard to move the processing track when the processing track window is the current active window.



Zoom Function: Zoom in & Zoom out

You can use three methods like the menu, keyboard, and the mouse adding keyboard to realize the zoom of the processing track.

Menu Method

Choose" View | Zoom in or Zoom out "to set a suitable scaling.

Mouse Method

Move the mouse into processing track window, and right click, and select "Zoom in" or "Zoom out" in the menu popued to switch between  and .

Prompt:

When using the mouse to zoom the processing track, user only need to use the mouse to select required area for zooming in or zooming out partially.

When the processing track interface reaches to the max size, it will be switched to zoom out mode automatically at this time.

When the processing track interface reaches to the min size, it will be switched to zoom in mode automatically at this time.

Keyboard Method

Activate this window firstly, then choose the key ("+" "-") on small keyboard to realize zoom.

Caution:

The keys "+" and "-" on the main keyboard are invalid.

Center View Function

Show the center of current machining scope.

Menu Method

Select "View | Center View" to perform.

Mouse Method

Move the mouse into processing track window, and click the right key of the mouse, a context menu will popup, and then select "Center" option.

Keyboard Method

First, activate this window, then press "Home" key on the keyboard.

Adjust to Window Size Function

The function will make the all processing track shown adaptively in the window. User will be able to see the entire content of the track without rolling the scroll bar.

Menu Method

Select "View | Fit to window size" from menu bar.

Mouse Method

Move the mouse into the display area; click the right key of mouse, a context menu will flip out, and then select "Fit to window size" option.

Keyboard Method

First, activate this window, then press "*" key on the keyboard.

Show Current Point Function

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

Menu Method

Choose "View | Show current point " to perform.

Mouse Method

Move the mouse into the display area of processing track window, click the right key of mouse, a context menu will flip out, then select "Show current point" option.

Keyboard Method

First, activate this window, and then press "End" key on the keyboard.

Customize Parameters (Attribute)

User can select menu "View | Customize...", or when the processing track window is active, click the right key of mouse, a context menu will flip out, then select "Customize" to perform the function.

The "Customize" dialog box is used to track the customized setting of window to realize the customer setting of tracking mode and color.

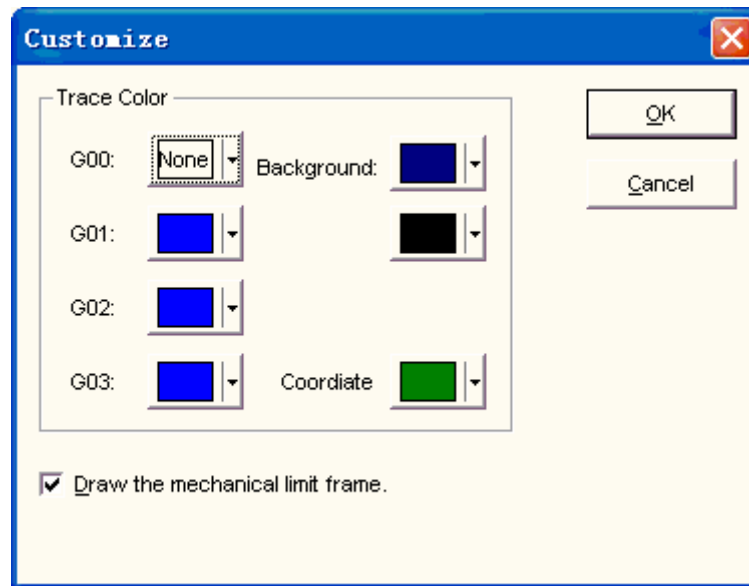


Fig. 4-37 Customize Dialogue

Click the color selecting button, system will flip out a pulling-down frame for you to choose a color, as shown in following figure.

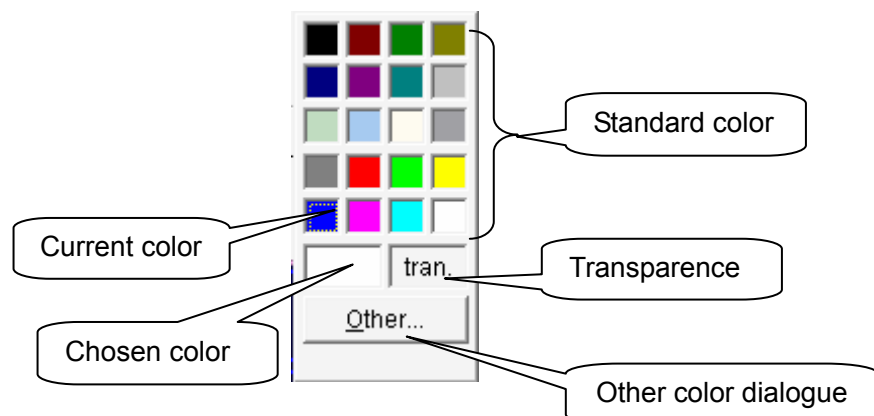


Fig. 4-38 Color Selection Box

Tracking Color

In the processing track window, you can allocate each element a different color respectively, including:

- (1) Color of G00 instruction: This color indicates the track color of G00 instruction.
- (2) Color of G01 instruction: This color indicates the track color of G01 instruction.
- (3) Color of G02 instruction: This color indicates the track color of G02 instruction.
- (4) Color of G03 instruction: This color indicates the track color of G03 instruction.

In addition, manual and point move track is indicated with color G01.

Background Color:

It is the background color of the processing track window. There are two kinds of background colors; user can set different colors to realize the gradual change between the two kinds of colors.

Coordinate Color:

It is the color of the hint coordinate displayed in the processing track window.

Prompt:

If the color you select is transparent, track of some instructions will be hidden. However, setting the background transparent is not a good idea, because it will lead that the processing track window can't be freshened correctly.

View Function

System provides many point of views in common use; you can switch quickly by the pressing number key on the small keyboard.

1) Southwest View	1
2) Bottom View	2
3) Southeast View	3
4) Left View	4
5) Front View	5
6) Right View	6
7) Northwest View	7
8) Top View	8
9) Northeast View	9

Revolving Function

It can be completed by keyboard operation only (viz. Alt + direction key). At the same time of pressing ALT key, press down the direction key, you will realize to revolve the processing track.

4.11. System Log Window

This window records the important operation and happened events, not only can you browse the log information which was happened since started, but also you can review the historical records.

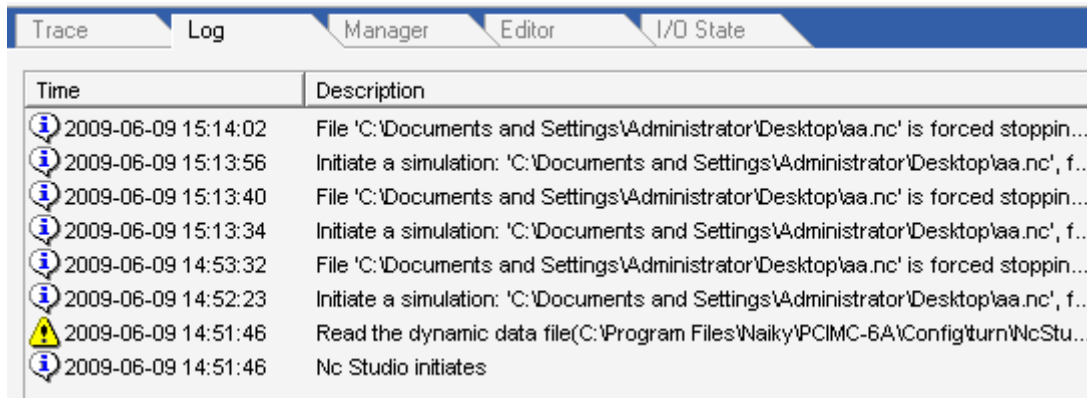


Fig. 4-39 System Log Window

The log information of the system's current record include:

- 1) System starting and exiting
- 2) Information of the start and end in processing automatically.
- 3) Changes of the workpiece coordinate.
- 4) Alarm information of the system.
- 5) Other system information.

Click the window with the right key of mouse, the context menu will be flipped out as shown in the following figure.



Fig. 4-40 Context Menu of System Log Window

Log Clearance Function

Clear the log information which listed in the system log window.

Menu method

Choose "Edit| Clear log" to perform.

Mouse method

Move the mouse into the log window, and click the right key of the mouse. When flip out a pop menu, choose "Clear log" option to perform.

Caution:

Remember to periodically clean up the system log record! Otherwise the file of the system log

record will be too big to affect the function of the system and the responding time.

Following options will be marked as " ✓ " in front the text when chosen. It will be cleared when clicked again, which means unselected.


Only Show Log of this time

User can click the right key of the mouse in the log window which will flip out a pop menu to choose "Show Log of This Time" option. You can also choose "View | Show Log of This Time" by menu method to perform.

To choose this option, user only can see the log of current operation. Otherwise, user can view not only the current log of operation but also all the log before.

Show Information Log


User can click the right key of the mouse in the log window which will flip out a pop menu to choose "Show information item" option. You can also choose "View | Show information item" by menu method to perform.

The information log option would be marked as .

To choose this option, user can view the system start and exit information in the log window which will not be listed when this option unchosen.

Show Warning Log


User can click the right key of the mouse in the log window which will flip out a pop menu to choose "Show warning item" option. You can also choose "View | Show warning item" by menu method to perform.

The warning log option will be marked as .

To choose this option, user can view the warning information occurred during the operation in the log window which will not be listed when this option unchosen.


Show Error Log

User can click the right key of the mouse in the log window which will flip out a pop menu to choose "Show error item" option. You can also choose "View | Show error item" by menu method to perform.

The error log option will be marked as .

To choose this option, user can view the the wrong information occurred during the operation in the log window which will not be listed when this option unchosen.

Prompt:

Basic log information diagram mark is .

4.12.Procedure Management Window

The procedure management window mainly used to manage user's processing file. As long as user saves the processing file into a specified folder, NcStudio can manage these files by an inner manager. In the management window, user can complete creating, calling, editing, deleting, renaming, loading procedure file and etc. It is extremely convenient to user.

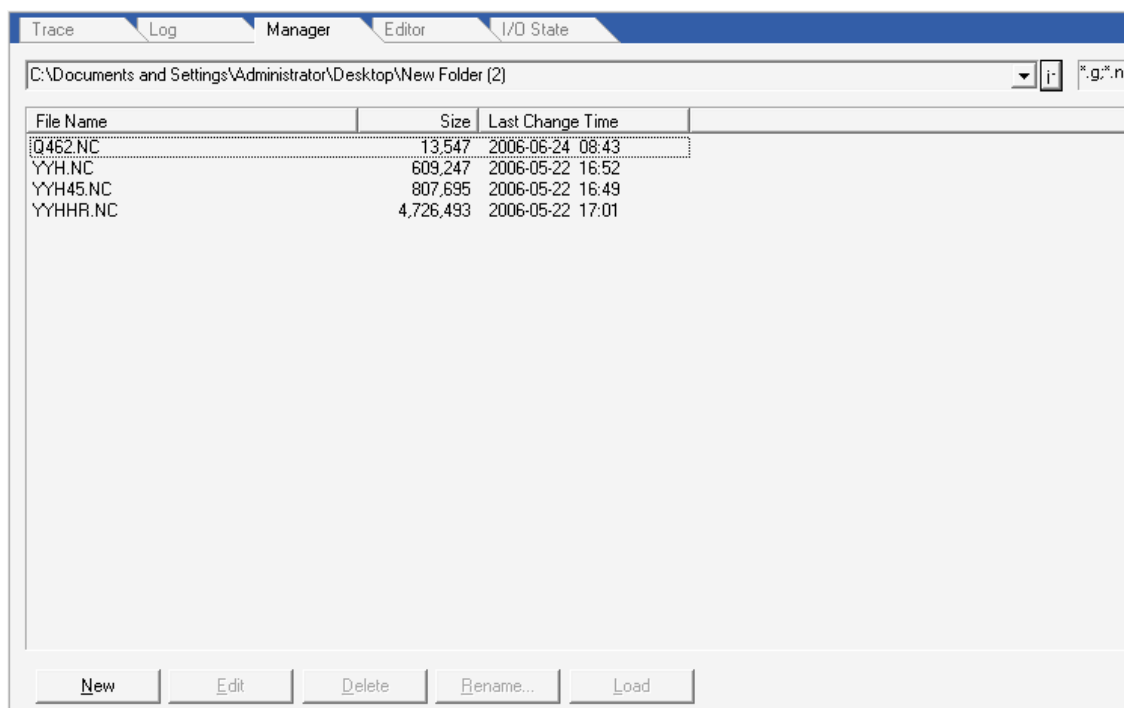


Fig. 4-41 Procedure Manager Window

Click with the right mouse key in procedure management window and a context menu will pop up as shown in fig. 4-42.

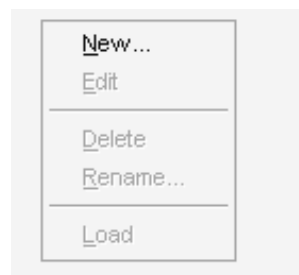


Fig. 4-42 Context Menu of Procedure Management Window

Create New Procedure File

User can create procedure file at any time. There are three ways to achieve this:


First: the user can select “File |New”, (shortcut: Ctrl+N);

Second: click with right key of the mouse in procedure management window, and select “New...” in the popup context menu.

Third: Click the button  under procedure management window.

The system will generate a new processing procedure file “Untitled1.nc” automatically and then the user can modify the saving position himself.




Click the rear button  to select a saving position.

In addition, the user can also modify the new created file, such as edit, delete, rename, load and etc. The detailed contents see the following chapter.

Open a Procedure File Existed

The “Files List Box” in procedure management window is used to display the processing procedure in current file folder.

User can name the path directly not only in “current processing procedure file folder” box , but also can click the rear button  to select other path from “Files List Box” which contains all processing files under the current path.

The extension name of processing file under current path is shown in the box of “Processing File Extension”, the user can delete or create new extension name. The “File List Box” will list all processing files with corresponding extension name.

Click and open the file to be current processing procedure while the processing files required to be opened is listed in “File List Box”.

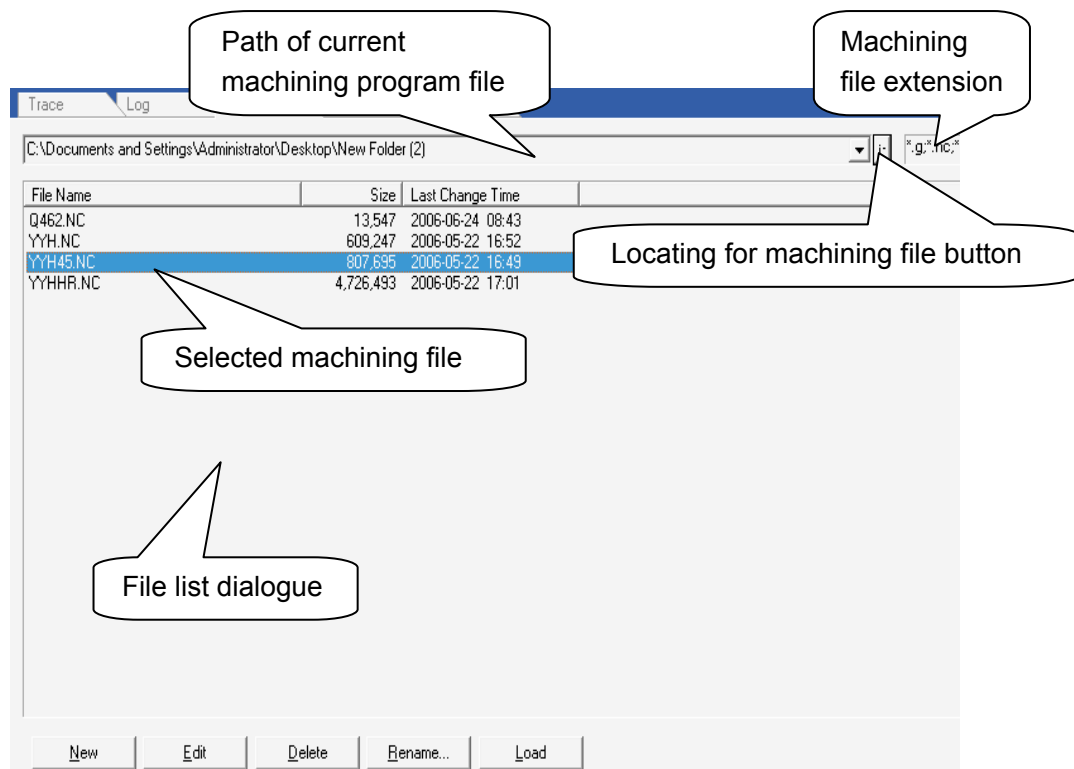


Fig. 4-43 Procedure Management Window

In addition, the user can realize the file opening function through menu "File | Open and Load..."

