

## **Preface**

Thank you for choosing our products.

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

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

Because of continuous updating, the products you bought may differ from the written in this manual.

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

Company Name: Weihong Electronic Technology Co., Ltd.

Headquarters Address: No.1590, Huhang Rd., Fengxian, Shanghai, PRC

Zip Code: 201400

Tel: +86-21-33587550

Fax: +86-21-33587519

Website: http://en.weihong.com.cn



### Warning:

- 1) Don't plug or pull out the cable connected with the integrated machine when power on.
- 2) Please don't plug or pull out the cable of manipulation boxes when power on.
- 3) For safety and disturbance reduction, the shell of computer and integrated machine must be grounded.
- 4) If machine is not in using, please shut off the power.
- 5) Please pull out the attaching plug 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) It is forbidden to touch the engraver to avoid injury caused by its sharp edge. And don't contact it with handkerchiefs and scarves either to prevent being caught in the machine or damaging the equipment.



### **Fast Master Chapter**

This chapter is applied to the condition that all electric wiring has been finished and the NcStudio system and motion card have been well installed.

Check if the conjunction is correct and test whether communication signals are normal.

Join the terminal board to port JP3 of the card with DB37M/F cable, add 24V power supply to the terminal board and power supply to the machine signal system (proximity switch, etc). Examine signal input indicator LED on the terminal board: if the origin switch is normally closed, three LEDs of X0、Y0、Z0 should be alight. Then users can touch the origin switch through artificial imitation. (For travel switch, artificial pressure can be used to observe whether signals can be received. For photoelectrical switch, manually obstruct the light to see if the signals can be gotten. For metal proximity switch, artificially touch it with metal to see if the signals can be gotten.) If the corresponding LED is out when being touched, it indicates the origin signals have been sent to the terminal board. If the origin switch connected is normally opened, LED should be normally out, and by artificially touching the switch, LED will become light, which shows the origin signals have been received by terminal board. The same method can be taken to test other ports to make sure the correctness of the wiring between the terminal board and machine tool. This method can greatly shorten the debugging time.

### Debug Signals

Open the computer and run NcStudio software. Open [DIAG]→[I/OPort] window, users will see many mapping InPorts and OutPorts. Green lamps indicate the signals are valid, while red lamps indicate the signals are invalid. Revise the polarity of the mapping InPorts according to the type of origin switch, E-STOP buttons, and other buttons you have chosen. Polarity of the normally open InPort is "N", and that of the normally closed InPort is "P". After all modification finished, exit from NcStudio and restart it. The modification will become effective. Supply power to the electrical box. Dots in [I/O Port] window, in front of signals of proceeding start, proceeding stop, tool calibration signal, zero points of the three axes and E-STOP InPort should be in red, which indicates all these signals are invalid. If the dots show in green, users should examine the circuitry in electrical box and the polarity of the signals. If there is no problem with the circuitry, users should revise the corresponding polarity to make sure the dots in front of the above signals in red.

Load processing program after checking. Press [Program Start] button under operation panel, and observe if the color of the dot in front of [Program Start] changes. It should be in green when being pressed and in red when being released. If no change occurs to this dot, please turn to step one to check if LED on terminal board functions normally, check if wiring conjunction of DB37M/F and DB50M/F cable with motion control card and terminal board becomes loose. Imitate to touch the



origin signals following the first step. Check the reference point signals of X-axis, Y-axis and Z-axis. Once the origin signal is being touched, the signal dot should be in green.

#### Parameters Settings

Set parameter [pulse equivalent] of manufacturer parameters. The smaller the pulse equivalent is, the higher the resolution will be. But if the pulse equivalent is too small, it will affect the maximum feed rate. The default pulse equivalent is 0.001 mm/P (The corresponding maximum feed rate is 9600 mm/min); after the pulse equivalent has been confirmed, please compute the electronic gear ratio of servo driver in accordance with value of this pulse equivalent. Electronic gear ratio= encoder resolution × pulse equivalent × mechanical deceleration ratio/ pitch; thereinto, mechanical deceleration ratio= reducer rotation speed input / rotation speed output= driven gear tooth number / driving gear tooth number. Move the machine manually to make sure the correctness of moving direction of each axis and note that NcStudio is abiding by right-hand coordinates system.