Edit Procedure File

User can edit procedure file through the following three ways:

First: The user can select "Edit |Edit Processing Procedure";

Second: Click with right key of the mouse in procedure management window, and select "Edit Processing Procedure" in the popup context menu.

Third: Click the button  under procedure management window.

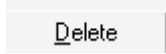
The system will switch to procedure edit window automatically at this time to convenient for the edit operation of user. For detailed edit operation, please refer to procedure edit window.

Delete Procedure File

User can delete procedure file selected through the following three ways:

First: The user can select "Edit |Delete Processing Procedure";

Second: Click with right key of the mouse in procedure management window, and select “Delete Processing Procedure” in the popup context menu.

Third: Click the button  under procedure management window.

The following dialog box will popup at this time to ask that if the processing procedure file should be deleted or if the file is edited or being used by others.

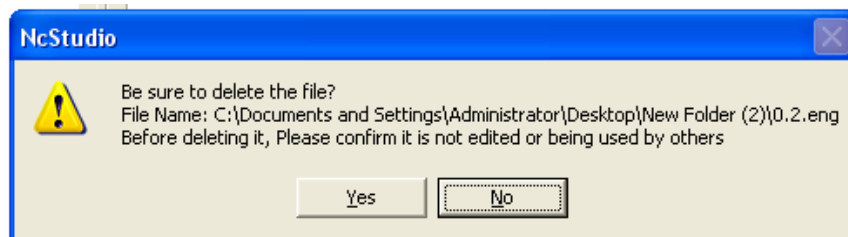


Fig. 4-44 Prompt Window of “Delete Procedure File”

Click “Ok” to delete.

Prompt:

The selected file will be shown in file list box with highlight state (blue).

Rename Procedure File

User can rename procedure file selected through the following three ways:

First: The user can select “Edit |Rename Processing Procedure”;

Second: Click with right key of the mouse in procedure management window, and select “Rename Processing Procedure” in the popup context menu.

Third: Click the button  under procedure management window.

The selected procedure file name is under edit state at this time as shown in the follow figure:

File Name	Size	Last Change Time
Q462.NC	13,547	2006-06-24 08:43
YYH.NC	609,247	2006-05-22 16:52
<u>YYH45.NC</u>	807,695	2006-05-22 16:49
YYHHR.NC	4,726,493	2006-05-22 17:01

Fig. 4-45 Rename Dialog Box

Press Enter key or click other blank area with mouse to complete the rename after input relevant file name.

Load Procedure File

User can load procedure file selected through the following five ways:

First: The user can select “Edit |Open and Load”;

Second: Click with right key of the mouse in procedure management window, and select “Load Processing Procedure” in the popup context menu.

Third: Click the button  under procedure management window.

Fourth: Click open and load button  on toolbar.

Fifth: Click with right key of mouse in automatic mode window, and select “Open and Load” from popup context menu.

4.13.Procedure File Edit Window

The upper of the edit window is the file name which is editing currently. A standard style edit window was as shown in below.

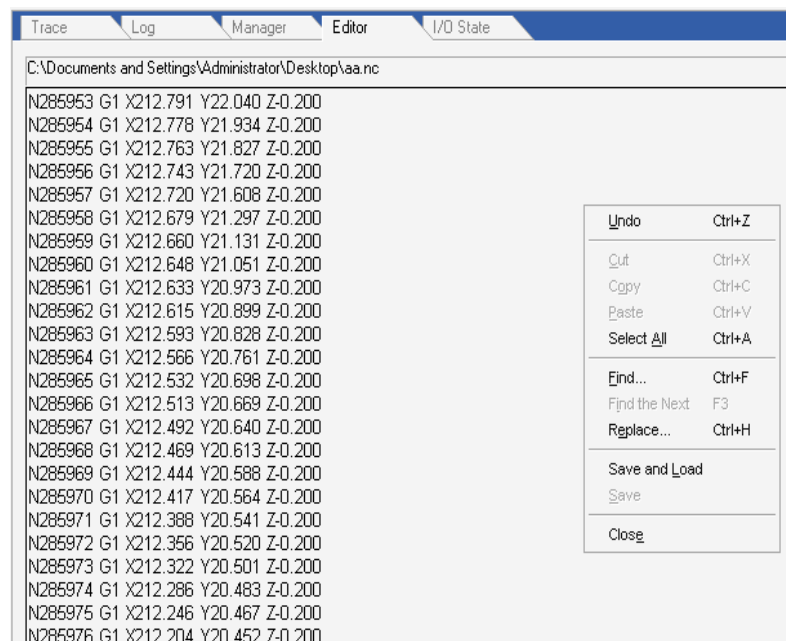


Fig. 4-46 Procedure File Edit Window

The editing ability of this editor is more powerful than Windows default Notebook. Theoretically, this editor can edit a processing procedure which size is over 1000M and satisfy the requirement of editing any complex procedure.

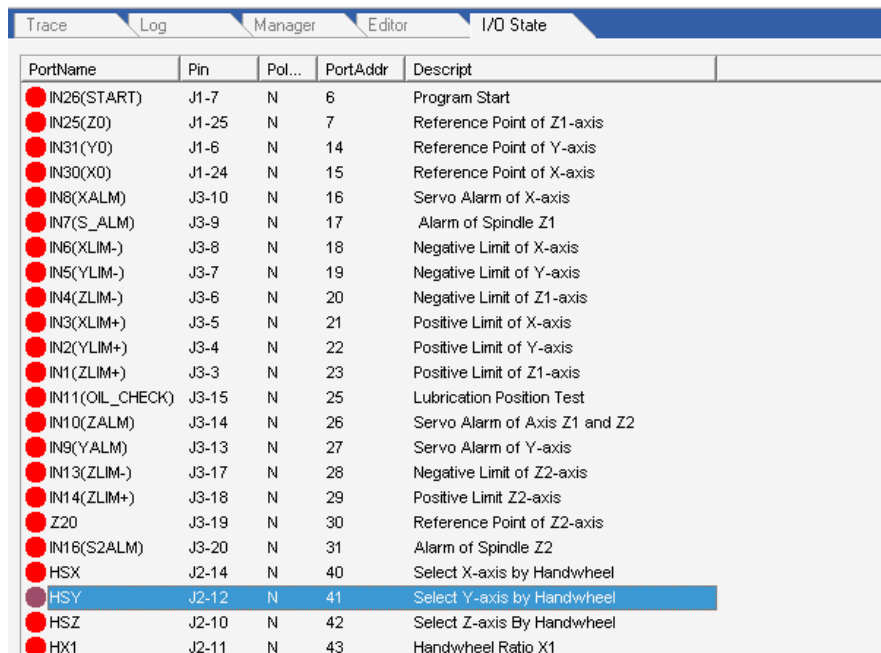
As long as input whatever word into the edit window, A mark with “modified” word would be appeared after the file name which on the top of the window to remind the user that the file

has been modified.

Clicking the right key of mouse in the editor window to popup a context menu, user can select proper operation according to the demand.

4.14.State of Input or Output (I/O) Window

The input/output state window displays the current states of I/O, which is very beneficial to the system monitoring and malfunction checking.







PortName	Pin	Pol...	PortAddr	Descript
IN26(START)	J1-7	N	6	Program Start
IN25(Z0)	J1-25	N	7	Reference Point of Z1-axis
IN31(Y0)	J1-6	N	14	Reference Point of Y-axis
IN30(X0)	J1-24	N	15	Reference Point of X-axis
IN8(XALM)	J3-10	N	16	Servo Alarm of X-axis
IN7(S_ALM)	J3-9	N	17	Alarm of Spindle Z1
IN6(XLIM-)	J3-8	N	18	Negative Limit of X-axis
IN5(YLIM-)	J3-7	N	19	Negative Limit of Y-axis
IN4(ZLIM-)	J3-6	N	20	Negative Limit of Z1-axis
IN3(XLIM+)	J3-5	N	21	Positive Limit of X-axis
IN2(YLIM+)	J3-4	N	22	Positive Limit of Y-axis
IN1(ZLIM+)	J3-3	N	23	Positive Limit of Z1-axis
IN11(OIL_CHECK)	J3-15	N	25	Lubrication Position Test
IN10(ZALM)	J3-14	N	26	Servo Alarm of Axis Z1 and Z2
IN9(YALM)	J3-13	N	27	Servo Alarm of Y-axis
IN13(ZLIM-)	J3-17	N	28	Negative Limit of Z2-axis
IN14(ZLIM+)	J3-18	N	29	Positive Limit Z2-axis
Z20	J3-19	N	30	Reference Point of Z2-axis
IN16(S2ALM)	J3-20	N	31	Alarm of Spindle Z2
HSX	J2-14	N	40	Select X-axis by Handwheel
HSY	J2-12	N	41	Select Y-axis by Handwheel
HSZ	J2-10	N	42	Select Z-axis By Handwheel
HX1	J2-11	N	43	Handwheel Ratio X1

Fig. 4-47 State of I/O Window

Caution:

The contents of that window are different according to different panel-card set and the different requirement of the user. Here the content provided is for reference only.

The icon before each interface means:

- : Green solid point means that there is signal input at this interface.
- : Red solid point means that there is no signal input at this interface.
- : Green hollow point means that there is signal output at this interface.
- : Red hollow point means that there is no signal output at this interface.

5. NcStudio™ Menu System

5.1. "File" Menu

This menu includes the following options:

Open and Load...	Ctrl+O
Unload	Ctrl+U
New	Ctrl+N
Open and Edit...	Ctrl+E
Edit Loaded File	Ctrl+P
Save	Ctrl+S
Save as...	
Save and Load	
Close	
Config	▶
Recent Loaded File	▶
Recent Edited File	▶
Exit	

Fig. 5-1 File Menu

Open and Load

The 'open and load' menu item whose shortcut key is "CTRL+O", is used to open procedure file existed and kept in disk and to popup the dialog box of "Open and Load" as shown in the following figure:

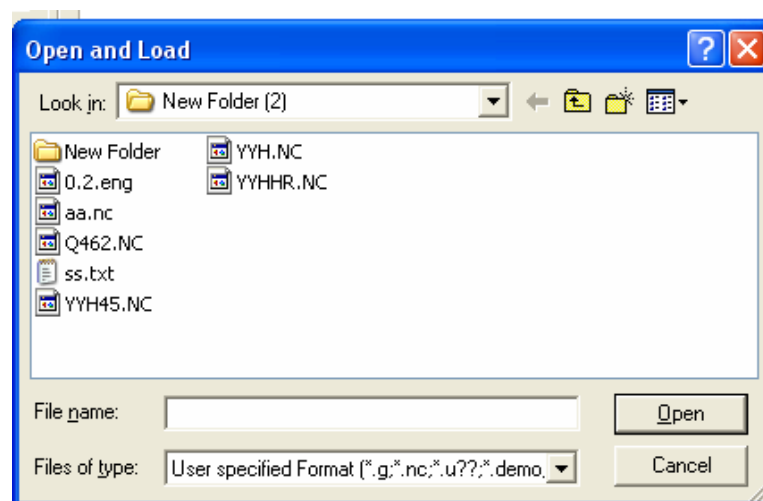
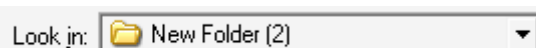


Fig. 5-2 Open and Load Dialogue

Click the rear part of button



, to review

processing file under other path; select the file and click button “Open”, and then the toolbar of system main window will show the file names for current processing procedures.

Unload

Unload the current processing procedure.

New Processing Procedure

It is used to create a new processing procedure.

Select this option and the procedure edit window will be active to allow the user edit the procedure at this window; click with the right key of mouse to popup a context menu, and then select “save” to complete.

Open and Edit

This menu item is used to open procedure file that saved in disk and to load it into edit window where can edit a file.

Edit Current Processing Procedure

The function is used to edit processing procedure loaded currently.

Save

The function is used to save processing procedure edited.

Save As

This function can save the current procedure files in the editor window into disk with another name.

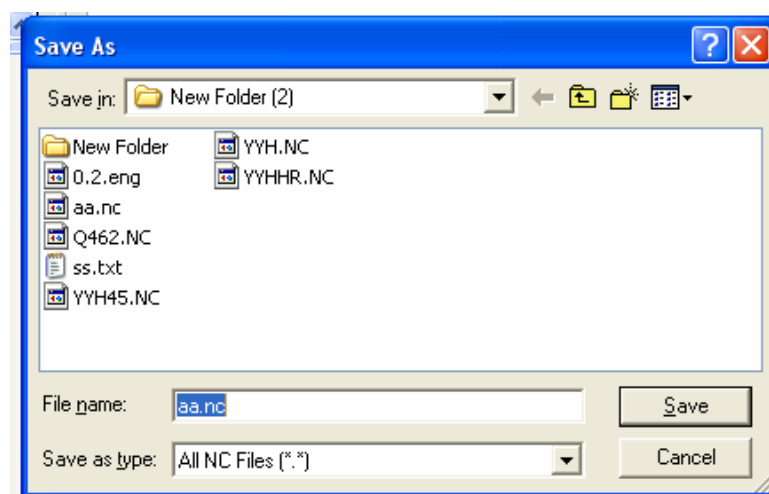


Fig. 5-3 'Save As' Dialog

Save and load

The function is used to save and load the procedure file edited currently into current processing procedure.

Close

The function is used to close the current procedure files when being edited.

Configuration

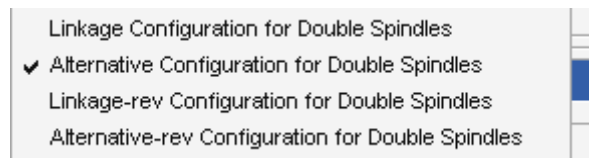


Fig. 5-4 Configuration Options

The user can select corresponding configuration as required. For detailed configuration, please refer to Manufacturer Manual.

Recent Loaded Processing Procedure

This menu pops out a submenu which shows the name of processed files which were loaded recently. If user wants to load any file again, clicking the name of the document directly will load it quickly.

Recent Edited Processing Procedure

This menu pops out a submenu which shows the name of processed procedure which were edited recently. If user wants to edit again, clicking the name of the document directly will load it quickly.

Exit

It is used to close Ncstudio™ system.

If the processing procedure is edited by user and is not saved, a dialog box shall popup for saving or not. If you want to save, click "Yes", otherwise, click "No", and if you want to exit the function, click "Cancel". If the user is carrying out automatic processing, the system will prompt the user to end the current processing task and then exit the system.

5.2. "Edit" Menu

In this menu, the items will be changed along with the current active window of the second window (including processing track, system log, procedure management, procedure edit and I/O state window).