That is to say, rightward movement of X-axis is the positive direction, and upward movement of Z-axis is the positive direction, while the positive direction of Y-axis is moving far away from the operator. (If Y-axis movement is the worktable movement, the positive direction is the direction of worktable moving towards the operator). If this direction is not right, you can revise [Axis Direction] parameter or relative servo driver parameter of manufacturer parameters. Set [worktable stroke] under manufacturer parameters on the basis of the actual machine tool dimension to enable the software limit. Set [mechanical reference point] of manufacturer parameters on the basis of sensor installation position of the three axes, execute [back to the reference point] action, and carefully observe the moving direction of the axes when backing to the reference point. If moving direction is abnormal, users need to reset the parameters of [back to mechanical reference point]. Set inverter parameters to make the inverter work under 0~10V analog voltage control mode. Spindle ON/OFF adopts forward rotation terminal control mode. Press [Start] button in the software, observe in hardware port window whether the color of signal dot in front of spindle turns green and check on the terminal board whether the corresponding green signal lamp LED of relay become light. At this time the spindle should start to rotate. If not, please examine the conjunction of inverter. Adjusting the spindle speed in the software will change the rotation speed of spindle noticeably; otherwise, examine the conjunction and the parameters setting of inverter. If the spindle rotates in a wrong direction, please reconnect spindle and inverter and exchange any two of three spindle wires (in usual case) to alter the rotary direction of spindle.

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



## **Contents**

1.	SUMMARIZATION		1
	1.1.	Software Characteristics	1
	1.2.	UPDATE RECORDS OF SOFTWARE	3
2.	SYST	FEM INSTALLATION & CONNECTION	5
	2.1.	Basic Configuration of NcStudio.	_
	2.1.	Installation of NcStudio.	
	2.2. 2.2.		
	2.2.	•	
	2.2.	•	
	2.3.	Uninstall of NcStudio System	
-	2.4.	CONNECTION OF NCSTUDIO AND DRIVER SYSTEM	
3.	THE	BASIC CONCEPTS OF NCSTUDIO	
;	3.1.	OPERATION MODE AND STATE	11
	3.1		
	3.1	•	
3	3.2.	COORDINATE SYSTEM	
	3.2.	1. Machine Coordinate System (MCS)	13
	3.2	2. Workpiece Coordinate System (WCS)	13
4.	OPF	RATION INTERFACE OF NCSTUDIO	15
	4.1.	OVERALL INTERFACE	
	4.2.	TITLE BAR	
	4.3.	MENU BAR  NC (NUMERICAL CONTROL) STATE BAR	
	4.4. 4.5.	INFORMATION PROMPT BAR	
	4.5. 4.6.	FUNCTION WINDOW	
	4.0. 4.7.	Manipulation Button Bar	
	4.7. 4.8.	OPERATION PANEL	
5.		O MODE OF NCSTUDIO	
	5.1.	OPERATION PANEL SECTION	
	5.1. 5.2.	FUNCTION PANEL SECTION	
•	5.2. 5.2.		
	5.2		
	5.2		
	5.2.		
	5.2.	,	
	5.2.		
6	NAAI	NUAL MODE OF NOSTLIDIO	65