5.2.1. Edit Menu When Processing Track Window Activated

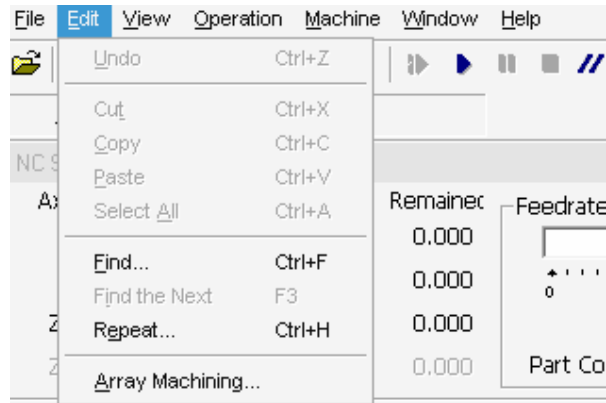


Fig. 5-5 Edit Menu 1

Clear View

Refer to “Clear View” in article 4.10 Processing Track Window.

Array Machine

The function is used to have array machining for the same processing procedure. The following dialog box will popup by selecting the option:

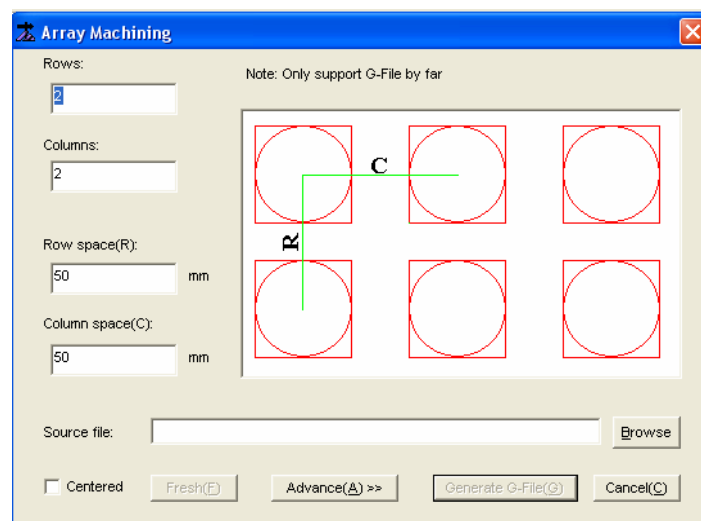


Fig. 5-6 Array Machine Dialog

Click and browse the button  to select array machining procedure as required.

Set column and row number, row space (the spacing between two origins of workpieces as the R length in the above mentioned figure), column space (the spacing between two origins of workpieces as the C length in the above mentioned figure), and click the button “Generate Processing File” to complete arraying; the new generated processing file will be

load into NcStudio system automatically.

Click the button “Advanced Mode” to set different row space and column space and the corresponding dialog box is shown as below:

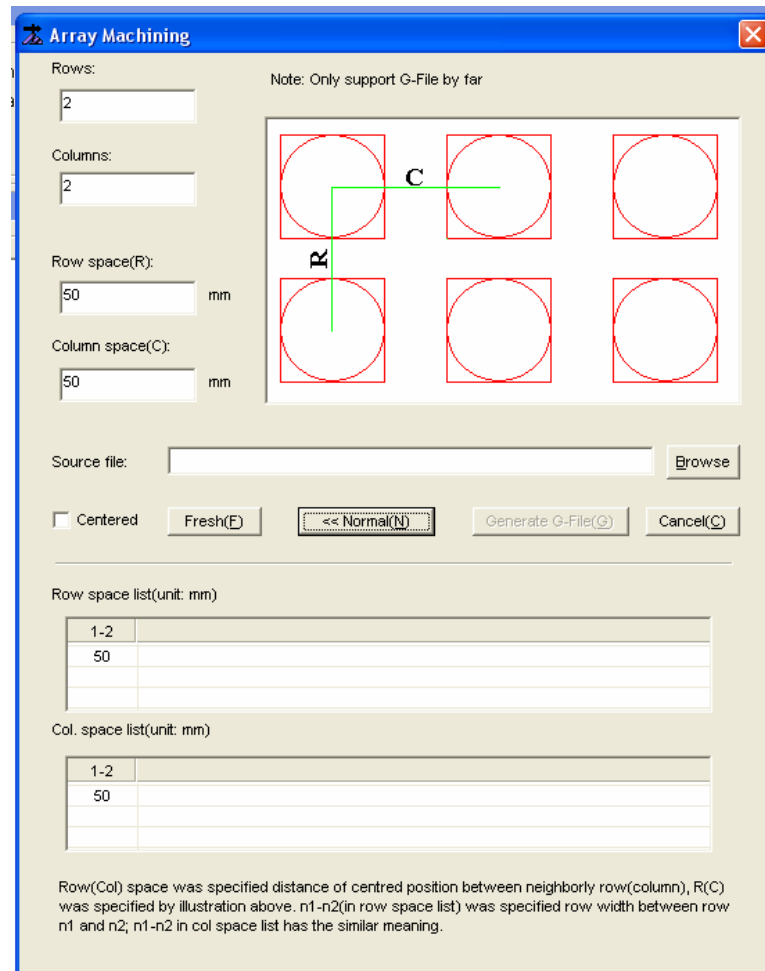


Fig. 5-7 Array Machine

“Row Space Table” and “Column Space Table” is used to set the space for row and column respectively.

5.2.2. Edit Menu When System Log Window Activated

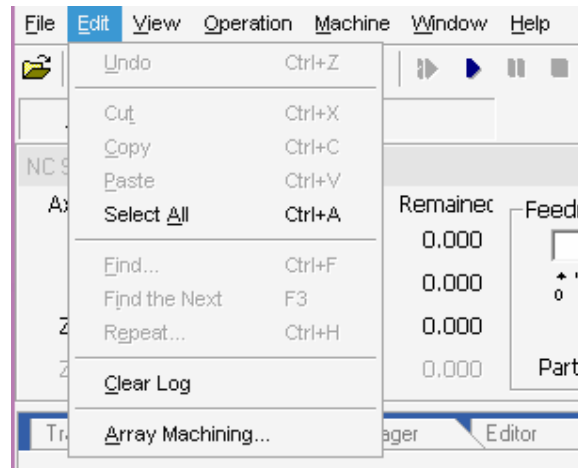


Fig. 5-8 Edit Menu 2

Clear Log

Refer to “Clear Log” in article 4.11 System Log Window.

Array Machine

Refer to “Array Machine” in article 5.2.1.

5.2.3. Edit Menu When Procedure Management Window Activated

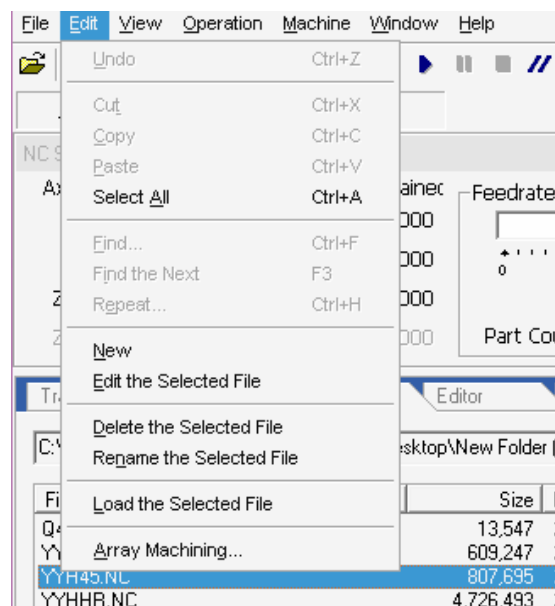
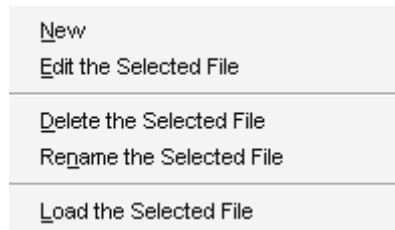


Fig. 5-9 Edit Menu 3



Please refer to the introduction of Procedure Management Window in 4.12.

5.2.4. Edit Menu When Procedure Edit Window and I/O Status Window Activated

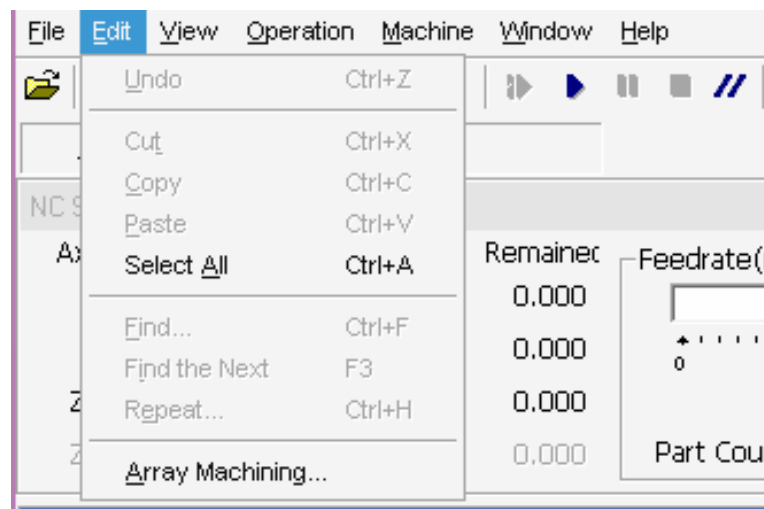


Fig. 5-10 Edit Menu 1

Array Machine

Refer to “Array Machine” in article 5.2.1.

5.3. "View" Menu

The items in “View” menu will be changed along with the current active window of the second window (including processing track, system log, procedure management, procedure edit and I/O state window).

5.3.1. “View” Menu When Procedure Management, Procedure Edit and I/O Status Window Activated

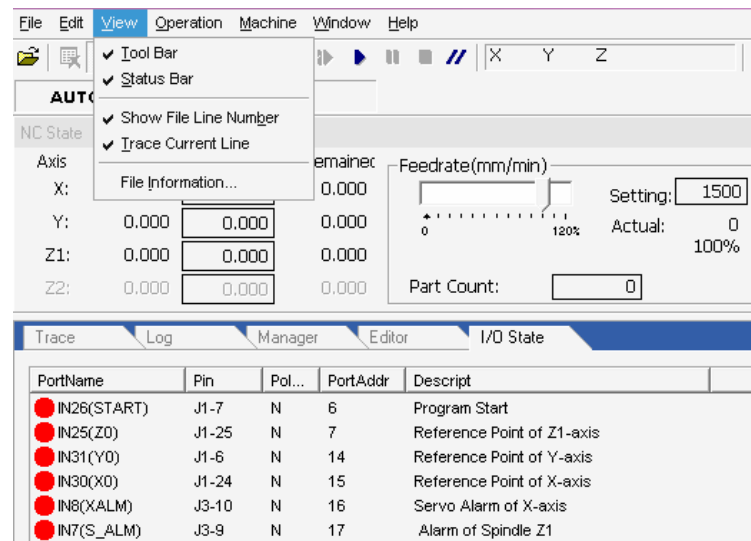


Fig. 5-11 “View” Menu 1

Show Row Number of Processing Procedure

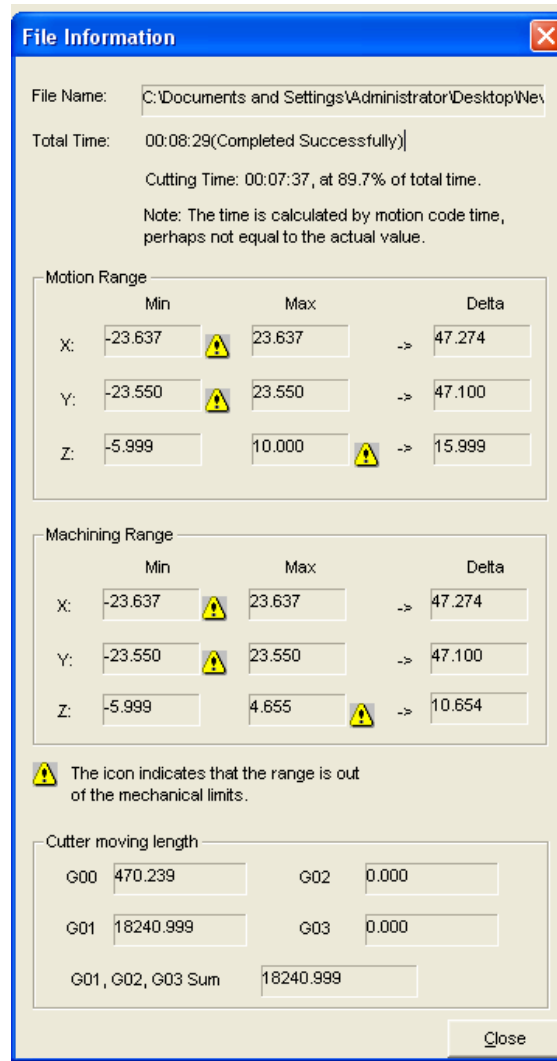
The function is used to show or hide the processing procedure row number in automatic window. The “View” menu is effective only when it is activated in automatic window.

Trace Current Row of Processing Procedure

The function is used to trace the row number of current procedure in automatic window during processing.

Processing Procedure Information

Click “Processing Procedure Information” to popup the following dialog box:



File Information

File Name: C:\Documents and Settings\Administrator\Desktop\New

Total Time: 00:08:29(Completed Successfully)

Cutting Time: 00:07:37, at 89.7% of total time.

Note: The time is calculated by motion code time, perhaps not equal to the actual value.

Motion Range			
	Min	Max	Delta
X:	-23.637	23.637	47.274
Y:	-23.550	23.550	47.100
Z:	-5.999	10.000	15.999

Machining Range			
	Min	Max	Delta
X:	-23.637	23.637	47.274
Y:	-23.550	23.550	47.100
Z:	-5.999	4.655	10.654

The icon indicates that the range is out of the mechanical limits.

Cutter moving length	
G00	470.239
G01	18240.999
G02	0.000
G03	0.000
G01, G02, G03 Sum	18240.999

Close

Fig. 5-12 Processing Procedure Statistical Information

The dialog box showed the statistic information in automatic processing, such as the processing time, processing range etc. It would be know exactly and quickly when combining with the function of simulation.

Time Spent

Statistic for total machine tool processing time and for machine tool cutting time

Motion Range

The motion range means the max and min workpiece coordinate values of machine tool in processing cycle.

Processing Scope

The processing scope means the max and min workpiece coordinate values of machine in actual cutting cycle.

5.3.2. "View" Menu When Processing Track Window Activated

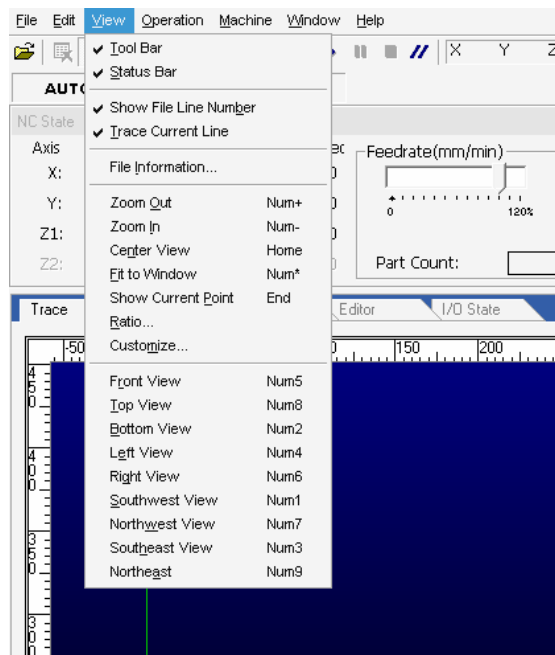


Fig. 5-13 "View" Menu 2

Zoom out, zoom in, center view, fit to window size, show current point, show ratio, customize, front view, top view, bottom view, left view, right view, southwest view, northwest view, southeast view, northeast view.