6.1.	OPERATION PANEL SECTION	65
6.2.	FUNCTION PANEL SECTION	68
7. RE	FERENCE POINT MODE OF NCSTUDIO	75
7.1.	OPERATION PANEL SECTION	
7.2.	FUNCTION PANEL SECTION	77
8. MI	ENU SYSTEM OF NCSTUDIO	79
8.1.	FILE MENU	79
8.2.	Work Mode Menu	83
8.3.	OPERATE MENU	85
8.4.	Machine Tool Menu	90
8.5.	WINDOW MENU	91
8.6.	HELP MENU	92
9. PA	ARAMETERS OF NCSTUDIO	93
9.1.	METHODS OF PARAMETERS MODIFICATION	93
9.2.	Manufacturer (Including Operator) Parameter List	93
10.	NCSTUDIO APPENDIX	111
10.1.	. Appendix of Shortcut Keys	111
11.	SOFTWARE LICENSE AGREEMENT	113



## 1. Summarization

NcStudio V9, CNC system for engraving machines, is independently designed by Weihong Electronic Technology Co., Ltd., with copyright reserved. This system can directly support G code, DXF format, PLT code format and ENG code of JDPaint processing file that are created by CAD/CAM applications such as UG, MasterCAM, CASMate, ArtCAM, AutoCAD, CorelDraw, and so on.

PC- based NcStudio takes great advantage of 32-bit computing and multi-tasks. At the same time, its interface is user-friendly and dependable.

Apart from functions of automatic mode, manual mode, and reference point mode, this CNC system also features the following functions: simulation, dynamic track, auto Z axis, breakpoint resume (advanced start), rotary axis machining, and so on.

## 1.1. Software Characteristics

NcStudio system includes the following functions:

- 1) Supporting numerical rotary table (varying in different editions);
- 2) Automatic machining, completely supporting G code of ISO standard, HP plotter (HP PLT format) and JDPaint (ENG format);
- 3) Supporting manual function. You are allowed to manipulate the machine not only with the help of machine tool input equipment like hand-held device, but also by using computer input equipment such as keyboard and mouse to realize manual operation;
- 4) Supporting array processing, repeatedly performing the processing program according to rows and columns of array that you have selected;
- 5) Supporting incremental feed to facilitate setting the amount of feed with specific and customized step length;
- 6) Supporting manual data input function. You can directly input G code and immediately execute it;
- Supporting processing wizard. Inputting several parameters will suffice to complete functions of milling frame and milling bottom;
- 8) Supporting single block mode. You can set the program selected to be executed as [Single Block], which is a good way to support error diagnosis and failure recovery action;
- 9) Supporting advanced auto functions including breakpoint resume and advanced start and so on;



- 10) Supporting function of "save/restore workpiece origin";
- 11) Supporting accurate homing of feed axes;
- 12) Supporting auto tool measurement to bring great convenience for users;
- 13) Supporting feed ratio on-line adjustment. During machining, you can adjust the feed ratio at any time. The least value is "0", indicating the processing is paused, and the maximum ratio is 120%;
- 14) Supporting new function of high-smooth speed connection algorithm. In a usual numerical control system, connection speed between two G codes commonly is a fixed value (e.g., it may be "0" or a certain small fixed value). However, in new version of CNC system, it adopts a particular adaptive prediction algorithm of processing speed. In terms of speed value, direction and maximum acceleration, with the function of forward prediction, the algorithm adaptively decides the connection speed between the current instruction and the next one. It has greatly increased the processing efficiency (from 30% to 300%) and improved the processing performance by eliminating the speed chatter marks left on the processing surface;
- 15) Supporting particular function of three-dimensional simulation display. Through simple operation, you can observe the processing results from different angles to understand the processing result more accurately and more intuitionally;
- 16) Supporting new function of loading machining track in advance. After loading the machining file, the system will automatically analyze the tracks, which you can directly call out the tracks to view;
- 17) Supporting simulation function. This application can rapidly simulate the machining process for program in a short time. In the meantime, you can check the machining file and result and learn the actual processing time;
- 18) Supporting keyboard operation. A powerful keyboard operation in the new version is provided to meet users' need;
- Supporting new Log function. The powerful log function can help you check the detailed processing information and system diagnosis;
- 20) Supporting built-in processing file manager. You just need to save the processing file to a specified directory; and then NcStudio can administrate it in a built-in file manager;
- 21) Supporting built-in processing file editor. At any time you can introduce a processing file into file editor for editing and modifying;