For above mentioned items, please refer to the introduction of processing track window in section 4.10.

5.3.3. "View" Menu When System Log Window Activated

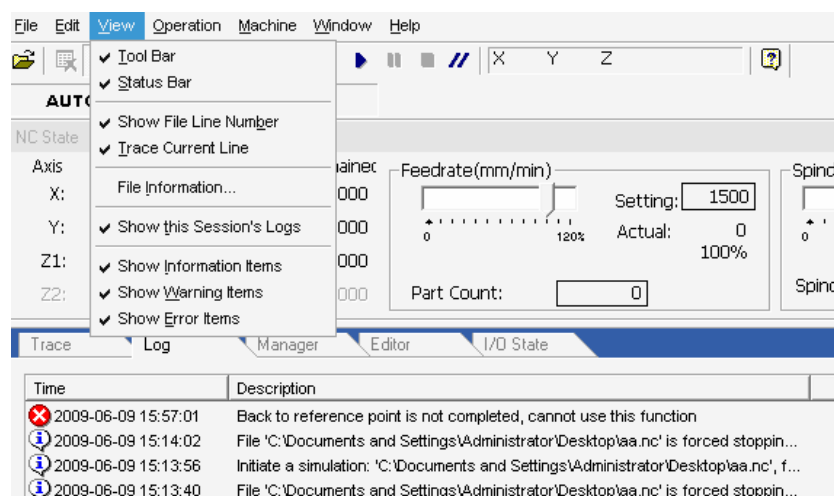


Fig. 5-14 "View" Menu 3

Only show this session's logs, show information items, show warning items, and show error items

Please refer to the introduction of log information window in section 4.11.

5.4. "Operation" Menu

"Operation" menu consists of the following items:

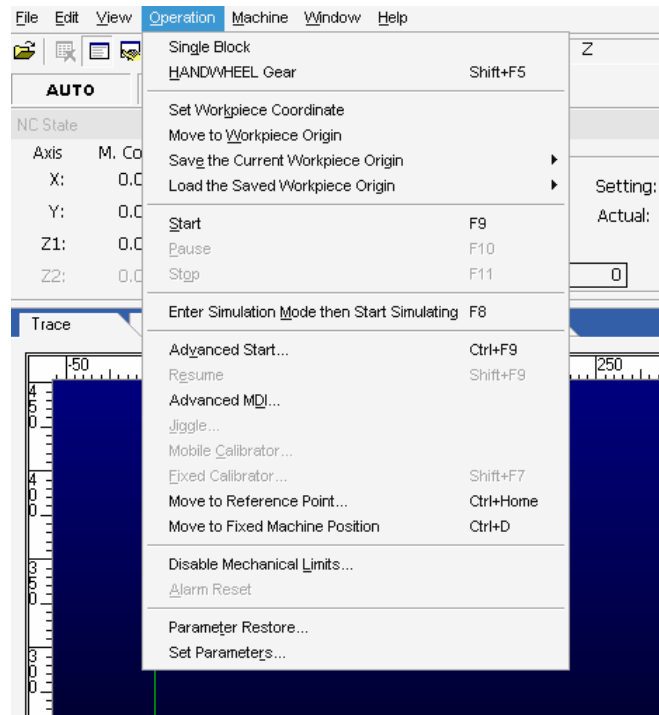




Fig. 5-15 Operation Menu

5.4.1. Single Block

While single stepping mode was selected, operating one order each time would be to the Pause mode. User has to click "start" button  to execute the next order to continue. When next order have finished, the system will be to the Pause mode again.

User can set the processing assignment as a single step mode in order to provide a fine support for diagnosis mistake and malfunction restoring.

5.4.2. Handwheel Guide

When it is operated with handwheel guide under auto mode, user shall click button  for machining, then the system will execute the processing procedure as the rotation of handwheel. Therefore the procedure processing will stop when the handwheel is stopped and the processing speed varies along with the speed of handwheel.

User can select the function to help the user to judge that processing procedure is correct prior to carrying out processing.

5.4.3. Set Workpiece Coordinate Offset

Select the option to popup the following dialog box:

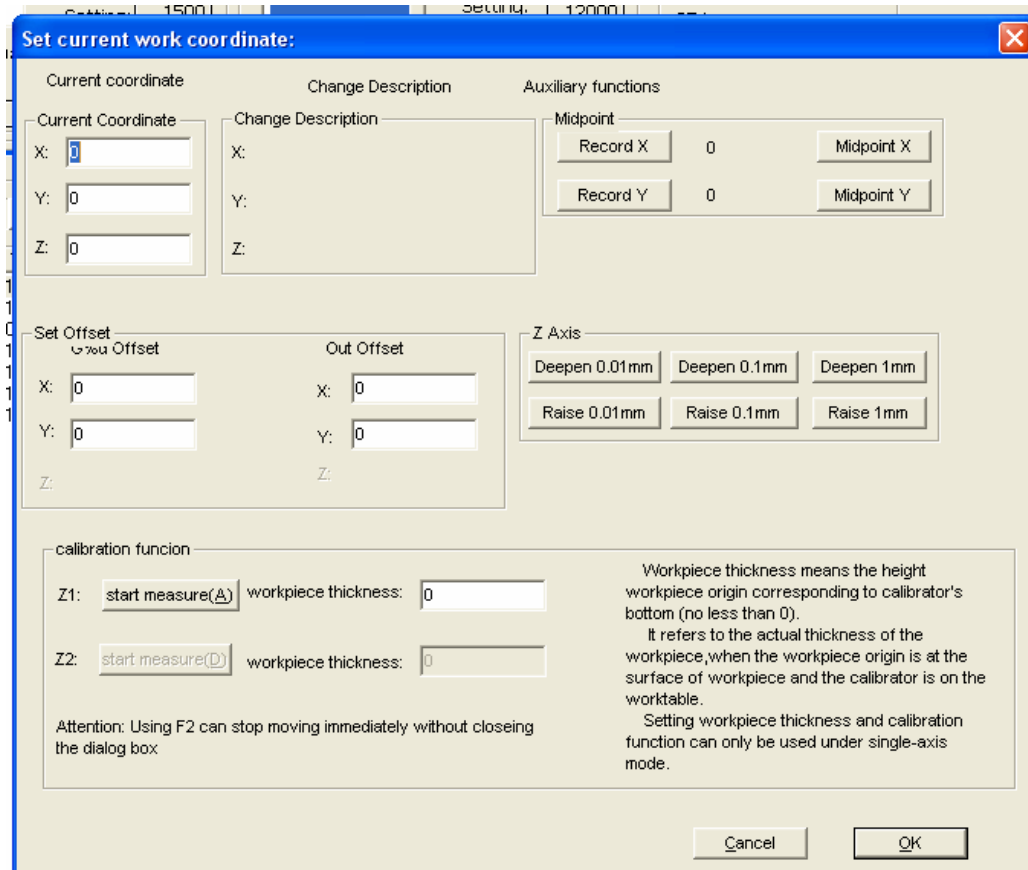


Fig. 5-16 Set Workpiece Coordinate Offset

For detailed contents, please refer to Calibration Operation Window in section 4.9.

5.4.4. Back to Workpiece Origin

Select "Set Workpiece Origin" and Z axis will rise up to safety height when the tool point is lower than safety height and then the X and Y axis will be returned to workpiece origin together firstly and then Z axis will be lower to safety height.

Users also can select the button  which located in the Tool Bar to realize this function.

5.4.5. Save the Current Workpiece Origin

This function is used to save the current workpiece origin into the file of processing system.

User can save the workpiece origin as well as save procedure name together to avoid procedure confusion after several times of workpiece origin saving and to help the operator to find the saved workpiece origins. The function can save ten coordinate data totally.

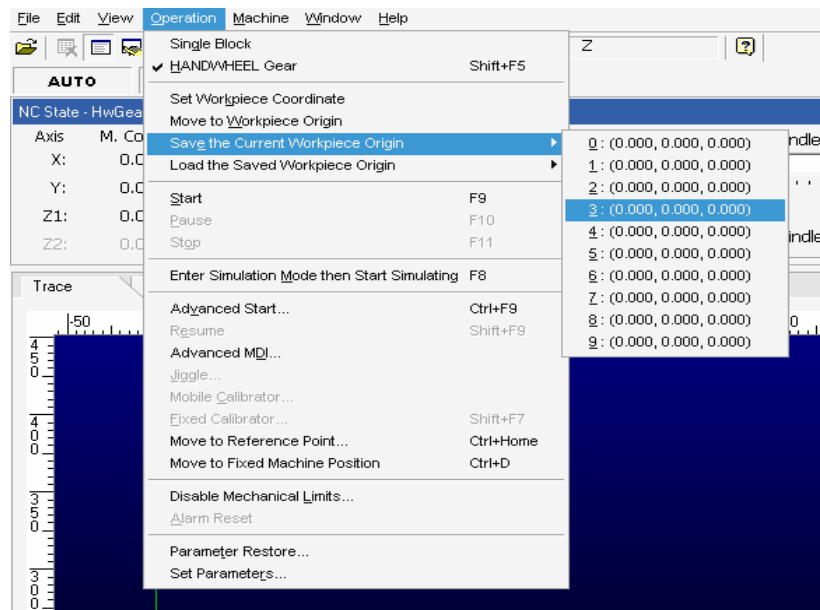


Fig. 5-17 (Save the Current Workpiece Origin)

5.4.6. Load the Saved Workpiece Origin

The function is used to read coordinate value of the saved workpiece origin.

5.4.7. Start


Click Start option and the system will enter into automatic processing mode. If the system is under simulation status, it will execute the processing procedure with simulation mode.

User also can choose the button  on the tool bar to operate this function.

5.4.8. Pause

In automatic processing, selecting the menu item of “pause”, the machine tool processing will pause and raise the cutter then turn into the state of “Auto | Pause”. If you want to keep on processing, you can select “Start”.

If the system is in the state of simulation it will be paused by selecting Pause. And if you want to keep on simulating, you can select Start.

User also can select the button  on the tool bar to operate this function.

5.4.9. Stop

In automatic processing, selecting this menu, the machine tool processing will stop and raise the cutter, and then the system will terminate the whole processing task, and enter into the state of "Auto | Idle". This method is to break off the processing procedure normally in processing.

If the system is under simulation status, the system will be paused and enter to the status of "Auto | Idle" after selecting the menu of "stop". If the user wants to re-simulate, you can continue by selecting the options "start", "Advanced start", "continue from the breakpoint" and so on.

User also can select the button  located on the tool bar to operate this function.

5.4.10. Enter Simulation Mode and Start Simulating

Select this menu, machine tool will execute simulating in high speed automatically from the first row which the function similar to the display function of CNC but priority to.

Running under the mode of simulating, the system will not drive the machine tool to do the relative actions but to show the processing trace of the cutter in high speed in the trace window. By simulating, user can foresee that the moving form the machine tool will do to avoid damaging the machine tool when program processing procedure. By simulating you can also know some additional information.

Once the simulation started, the menu was replaced by "Stop Simulating and Leave Simulation Mode". If you execute this function, simulating will be terminated immediately.

User also can select the button  located on the tool bar to operate this function.

5.4.11. Advanced Start

Selecting this function, the system will pop out a dialog box of "Start (with advanced options)", as showed in below.

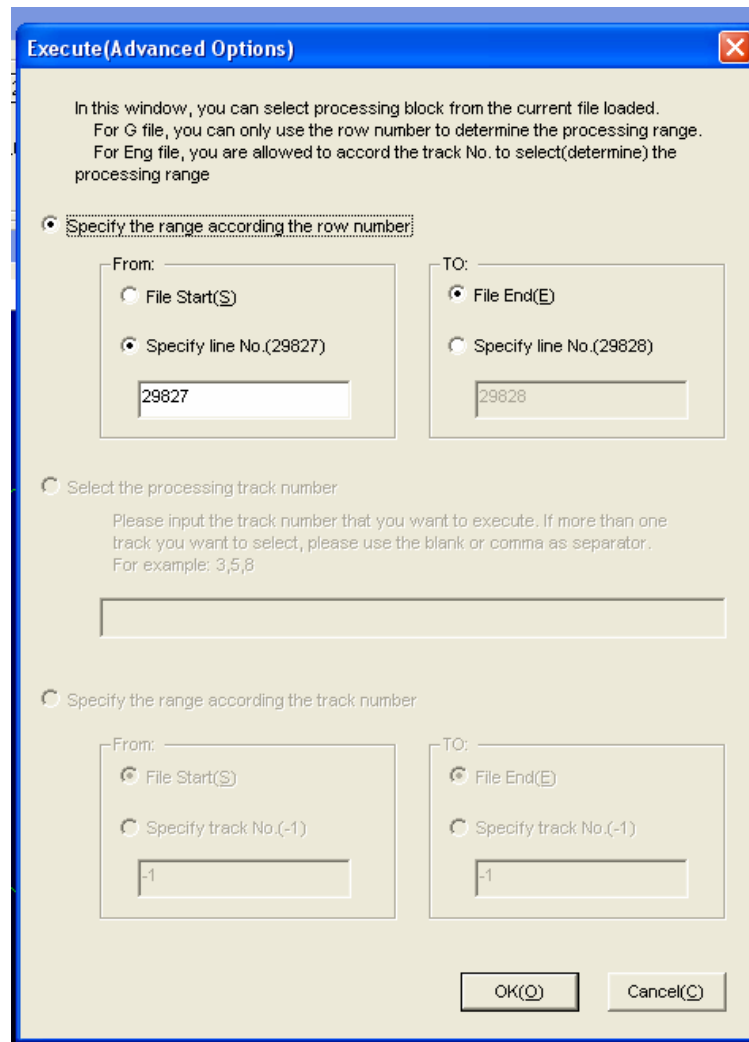
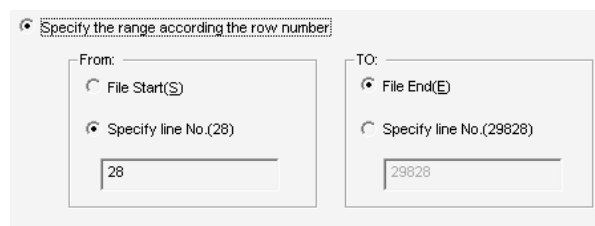


Fig. 5-18 Advanced Start

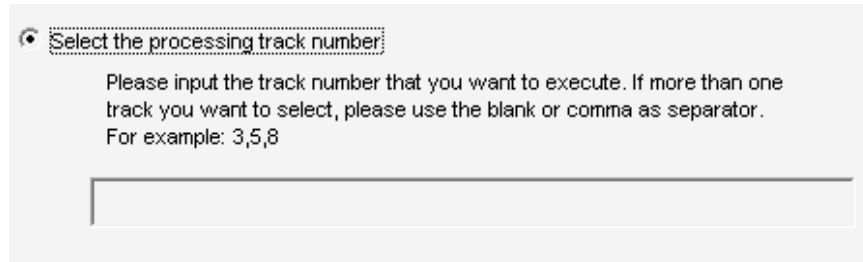
The function is used to execute the assigned section in whole processing procedure. The user can carry out processing in accordance with selection scope of file types in dialog box.

Assign Range by Row Number



User can select any section for machining in the program according to row number.

Assign Processing Path Number

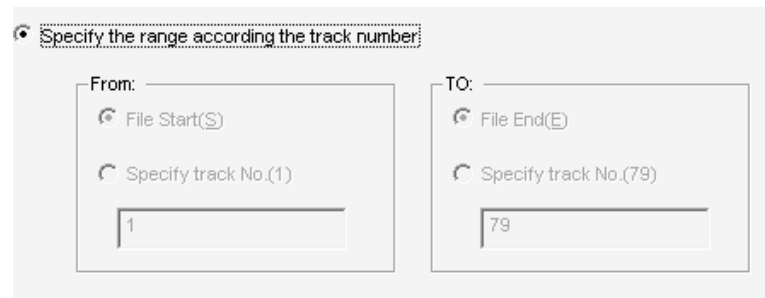


Select the processing track number:

Please input the track number that you want to execute. If more than one track you want to select, please use the blank or comma as separator.
For example: 3,5,8

User can select any section for machining in the program according to processing path number.

Assign Range by Path Number



Specify the range according the track number:

From:

☒ File Start(S)

☐ Specify track No.(1)

1

TO:

☒ File End(E)

☐ Specify track No.(79)

79

User can select any section for machining in the program by assigning processing path range.

5.4.12. Breakpoint Resume

When you execute this function, system will execute from the breakpoint that is broken off last time.

User can select the function after unexpected power off or emergency stop and then the machine tool will move to breakpoint rapidly for processing again which will save user's processing time.

User also can select the button  located on the tool bar to operate this function

5.4.13. Execute Processing Instruction

The system will pop out the "Advanced MDI" dialogue box when the option is selected. The dialogue includes "Rectangle Mill", "Rectangle Frame Mill", "Round Mill", "Round Frame Mill" and "MDI". As long as customer input some parameters in the previous four windows, user can complete the function of milling or drawing.

Rectangle Mill Window

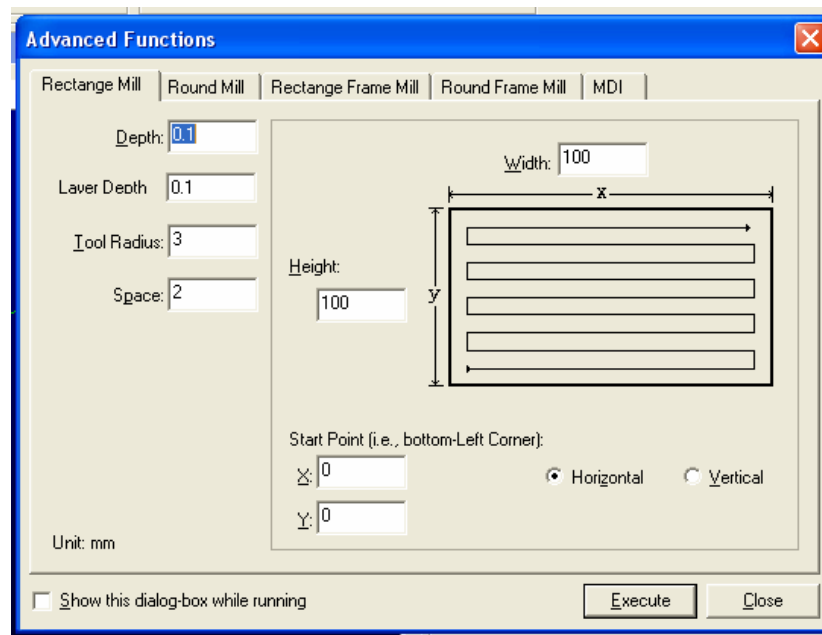


Fig. 5-19 Rectangle Mill Window

Round Mill Window

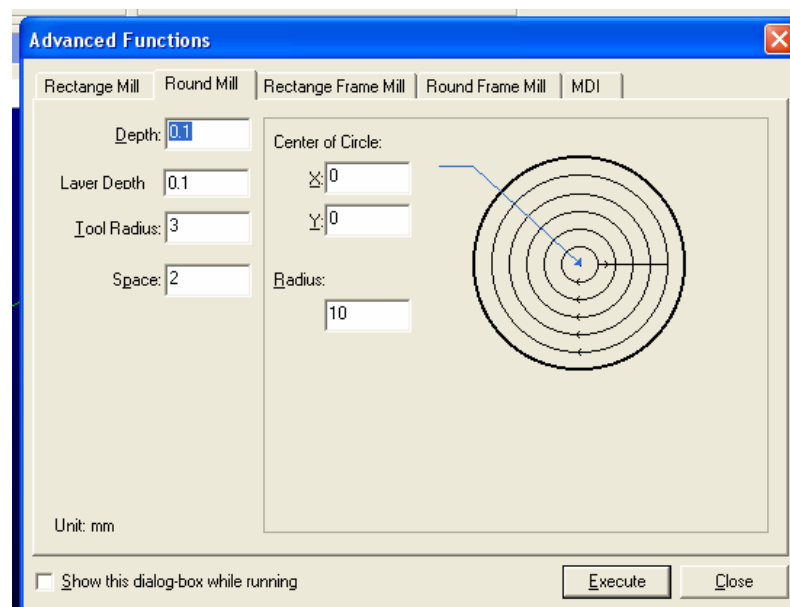


Fig. 5-20: Round Mill Window

Rectangle Frame Mill Window

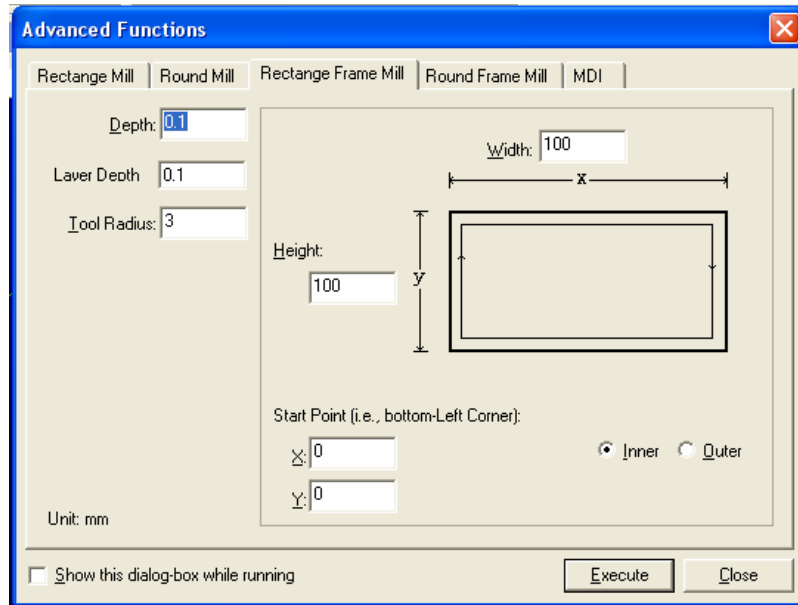


Fig. 5-21 Rectangle Frame Mill Window

Round Frame Mill Window

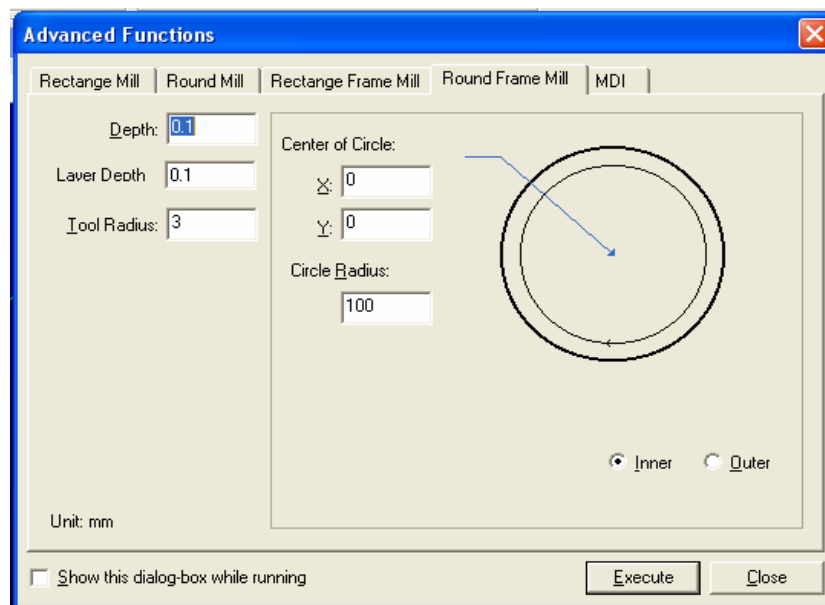


Fig. 5-22 Round Frame Mill Window

MDI window

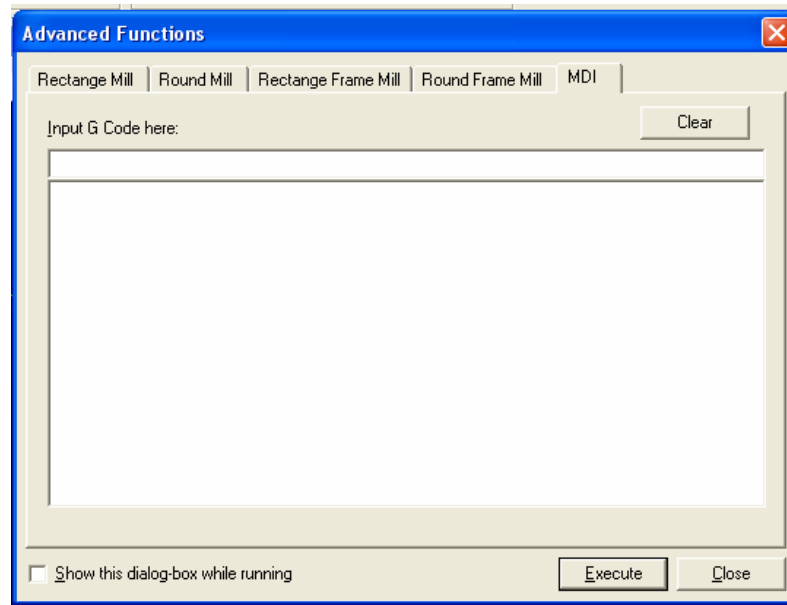


Fig. 5-23 MDI Window

The system will execute the instruction entered immediately by inputting standard instruction (G instruction, T instruction, M instruction) into the editor box of MDI window, and pressing “Enter” key. Meanwhile, it would record history input instructions in the List Box for user’s recall.

The semicolon “;” should be used to separate each instruction while there are many instruction entered. When incorrect instruction is inputted, the system will sent out prompt information.

Selected this option “Show this dialog-box while running”, the system can display the MDI window when executing commands. It is convenient for user to modify, call the parameter of default processing procedure file and input G command immediately.

5.4.14. Jiggle

This function is only valid under the state of pause in the course of automatic process. This function is used to realize the tiny regulation of the depth without breaking off processing cycle.

The result of jiggle operation is only effective in the current processing task. Therefore the jiggle operation will be invalid after “Stop” function executed and “Start” or “Breakpoint Resume” function used again.

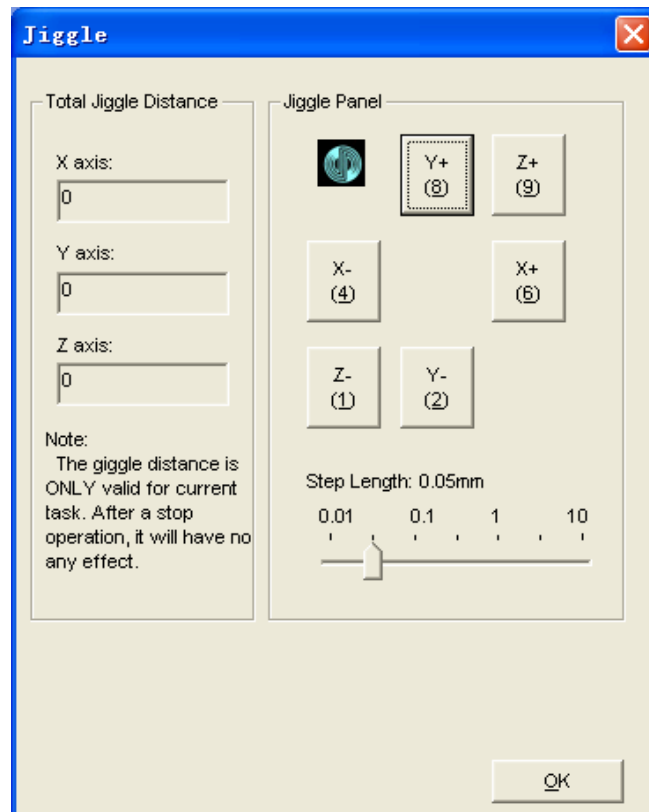


Fig. 5-24 Jiggle Dialog

5.4.15. Back to Machine Reference Point

Select the menu to popup the following dialog box:

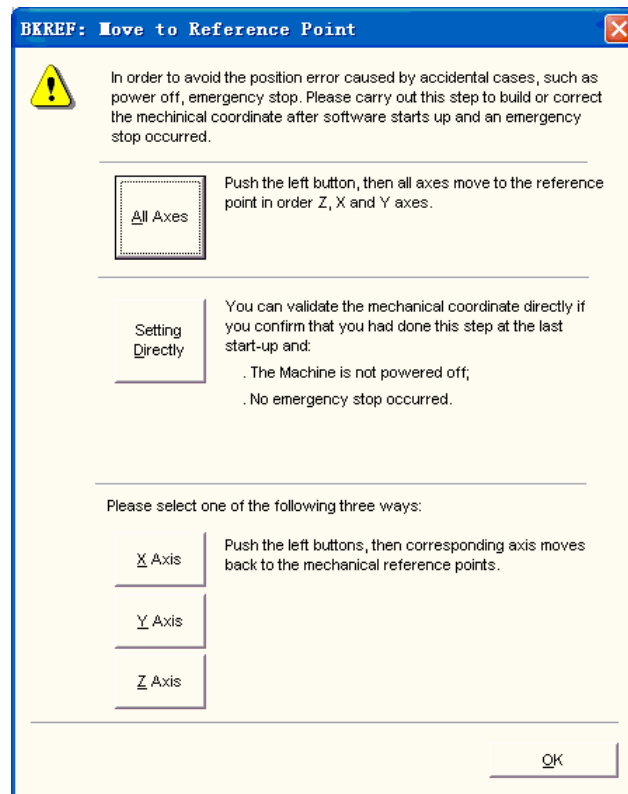
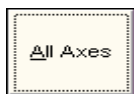
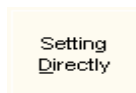


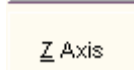
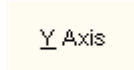
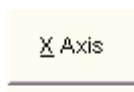
Fig. 5-25 Back to Machine Reference Point Dialogue



: Push the left button, all axes will move to the machine reference point in order (Z axes, XY axes, the order of Z1 and Z2 can be set by manufacturer)



: set the current X, Y coordinate to be mechanical coordinate and confirmed that current position is coincided with mechanical coordinate. If the machine tool was closed or stopped urgently, you are recommended not to execute the operation. The thing that should pay attention to is Z axis can not be set directly.



: return back each axis to corresponding mechanical origin. The corresponding mechanical coordinate can be reset only after Z1 and Z2 axis turned back to mechanical origin.

Axis	M. Coord.	W. Coord.	Remainder
✱ X:	0.000	<input type="text" value="0.000"/>	0.000
✱ Y:	0.000	<input type="text" value="0.000"/>	0.000
✱ Z1:	0.000	<input type="text" value="0.000"/>	0.000
✱ Z2:	0.000	<input type="text" value="0.000"/>	0.000

When all axis returned back to mechanical origin, “✱” mark will be presented at the front of each axis in NC bar.

There are several means to activate the interface of “Back to Machine Reference Point” function:

- 1) When the software is rigid Starting up.
- 2) Menu selection: “Operation” —> “Back to Machine Reference Point”.
- 3) Shortcut key “Ctrl+Home”.