- 22) Supporting display of the file processing information. By simulation or actual processing, [file processing information] window can help you count such important information as processing time, processing range, the amount of finished workpiece, etc;
- 23) Supporting particular [Parameters Auto backup] function. The system can automatically save the parameters setting information which can save you a lot of time and trouble to set parameters repeatedly;
- 24) Supportting backupof 8 parameter archives in [Parameters Back Up] window. You can save different parameter settings of different processing files, and load them at any time;
- 25) [I/O Port] window can do map simulation, test and regulate polarity, which helps you well realize hardware detection, processing supervision, and fault diagnosis;
- 26) Newly-added PLC module, through which you can control I/O ports;
- 27) Supporting intelligent look-ahead; the system will decide interpolation strategy by looking ahead some distance in order to improve the integral smoothness of workpiece;
- 28) Supporting MPG (Manual Pulse Generator), you can determine whether to use MPG or not according to your needs;
- 29) Supporting multi-language. Currently, English and Chinese have been embedded in the software, and other languages will be added in demands;
- 30) Suitable parameters setting to improve machining efficiency with high processing accuracy.

## 1.2. Update Records of Software

In order to provide convenience for you to learn the latest information of our software rapidly, the update records of each software version are listed here as following. Therefore you can compare the records with the software version number that you have received, and upgrade the software promptly. If you want customization service, please feel free to contact us.

- Strengthening simulation ability and stability: [Back to the Reference Point] and [Save the Reference Point] can be tested during simulation process; [Stroke Range] and [Position Limit] are added in [Simulation] window;
- [System maintenance]: you can maintain the system and make installation package for the system, which is stable with entire parameters;



- 3) Further enhanced function: icons in [I/O Port] interface vary with different states with more user-friendly interface;
- 4) Under [Auto] mode → [Position] function window → [Information] screen, add statistics function
  of machining time, cutting time, and workpiece count;
- 5) New hardware time encryption to protect the interests of customers;
- 6) Add encoder origin signal, servo driver alarm signal and servo driver enabled signal;
- 7) Built-in optical positioning: it can tune the theoretical tool path to coincide with actual workpiece installation position and actual array position, so as to process a workpiece with certain accuracy qualification according to pre-set path.



# 2. System Installation & Connection

It is very simple to install NcStudio application. Advanced or regular customers can quickly refer to fast master chapter.

## 2.1. Basic Configuration of NcStudio.

#### Host Computer

CPU: Main frequency 1G or above

Memory: 512 M or above.

Hard disk: 20G or above

Display adapter: 1024\*768 at least,

Display: 14" VGA or above

CD-ROM: 4X or higher (optional)

Main board expansion: More than two slots of PCI

### 2.2. Installation of NcStudio

If there is already an old version of NcStudio, please delete it, and then install the new version. Regarding how to delete the old version, please turn to chapter 2.3 "uninstall of NcStudio system".

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

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

## 2.2.1. Installation of NcStudio Software

Software installation can be divided into several steps as following:

1) After the computer is booted up, the system will automatically enter into the operating system.



2) Put the setup CD of NcStudio system into the CD-ROM. Double click the "My computer" icon,

then double click the CD-ROM icon. Find out the SETUP.EXE file and double click its icon
The first dialogue box that appears is the [Language of the theme] dialogue, see Fig. 2-1. In V9
version, you can switch over between English and Chinese in the system. If you click [Chinese]
and run NcStudio system after installation, the system will take the Chinese interface as the
default interface. The same goes for [English]. Of course you can also perform this function in
[System] window. If there is already NcStudio on the computer, the new package will delete the
old version and install the new version to update the software. At this time, an update prompt will
appear on the screen, as Fig. 2-2.