5.4.16. Move to Fixed Machine Position

The machine tool will move to fixed machine position when the function is selected. Setting a proper fixed machine position value, the user can replace cutter or workpiece conveniently.

For fixed machine coordinate, please refer to parameter “N4210”, “N4211”, “N4212”.

5.4.17. Disable Mechanical Limits

The system will mask the limit bit temporarily when encountered a hard limitation. The system will mask the function to avoid warning. The user can recover its normal position by moving the machine tool through manual window to disengage limit switch, as shown in below:

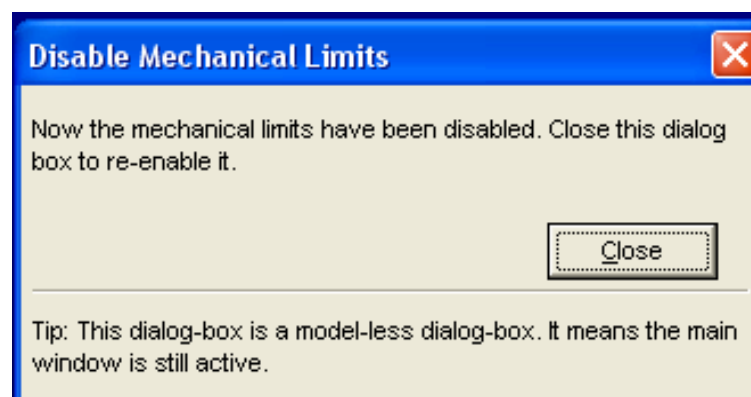


Fig. 5-26 Disable Mechanical Limits

Caution:

Please note your hand moving direction when disable mechanical limits, or the machine tool may be damaged!

5.4.18. Alarm Reset

The system will alarm when detected that the workbench's travel path off the soft limitation. To choose this option can let the workbench back to IDLE state.

5.4.19. Auto Backup Parameters

This function is used to backup parameter automatically. Select the option, a dialog box will popup:

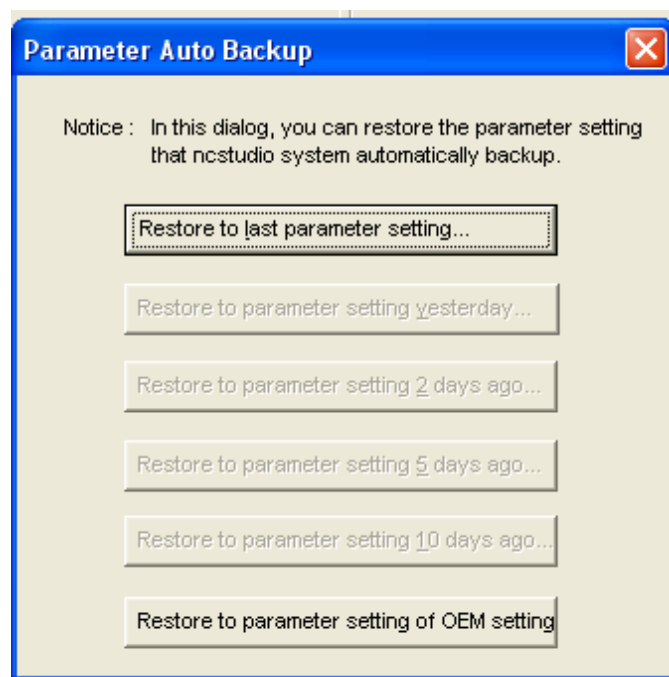


Fig. 5-27 Auto Backup Parameter

User can recover all automatic backup parameter as required. The system provide user with six kinds of parameter backup consisting of “Last Time”, “Yesterday”, “The Day Before Yesterday”, “Five Days Ago”, “Ten Days Ago”, “Factory Parameter”.

5.4.20. Parameter Setting

The function is used to open parameter window for parameter setting, for details please refer to Chapter 6.

5.5. “Machine” Menu

“Machine” menu contents the following options:

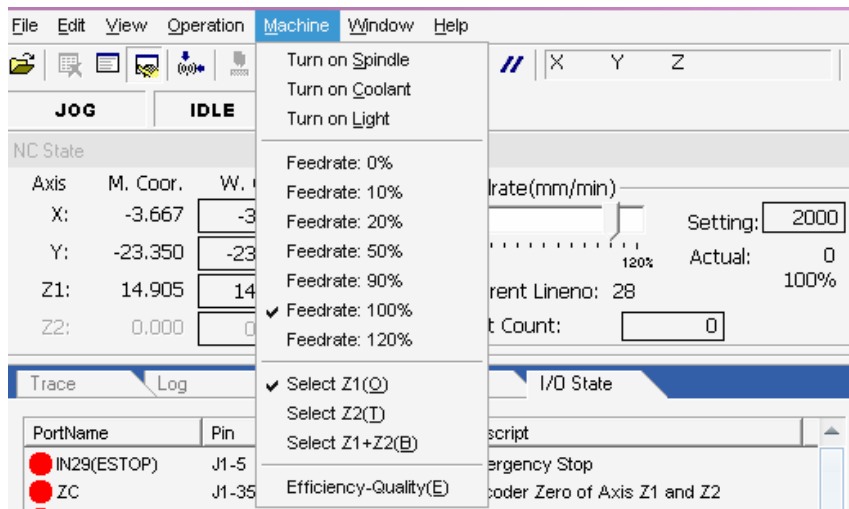


Fig. 5-28 "Machine" Menu

5.5.1. Spindle Starting

The function is used to control the starting of spindle.

5.5.2. Coolant On

The function is used to control the starting of coolant pump.

5.5.3. Lighting

The function is used to turn on the lamp on machine tool.

5.5.4. Feederate

Feederate: 0%, 10%, 20%, 50%, 90%, 100%, 120%

The function is used to adjust the feedrate to 0%, 10%, 20%, 50%, 90%, 100%, 120%. The function is equivalent to the feedrate adjusting button located on NC information bar.



Fig. 5-29 Feedrate Adjusting Button

5.5.5. Select Z1 Axis, Z2 Axis and Z1+Z2 Axes

The machine tool consists of four axes: Z1 Axis, Z2 Axis, Z1 and Z2 Axes. And there are three modes for Z axis: Z1 axis, Z2 axis and Z1+Z2 axes, among which, "Z1+Z2" mode is only for linkage mode. Under single axis mode (Z1 mode or Z2 mode), machine tool will move a single axis only; under double axes mode (Z1+Z2 mode), the machine tool will move two Z axes together with the same actions.

Double Spindle Linkage Configuration

When switching “Select Z1 Axis” to “Select Z2 Axis” under the configuration, the machine tool will be immobility, namely the heights of both Z1 and Z2 remain the same separately.

When switching to “Select Z1+Z2 Axes” under the configuration, the machine tool is movable if the Z1 workpiece coordinate is not in compliance with Z2 workpiece coordinate. The machine tool shall be immobility; when the machine tool is stillness, the workpiece coordinates for both Z1 shall remain the same as Z2 axis.

Double Spindle wheels configuration

Under the configuration, the function of “Select Z1+Z2 Axes” is invalid.

When “Select Z1 Axis” or “Select Z2 Axis” selected, the current axis will be moved to machine reference point and the axis selected will not be moved. For example, the current axis is Z1 axis, and the selected axis is Z2 axis. Z1 will be turned back to machine reference origin firstly and then the Z2 axis will be activated.

In the wheel configuration, if it is encountered with T instrument, the machine tool shall have the following actions (e.g.: switch from T1 to T2):

- 1) stop Z1spindle
- 2) raise Z1 to machine reference origin
- 3) Close Z1enable function
- 4) Select Z2, Z2 to valid enable function
- 5) Switch Z2 to the workpiece coordinate position which is the very coordinate before Z1 cutter replaced.

Caution:

The resetting function of current workpiece coordinate is able to be carried out only under single axis mode regardless with the wheel motion or linkage mode configuration.

5.5.6. Efficiency - Quality Adjustment

Click the option to popup the following dialog box:

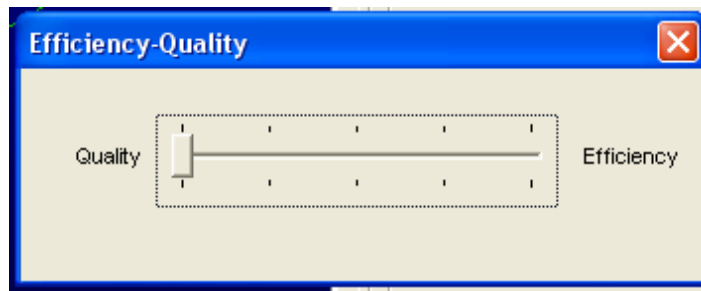


Fig. 5-30 Efficiency – Quality Adjustment Button

The quality is related to efficiency: when the quality takes the priority, the quality of processing is higher; when speed takes the priority, the efficiency of processing is higher.

5.6. "Window" Menu

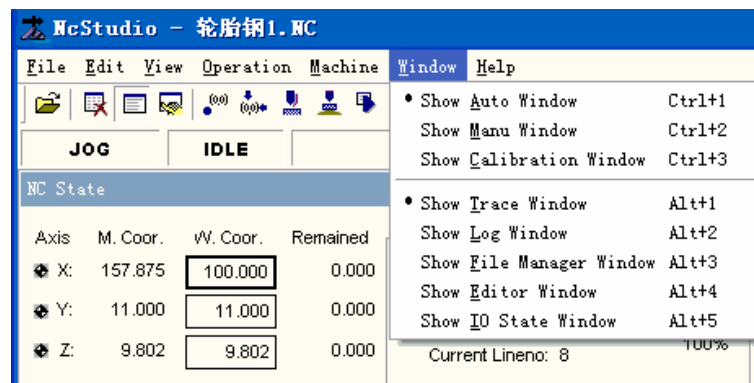


Fig. 5-31 "Window" Menu

This menu is used to interchange among the windows.

5.7. "Help" Menu

Select "Help" menu to popup the following menu:

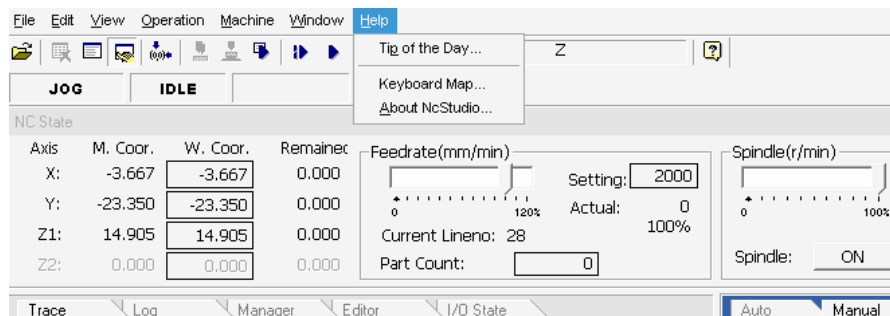
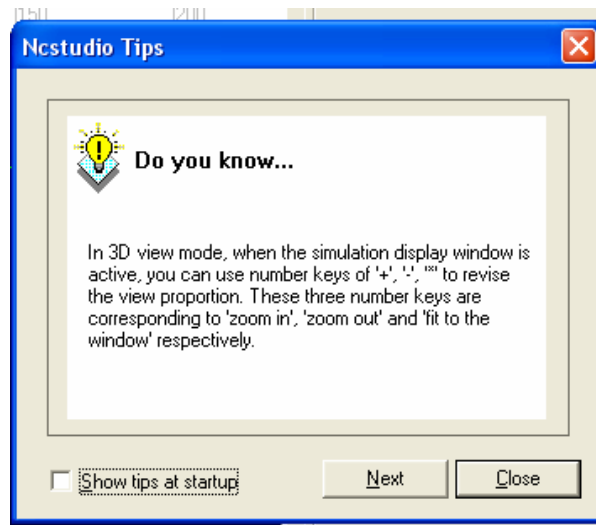


Fig. 5-32 "Help" Menu

5.7.1 "Daily Tip"

When user selects this option, a dialogue will pop out and tell some related information and

operating methods about NcStudio™. For example:



5.7.2 Description of Shortcut Key



Fig. 5-33 Shortcut Key List Window

5.7.3 About Ncstudio

This item is used to prompt user the information about system software and hardware. When this item is selected, a dialog will pop up and tell user some information such as the Version No., Type of the Motion Control Card, and Register Information of the NcStudio.



Fig. 5-34

5.7.4 Visit NcStuidio

User can use this item to visit NcStudio homepage of our website and to know our new products and other relative information.

6. Parameter Configuration

There are abundant processing parameters in Ncstudio, which can make the system more competent for each processing mission. This chapter will introduce operation authorization parameter only. For the authorization parameter for manufacturer which is used for inside communication for developers, please refer to Manufacturer's Manual.

Ncstudio parameters consist of operation parameter, feed axis parameter, spindle parameter, handwheel parameter, compensation parameter, origin parameter, tool parameter, other parameter and parameter overview.

6.1. Right of Parameter Modification

The parameter shown varies with different authorization parameter. There are passwords for manufacturer parameter and developer parameter.



Fig. 6-1

6.2. Parameter Modification Method

The method of the parameter modification is to select the parameters which need to modify by arrow key on the keyboard. Input the value in parameter inputting area after pressed the "Enter" key or double clicked the line where the parameters are.

For the parameter whose value is "true" or "false", inputting "1" means "true", inputting "0" means "false", user also can input "true" or "false" to modify the parameter.

Prompt:

All parameters can't be modified during processing.

6.3. Operator Authorization Parameters

6.3.1. Operation Parameter

【N4005】 SpindleActionsWhenFinished

Type	Int
Unit	None
Range	0: do not move; 1: Back to fixed point; 2: Back to workpiece origin.
Default	0
Valid time	Be valid immediately. Need not to restart.

Remark:

0: do not move. It means that after the normal completion of processing procedure, the spindle axis will stop at the coordinate value when processing finished.

1: Back to fixed point. It means that after the normal completion of processing procedure, the machine will return back to the fixed mechanical coordinate value set by “N4210”, “N4211”, and “N4212”.

2: Back to workpiece origin. It means that after the normal completion of processing procedure, the machine will return back to the workpiece coordinate origin of current procedure.

【N4006】 G73_G83CutterRetractRate

Type	Float
Unit	Mm. (millimeter)
Range	-99999~99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Remark: It means the retract rate after each feed operation by G73_G83 “high speed” reciprocating chip convey drilling for deep hole.

【N4007】 G76_G87 DirectionWhileFixedDrillStop

Type	Int
Unit	None
Range	0: +X , 1: -X,2: +Y , 3: -Y
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Remark: the above mentioned direction is only effective when it is under X-Y(G17).]

【N4025】ManualLowSpeed

Type	Float
Unit	mm/min(millimeter/minute)
Range	0~ manual high speed
Default	1000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The value means the motion speed of machine driven manually with 100% of feedrate (the manual speed varies along with the adjustment of feedrate).

【N4026】ManualHighSpeed

Type	Float
Unit	mm/min(millimeter/minute)
Range	Manual high speed ~Max speed supported by the hardware
Default	2400
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The setting value means the machine motion speed when feedrate is 100% and manual direction key and "high-speed" key ("0" key on small keyboard) are pressed down simultaneously.

【N4027】ToolFallingSpeedOnPauseAndContinue

Type	Float
------	-------

Unit	Mm/Min. (millimeter/minute)
Range	0~ Max speed supported by the hardware
Default	600
Valid time	Be valid immediately. Need not to restart the system.
Instruction	It means the tool falling (back to the cut point before pausing) speed of Z axis when it is paused and continued for processing.

【N4028】ToolRaisingSpeedOnPause

Type	Float
Unit	mm/min. (millimeter/minute)
Range	0~ Max speed supported by the hardware
Default	600
Valid time	Be valid immediately. Need not to restart the system.
Instruction	It means the tool raising speed on pause.

【N4029】JiggleFeedrate

Type	Float
Unit	mm/min. (millimeter/minute)
Range	0~ Max speed supported by the hardware
Default	60
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The speed when jiggling.

【N4030】JiggleStepLength

Type	Float
Unit	Mm. (millimeter)
Range	0.01~0.5
Default	0.01
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The jiggling length corresponding to the button knocked down.

【N4031】RapidTravelFeedrate

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0.001 ~ Max Speed
Default	3000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The motion speed of G00 order, namely the IDLE stroke speed

【N4032】DefaultFeedrate

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0.001 ~ RapidTravelFeedrate
Default	1500.000000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The interpolation speed of G01, G02 and G03 order, namely processing speed.

【N4034】 UseDefaultFeedrate

Type	Bool
Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

Select “false” for using the speed specified in processing file; Select “true” for using the default feedrate set with parameter “N4032”.

For some processing files (DXF, PLT format etc.) without specified processing speed, no matter you select “false” or “true”, the system will operate with the default feedrate speed set with parameter “N4032”.

【N4035】 UseDefaultSpindleRev

Type	Bool
Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid after restart the system

Remark:

Select “false” for using the spindle rev specified in processing file; Select “true” for using the default rev. set with parameter “N0002”.

For some processing files (DXF, PLT format etc.) without specified spindle rev., no matter you select “false” or “true”, the system will operate with the default rev. set with parameter “N0002”.

【N4044】 SpeedConfirmationMethodWhenZ-DownFeeds

Type	Int
Unit	None

Range	0: without special treatment; 1: the downward speed of Z cutter is effective when Z axis moves in negative position alone; 2: the downward speed of Z cutter is effective when Z axis moves in negative position; 3: Adjust feedrate slowly.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

0: do not have special treatment; there will be no Z direction tool falling speed set by "N4045", when Z axis is processing downward.

1: the downward speed of Z cutter is effective when Z axis moves in negative position alone; the system will adopt the Z down feedrate set by "N4045" when Z axis is processing downward alone.

2: the down forward speed of Z cutter is effective when Z axis moves in negative position; the system will adopt the Z down feedrate set by "N4045" for processing when Z axis (regardless the movement of X and Y) moves in negative direction.

3: Adjust feedrate gradually. Adjust the initial feedrate set by "N7018" to be default feedrate through the time set by "N7019".

【N4045】Z_DownFeedrate

Type	Float
Unit	Mm/Min. (millimeter/minute)
Range	0.001~9999
Default	500.000000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The parameter is valid when "N4044" parameter is selected to be "1" or "2".

【N4047】OptimizingZ-ToolRaisingSpeed

Type	Bool
-------------	------

Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	It is invalid to located instrument G00 regardless that the tool raising speed is optimized when Z axis moves upward vertically during processing.

【N4048】RaisingZAxisToolToSpecifiedWorkpieceCoordinate_ ValidOnPause.

Type	Bool
Unit	None
Range	0(false):Don't use. 1(true): Use.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

1. When parameter 【4048】 is selected with “false” for pausing, the parameter “ ‘N4050’ Z-ToolRaisingRateOnPause” is valid, meanwhile, the parameter “‘4049’Z-AxisPositionOnPause” is invalid.

2.When parameter 【4048】 is selected with “true” for pausing, the parameter “ ‘N4049’ Z-AxisPositionOnPause” is valid, meanwhile, the parameter “ ‘N4050’ Z-ToolRaisingRateOnPause” is invalid.

【N4049】 Z-AxisPositionOnPause

Type	Float
Unit	Mm (millimeter)
Range	0 ~500
Default	100
Valid time	Be valid immediately. Need not to restart the system.

【N4050】 Z-ToolRaisingRateOnPause

Type	Float
Unit	Mm (millimeter)
Range	0 ~500
Default	10
Valid time	Be valid immediately. Need not to restart the system.

【N4051】 SafetyHeight

Type	Float
Unit	Mm. (millimeter)
Range	0.001 ~ 1000.0
Default	10.000000
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The height is corresponding to workpiece coordinate origin, as the system regards it as safety height for horizontal movement. The parameter is used to execute the operation of "Back Workpiece Origin" and Breakpoint Resume.

【N4063】 ArcIJKIncrementModeValid

Type	Bool
Unit	None
Range	0(false): invalid 1(true): valid
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.

Remark:

1: when "true" is selected, the center coordinate is relative to the starting point for processing arc.

0: when "false" is selected, the center coordinate is relative to the coordinate of workpiece origin.

【N4068】ToolReplacingPromptValid

Type	Bool
Unit	None
Range	0(false): not prompt; 1(true):prompt
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	<p>When 0 “false” is selected, the system will not have a pause but continue to carry out processing when it is encountered with tool replacing instruction.</p> <p>When 1”true” is selected, the system will have a pause and prompt: there is a cutter required to be replaced, if you want to replace it, please stop the system and carry out breakpoint resume after the replacement; if you do not need to replace the cutter, please press “Start” to continue.</p>

Translation Parameter for PLT File**【N4070】ToolRaisingHeightWhenMovingWithIdleStroke**

Type	Float
Unit	mm. (millimeter)
Range	1~500
Default	1
Valid time	Be valid after the procedure file reload.

【N4071】PLTUnit

Type	Float
Unit	Mm/PLU. (millimeter/plu)
Range	40 or 1016
Default	40
Valid time	Be valid after the procedure file reload.

【N4072】 PLTToolsDistanceWhileProcessArea

Type	Float
Unit	mm. (millimeter)
Range	0.0001 ~ 99999.0
Default	0.025000
Valid time	Be valid after the procedure file reload.
Instruction	The distance of tools in PLT area when processing should be less than tool diameter.

【N4073】 TwoDimensionalFileDepth

Type	Float
Unit	mm. (millimeter)
Range	-99999 ~ 0.0
Default	-1.000000
Valid time	Be valid after the procedure file reload.
Instruction	Specify the processing depth of two dimensional file.

Translation Parameter for DXF Files**【N4080】 ToolRaisingHeightWhenMovingWithIdleStroke**

Type	Float
Unit	mm. (millimeter)
Range	1~500
Default	1
Valid time	Be valid after the procedure file reload.

【N4081】TwoDimensionalFileDepth

Type	Float
Unit	mm. (millimeter)
Range	-99999~0
Default	-1
Valid time	Be valid after the procedure file reload.
Instruction	Specify the processing depth of two dimensional file.

【N4082】ProcessingCapacityEveryTime

Type	Float
Unit	mm. (millimeter)
Range	-99999~0
Default	-1
Valid time	Be valid after the procedure file reload.
Instruction	It is processing depth of two dimensional file when carrying out layer processing.

【N4083】UseFirstPointAsOriginInDXFFiles

Type	Bool
Unit	None
Range	0(false):Don't use first point as origin. 1(true): Use the first point as origin.
Default	1(true)
Valid time	Be valid after the procedure file reload.
Instruction	Specify whether use the first point as origin in the DXF file.

Remark:

0: False Set the origin coordinate in DXF file to be workpiece origin.

1: True The system will set the customized point in DXF file to be workpiece origin. For example, when drawing with CAD, we can set a point (it will not be processed) at any part in the drawing (recommended to near the figure or in the figure), and the system will default the point to be workpiece origin. If there are several points contented in DXF file, the system will regard the first point to be workpiece origin.

【N4084】ShapeUniqueProcessingValid

Type	Bool
Unit	None
Range	0(false); 1(true)
Default	0(false)
Valid time	Be valid after the procedure file reload.
Instruction	Process a shape each time; treat another one until the former shape processing is completed.

ENG File Translation Parameter

【N4090】ToolRaisingHeightWhenMovingIdleStroke

Type	Float
Unit	mm. (millimeter)
Range	1~500
Default	1
Valid time	Be valid after the procedure file reload.

【N4091】PauseAndPromptWhileChangeTools

Type	Bool
Unit	None

Range	0(false):Don't pause and prompt while change tools. 1(true): Pause and prompt while change tools.
Default	1(true)
Valid time	Be valid after the procedure file reload.
Instruction	Specify whether pause and prompt while change tools.

【N4092】 ProcessingCycleTimesOfENGFile

Type	Int
Unit	None
Range	1~99999
Default	1
Valid time	Be valid after the procedure file reload.
Instruction	It is necessary to re-circulate the processing with several times when carrying out ENG file processing.

【N4093】 CutterChoosingForProcessingWithENGFile

Type	Bool
Unit	None
Range	0(false): not use cutter choosing for processing; 1(true):use cutter choosing for processing
Default	0(false)
Valid time	Be valid after the procedure file reload.
Instruction	Use the function you can carry out processing with chosen cutter and treat its corresponding processing files only.

【N4094】 PauseTimeAfterEachCirculationFinished

Type	Int
------	-----

Unit	None
Range	0~99999
Default	0
Valid time	Be valid after the procedure file reload.
Instruction	Specify the pause time after each circulation finished when it is processing the ENG file.

【N4095】DeepProcessingMethod

Type	Int
Unit	None
Range	0~1
Default	0
Valid time	Be valid after the procedure file reload.
Instruction	Deep processing method: 0, reciprocating chip breaking; 1, high speed reciprocating chip breaking.

【N4096】ToolRestractValue

Type	Float
Unit	Mm(millimeter)
Range	1~99999
Default	1
Valid time	Be valid after the procedure file reload.
Instruction	Tool retract rate after the tool is punched into a deep hole with high speed reciprocating chip breaking.

Fix Calibrator Position

【N4200】X 【N4201】Y 【N4202】Z

Type	Float
-------------	-------

Unit	Mm(millimeter)
Range	Work bench range lower limit(machine coordinate) ; Work bench range upper limit (machine coordinate)
Default	X: 0 Y: 0 Z: -1
Valid time	Be valid immediately. Need not to restart the system.

X: The value is set to be X axis machine coordinate value when the tool nose is reached into calibration range (it is better to get close to the center).

Y: The value is set to be Y axis machine coordinate value when the tool nose is reached into calibration range (it is better to get close to the center).

Z: The value is set to be Z axis machine coordinate value when the tool nose is located in a certain height range of calibrator (the tool is operated with idle stroke speed in the area above the height range, and it is operated with calibration speed in the area below the height range).

Machine Coordinate of Fixed Point

【N4210】X 【N4211】Y 【N4212】Z

Type	Float
Unit	Mm(millimeter)
Range	Work bench range lower limit(machine coordinate); Work bench range upper limit (machine coordinate)
Default	X: 0 Y: 0 Z: 0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The matching use of machine coordinate of fixed point and parameter 【N4005】 .

【N7018】 PercentInitialSpeedForLowSpeedFeedAtZAxisDirection

Type	Int
Unit	None

Range	1 ~ 100
Default	5
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The parameter is valid when 【N4044】 is set to “Adjust feedrate slowly”

【N7019】Z-LowFeedRateRecoversToSettingMagnificationTime

Type	Float
Unit	S (seconds)
Range	0~10
Default	5
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The parameter is valid when 【N4044】 is set to “Adjust feedrate slowly”, and in the period, the feedrate will recovers from 【N7018】 setting value to initial setting feedrate.

6.3.2. Feedrate Parameters

【N1002】ComfirmPriorToResettingWorlpieceCoordinate

Type	Bool
Unit	None
Range	0(false): Need not to have a confirmation 1(true): Need to have a confirmation
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Confirm that whether the resetting of workpiece coordinate value (set the current point to workpiece origin) should be prompted, if there is a prompt, it may avoid incorrect operation.

The check for workpiece coordinate range is valid.

【N1150】 X, 【N1151】 Y, 【N1152】 Z

Type	Bool
Unit	None
Range	0(false): invalid. 1(true): valid.
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	True is for check the current point for locating in workpiece coordinate extents, and false is for unchecking.

Workpiece Coordinate Lower Limit**【N1160】 X 【N1161】 Y 【N1162】 Z**

Type	Float
Unit	mm(millimeter)
Range	-99999~99999
Default	-10000
Valid time	Be valid immediately. Need not to restart the system.

Workpiece Coordinate Upper Limit**【N1170】 X 【N1171】 Y 【N1172】 Z**

Type	Float
Unit	mm (millimeter)
Range	-99999~99999
Default	10000
Valid time	Be valid immediately. Need not to restart the system.

6.3.3. Spindle Parameter**【N0004】 RevStopWhenStopped**

Type	Bool
Unit	None
Range	0(false):not stop; 1(true):stop
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether spindle is topped

【N0005】RevStopWhenPaused

Type	Bool
Unit	None
Range	0(false):not stop; 1(true):stop
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether spindle is topped

6.3.4. Origin Parameter**【N2001】NeedToBackToMachineReferencePointPriorToProcessing**

Type	Bool
Unit	None
Range	0(false): Need ; 1(true): Need not
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	When “True” is selected, the system will have a prompt for backing to machine reference point prior to carrying out processing; when “False” is selected, the system will enter into processing state without

	having any prompt.
--	--------------------

6.3.5. Cutter Parameter

【N6121】Diameter

Type	Float
Unit	Mm. (millimeter)
Range	0~99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The diameter of cutter selected.

【N6122】Length

Type	Float
Unit	Mm. (millimeter)
Range	0~99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The length of cutter sselected.

【N6123】DiameterWearExtent

Type	Float
Unit	mm (millimeter)
Range	0~99999

Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The diameter wear extents of cutter selected

【N6124】LengthWearExtent

Type	Float
Unit	mm (millimeter)
Range	0~99999
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	The length wear extent of cutter selected.

Cutter Position Offset

【N6130】X 【N6131】Y 【N6132】Z

Type	Float
Unit	mm (millimeter)
Range	Work bench range lower limit(machine coordinate); Work bench range upper limit (machine coordinate)
Default	X: 0 Y: 0 z: 0
Valid time	Be valid immediately. Need not to restart the system.

Remark: the three values of 【N6130】X, 【N6131】Y, 【N6132】Z mean the offset values of the cutter corresponding to the first cutter.

6.3.6. Other Parameter

【N7012】AdditionalCalibrationFunction

Type	Bool
-------------	------

Unit	None
Range	0(false): not use; 1(True):use
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Select "true" for using of additional calibration function; Select "false" for not using of additional calibration function.

【N7017】 InformTypesForCompletionOfProcessingTask

Type	Int
Unit	None
Range	0:Red light goes out; 1: Red light comes on for 3 seconds; 2: Red light always comes on until there are mouse or keyboard input by user and then the yellow light comes on.
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Remark:

0: Red light comes on

- a) When the software is under idle state, red light and green light shall go out, and the yellow light comes on;
- b) When the software is under normal processing state, the red light and yellow light shall go out and the green light comes on;
- c) After the completion of processing task, the red light and green light shall go out and yellow light comes on;

1: Red light comes on for 3 seconds

- a) When the software is under idle state, red light and green light shall go out, and the yellow light comes on;
- b) When the software is under normal processing state, the red light and yellow light shall

go out and the green light comes on;

c) After the completion of processing task, the green light and yellow light shall go out and the red light comes on for 3 seconds and then the yellow light comes on.

2: Red light always comes on until there are mouse or keyboard input by user and then yellow light comes on.

a) When the software is under idle state, red light and green light shall go out, and the yellow light comes on;

b) When the software is under normal processing state, the red light and yellow light shall go out and the green light comes on;

c) After the completion of processing task, the green light and yellow light shall go out and red light always comes on until there are mouse or keyboard input by user and then yellow light comes on.

【N7100】InterfaceSignalChangeforSystemStop

Type	Int
Unit	None
Range	0~FFFFFFFFFFFFFFFF
Default	0
Valid time	Be valid immediately. Need not to restart the system.
Instruction	After hexadecimal value is changed into binary system, the interface corresponding to the position 1 in binary system will have a signal change to stop the whole system.

【N7110】InterfaceSignalChangeForSystemPause

Type	Int
Unit	None
Range	0~FFFFFFFFFFFFFFFF
Default	0
Valid time	Be valid immediately. Need not to restart the system.

Instruction	After hexadecimal value is changed into binary system, the interface corresponding to the position 1 in binary system will have a signal change to pause the whole system.
--------------------	--

Double Z-Axis Parameter

【N7123】StartSpindleOrNotAfterCutterReplaced

Type	Bool
Unit	None
Range	0(false): Not start 1(true): Start
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether the current spindle should be started immediately after cutter replaced.

【N7124】SwitchToZ1AxisOrNotAfterProcessingCompleted

Type	Bool
Unit	None
Range	0(false):Not switch; 1(true): Switch
Default	0(false)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Specify whether the current spindle should be switched to Z1 Axis after processing completed.

【N7125】StopSpindleWhenCutterReplaced

Type	Bool
Unit	None
Range	0(false): Not stop 1(true): Stop

Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Select "false" for cutter replacement directly without stopping current spindle; Select "True" for cutter replacement after the current spindle stopped.

【N7126】 BackToOriginalPositionAfterCutterReplaced

Type	Bool
Unit	None
Range	0 (false): Not turn back; 1(true):Turn back.
Default	1(true)
Valid time	Be valid immediately. Need not to restart the system.
Instruction	Select "false" for not turning back to the coordinate before cutter replaced; Select "True" for turning back to the coordinate before cutter replaced.

7. Operation Instruction for Customized Small Keyboard

NcStudio™ version 5.4.53 and above support customized small keyboard.

For user's convenience of using the Ncstudio™ software, a customized small keyboard which arrayed in the manner of 5×4 in the layout with 20 keys totally was equipped with especially as shown in below:

Start Pause	Stop	Break point Resume	Jog
Spindle switch	Set to be origin	Back to origin	× 1
X+	Y+	Z+	× 10
X-	Y-	Z-	× 100
Shift	Calibrator	F-	F+

Fig. 7-1 Small Keyboard

Each key has specified function in the software, which listed as below:

- 1) Start/Pause: After pressed this key, the system begin to run the processing file if a NC file have been loaded and the system is idle; if the system is in running mode, pressing this key will make the system pause.
- 2) Stop: Stop the current operation. If the system is in the status of processing, pressed this key, processing will stop. But the mobile calibrator will be stopped after this key was pressed if the system is under mobile calibrator status.
- 3) Break Point Resume: The system will be resumed to machine after this key was pressed if the system is in the status of processing pause.
- 4) Jog: Switch the manual data input (MDI) mode to manual jog.
- 5) Spindle switch: Turn on the spindle after this key was pressed when the spindle was turned off. Otherwise, turn on the spindle.
- 6) Back to origin: The system will execute the order of "back to workpiece origin" when pressed.
- 7) × 1: After this key was pressed, the jog stepping length will be set as 0.01mm.
- 8) × 10: After this key was pressed, the jog stepping length will be set as 0.1mm.

- 9) $\times 100$: After this key was pressed, the jog stepping length will be set as 1mm.
- 10) X+: Move the cutter along the X axis' positive direction. If the machine tool is in manual continual jog state, the cutter will be moved to the X axis' positive direction continually with manual low speed by pressing the key; the cutter will be moved to the X axis positive direction with manual high speed by pressing "Shift" and "X+" simultaneously; if the machine tool is under the state of X1,X10 or X100, the cutter will be moved to X axis positive direction with corresponding distance by pressing X+ key.
- 11) X-: Move the cutter towards the X axis' negative direction. The speed, the stepping length and the combination keys are the same as X+ key.
- 12) Y+: Move the cutter towards the Y axis' positive direction. The speed, the stepping length and the combination keys are the same as X+ key.
- 13) Y-: Move the cutter towards the Y axis' negative direction. The speed, the stepping length and the combination keys are the same as X+ key.
- 14) Z+: Move the cutter towards the Z axis' positive direction. The speed, the stepping length and the combination keys are the same as X+ key.
- 15) Z-: Move the cutter towards the Z axis' negative direction. The speed, the stepping length and the combination keys are the same as X+ key.
- 16) Shift: Nothing will happen if press this key separately. It will be valid only when you press the key combined with "X+", "X-", "Y+", "Y-", "Z+", or "Z-".
- 17) Calibrator: The system will mobile calibrate after this key was pressed.
- 18) F-: The feed rate will be decreased if this key is pressed.
- 19) F+: The feed rate will be increased if this key is pressed.

8. Operation Steps

8.1. Power on

Before starting the machine tool, you should make sure all the connections of machine tool and computer are normal firstly. Then turn on the power of machine tool and the computer. After the system was loaded, enter the NcStudio™ digital control system.

8.2. Machine Reset (Optional)

Only the machine tool with the function of “Back to machine reference point” is involved in this section as required.

Choose “Back to Machine Reference Point” menu if machine tool support operation of it. Machine tool will back to Machine Reference Point automatically, and calibrate the coordinate system.

In some circumstance, user is not necessary to execute the operation of machine restoration when restart the system and continue previous operation if stopped normally while the current coordinate information have been saved when the Ncstudio system existed normally.

Additionally, if user confirm current position is correct, it is not necessary to execute such operation.

8.3. Loading the Processing Procedure

Before processing, the user need loading the processing procedure generally, otherwise, the corresponding automatic function is null.

When the menu of " File | open... " is selected, the system will flip out a dialog box of the standard document operation, which you can choose the driver, path, and the file name that you want to open. After clicked the button of “open”, the processing procedure will be loaded. Then, to press the key of F2 to switch to the window of “processing procedure”, the user can view the current processing procedure.

8.4. Manual Operation

Display the Interface of Manual Operation.

When the menu of "Window| showing Manu window " is selected, a manual operation interface will be displayed in the parameter list window, which can have manual operation to the machine tool.

Manual Moving

User can have manual moving to the machine tool by the corresponding key in the small keyboard while the NUMLOCK light is bright.

The corresponding key is:

4	negative direction of X-axis
6	positive direction of X-axis
8	positive direction of Y-axis
2	negative direction of Y-axis
9	positive direction of Z-axis
1	negative direction of Z-axis

The machine tool will be moved manually with high speed by pressing down any of these keys together with 0 key located on small keyboard.

8.5. Set Workpiece Origin

When processing, the origins of the X, Y, and Z axes are the workpiece origin. Therefore, we should set the workpiece origin as below prior to processing:

Moving the X, Y axes to position of expected origin manually, choosing the menu of "Set current point as workpiece origin" or clearing the coordinate value of current position as zero, the current position will be the start point to process when execute processing file.

Workpiece origin setting of X, Y axes was completed after the steps listed above. But more exact operation are needed to set workpiece origin of Z axis, the system cooperating with workpiece hardware provide the function of Z axis calibrator.


Select the menu of "operate |mobile calibrator..." which help you to complete mobile calibrator.

After these operations, the workpiece origin is confirmed.

8.6. Execute Auto Machining


Auto machining means that system will carry out the machining program automatically.

Start Auto Machining

Select the menu of "Operation (O) |Start(S)" or click the button  on the toolbar, the machine tool will execute the processing automatically from the first sentence of the procedure. Hotkey: F9.

Machine Tool Stop


During auto machining, if you want to stop the processing procedure, the machine tool will stop processing immediately and enter into "idle" state through selecting the menu

"Operate| Stop", clicking the button  on toolbar or use hotkey "F11", this is the recommended method for stopping the system accurately and orderly.

Note:

When the connection characteristic of super-speed smooth speed is valid, system will stop while connection speed is zero.

Machine Tool Pause

During automatically processing, if it is necessary to pause the processing, select the menu of "operate | pause" or corresponding button  in the toolbar, the machine tool will stop after finished the current sentence of processing, which also can be performed by hotkey of "F10" .If you want keep on processing, you can choose the menu of "operate| Start".

Program Jumping Execution

Selecting the menu of "Advanced Start (A)", the system will flip out a dialog box for you to select the starting row and end row. For detailed operation, please refer to "Advanced Start" in 5.4-Operation Menu.

9. Precautions in Operation

9.1. Precautions for Multi-Task Executing

Because the Windows is time-sharing operating system, general speaking, when runs the auto processing, you also can run other tasks (for example, edit the processing procedure), but please pay attention to the following two items:

For the procedure of Windows taking a lot of memory, please do not open too much windows, which confirmed by the size of computer memory.

The movement of some applications themselves may be not stable, for example, some game procedure, VCD player etc. They may take the system resources without limit in running, such as the memory, CPU time-piece etc, which can cause the computer crack down. So, in processing time, please do not start these procedures, so as not to result in accidental processing interruption.

9.2. Precautions for Backing to Machine Reference Point

During the process of backing to machine reference point, according to difference requirement of system, difference procedure would be occurred. In the system which requires high precision, ultimately, the procedure of the correcting would be slow. Please pay attention to the statue window, to back to the machine reference point after the system in status of idle, otherwise, backing to machine reference point would not complete normally but terminated manually.

The consequences of manual termination during the process of backing to machine reference point are listed as below. 1. Because the software limit signal did not disappear, port alarm may occur; 2. Inexact allocation (the function of calibrating machine reference point is artificially destroyed, so it may lead to nonstandard machine coordinate); 3. Software limit doesn't function (Because the process of backing to reference point did not complete, system considers the software limit is invalid; only when backing to the reference point is finished, software limit can effect).

10. Software Permission Agreement

Permission:

Shanghai WeiHong Technology Co. Ltd. (hereafter referred to as WeiHong Company) awards you the software right of use. But you must guarantee toward to the WeiHong Company: Does not use, copy, revise, rent, transfer either this system or any part of the software for other purpose beyond the articles stipulated in this agreement .

You must guarantee that:

1. Only be used this system on one machine;
2. Only be used on this machine. In order to backup or manage the archives, permit to copy this system by the read-only mode.
3. Only when the third party accepts the provision and condition of this agreement, permit to authorize to transfer this system and permission agreement to the third party.
4. When transferring, the original documents, supplementary documents and all copies must be transmitted to the other party together; or destroyed the surplus products completely;
5. Only under the circumstance of accepting one provision of items below, permit to use this system on the multiuser environment or the networks:

This agreement definite provision permitted to use in the multiuser environment or the network system;

Usage of each pitch point and terminal of the system have all purchased the use permission.

You must guarantee not do those below:

1. Transfers the permission any more;
2. Reverse project, disassemble or disintegrate this system;
3. Copy or transmit this system completely or partially except the regulation listed in this permission.

When you hand over the system or the copy to another completely or partially, your permission power terminated automatically.

Copyright and Property Rights:

This system and the documents enjoy the copyright, and are protected by the international agreement provision. You can not remove the copyright from the copyright statement of the software. You guarantee to copy duplicates of the copyright statement for this system (completely or are partial). You agreed to stop copying the system and documents by any illegal forms.

After-sale Guarantee:

The WeiHong company guarantees, in the normal use situation, from sells the date in 90 days, the software carrier has the material or the craft flaw, when really has the flaw after confirmation, the WeiHong company's complete responsibility is to exchanges a software carrier; also it is the only compensates. When the accidents, abuses or error usage resulted in paralysis of software, the after-sale guarantee is invalid. The carrier which exchanges from WeiHong company enjoy surplus time of original guarantee time, or 30 day-long guarantee; Takes its elder first.

Except the after-sale guarantee above, this system does not enjoy the after-sale guarantee of any other forms.

Responsibility limited:

The assurance declared above, regardless of listed or hided, is complete content of guarantee, including commercial character and compatible guarantee for special application goal. No matter either follows this agreement and other provisions or not, produced by using this system: The loss of profit, the usability lose, the commercial severance, or indirect, special, accidental and inevitable destruction of any forms, or the other party's any suing for damages, the WeiHong company, its agent and its seller are not responsible generally, even if the WeiHong company is informed the possibility occurrence of this kind of matter.

Permission termination:

If you disobey no matter what a provision and the condition of this agreement, the WeiHong company may terminate this permission. When the permission was terminated, you must destroy this system, documents and all copies immediately, or return them to WeiHong company.

Applicable Law:

Intellectual property rights Protection Rule, Copyright Law, Patent Law and so on.

Now, we affirm you already have read and understood this permission in detail, and agree to obey all of the provisions and conditions of the agreement strictly.

Shanghai WeiHong Electronic Technology Co., Ltd.

11. Appendix: Shortcut Keys List

11.1. General Shortcut Key

ESC	Switch between windows
TAB	Switch between control buttons
Ctrl+1	Display Auto Window
Ctrl+2 /Scroll Lock	Display Manual Window
Ctrl+F1	Switch to Z1
Ctrl+F2	Switch to Z2
Ctrl+F7	Floating Calibration
Ctrl+F9	Advanced Start
Ctrl+ Home	Back to Machine Reference Point
Ctrl+ Enter	Full Screen
Ctrl+ TAB	Switch between Collapsing Windows
Ctrl+ Del	Clear Machining Track
Ctrl+Shift+F9	Execute Machining Instruction
Ctrl+ A	Select All
Ctrl+ C	Copy
Ctrl+ E	Open and Edit
Ctrl+ F	Find
Ctrl+ H	Replace
Ctrl+ I	Information of machining Procedure
Ctrl+ N	New Procedure

Ctrl+ O	Open and Load
Ctrl+ P	Edit Current Processing Program
Ctrl+ S	Save
Ctrl+ U	Unload
Ctrl+ V	Paste
Ctrl+ X	Cut
Ctrl+ Z	Undo
ALT+1/F4	Display Processing Track Window
ALT+2	Display System Log Window
ALT+3	Display Manager Window
ALT+4	Display Editor Window
ALT+5	Show I/O State Window
F3	Find Next
F5	Direct Setting
F6	Set the Current Workpiece Coordinates
F7	Back to Workpiece Origin
F8	Enter (Exit) Simulaton Mode
F9	Start
F10/Pause Break	Pause
F11	Stop
F12	Back to Position of Tool Change
Shift+F6	Set Current Point as Workpiece Origin
Shift+F7	Fixed Calibration
Shift+F9	Breakpoint Resume

11.2.Shortcut Key of Manual Window

Scroll Lock	Activate Manual Window
4(Small keyboard)	Manual Direction of X- (Including Jog and increment)
6(Small keyboard)	Manual Direction of X +(Including Jog and increment)
2(Small keyboard)	Manual Direction of Y- (Including Jog and increment)
8(Small keyboard)	Manual Direction of Y+ (Including Jog and increment)
1(Small keyboard)	Manual Direction of Z- (Including Jog and increment)
9(Small keyboard)	Manual Direction of Z+ (Including Jog and increment)

11.3.Shortcut Key in Track Window

Home	Center
End	Show Current Machining Point
+ (Small Keyboard)	Zoom In
- (Small Keyboard)	Zoom Out
* (Small Keyboard)	Adjust to Window Size
5 (Small Keyboard)	Front View
8 (Small Keyboard)	Top View
2 (Small Keyboard)	Bottom View
4 (Small Keyboard)	Left View
6 (Small Keyboard)	Right View
1 (Small Keyboard)	Southwest View

7 (Small Keyboard)	Northwest View
3 (Small Keyboard)	Southeast View
9 (Small Keyboard)	Northeast View
Alt+→ or Alt+←	Rotate round Z-axis
Alt+ ↑ or Alt+ ↓	Rotate round X-axis
Alt+PgUp or Alt+PgDn	Rotate round Y-axis