Fig. 2-1 Installation language selection dialogue



Fig. 2-2 Update prompt dialogue box

3) Click [Yes (Y)]. The prompt will ask you to save parameters, then old version will be deleted and installation will go on. Please see Fig. 2-3 below and be careful to save the parameters before installation.



Fig. 2-3 Saving parameters before installation

4) Click [Yes (Y)]. Installation begins. NcStudio will be installed to the default directory, C:\Program Files\Naiky. Installation progress bar will display the installation speed. See Fig. 2-4. After installation, prompt will remind you whether to put generated files and executive files separately



and whether to shut down computer (to facilitate installation of the control card). The recommended operation is [Yes], see Fig. 2-5 and Fig. 2-6.

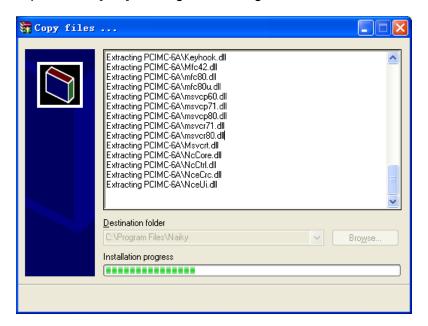


Fig. 2-4 Software installation interface

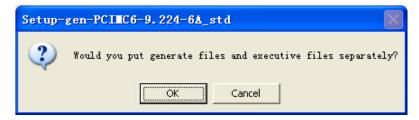


Fig. 2-5 Whether to put generate files and executive files separately

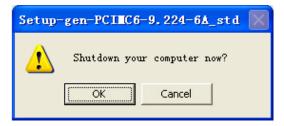


Fig. 2-6 Whether to close the computer at once (suggested operation: Yes)

5) Here the software installation has been finished.

### 2.2.2. Installation of Control Card & Other Devices

Installing NcStudio motion control card is divided into the following steps:

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



#### Note:

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

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

After the installation of motion control card and other external equipment have been finished, please restart the computer. The operating system will report a new hardware has been found, and eject a hardware update guide dialogue. See Fig. 2-7. You should choose [Install from a list or specific location] and click [Next]. See Fig. 2-8. Then the operation system will install the driver application for NcStudio software automatically. Up to this point, the whole installation of NcStudio software and the motion control card has been completed. To start NcStudio software, click [Start] menu on the desktop  $\rightarrow$ click [Applications]  $\rightarrow$ click [NcStudio]. You can also start it by the relative shortcuts.

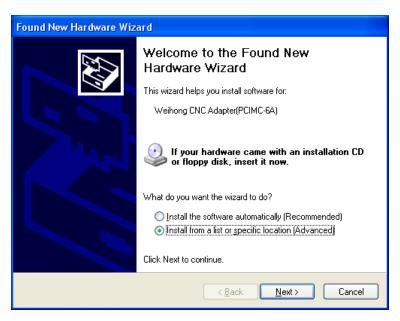


Fig. 2-7 Page one of hardware update guide



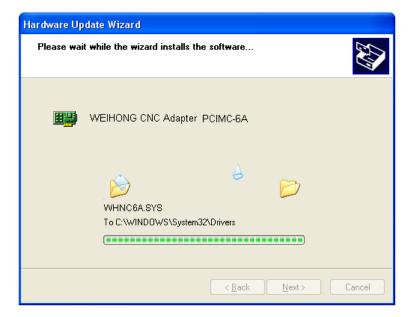


Fig. 2-8 Page two of hardware update guide

Note:

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

### 2.2.3. Other Installation Problems

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

## 2.3. Uninstall of NcStudio System

NcStudio is green software featuring the following advantages: easy to install and uninstall with installation info not written into registry, portable and available of copying to a USB. To uninstall NcStudio system, you only need to delete its document folder under "C:\Program Files\Naiky".

When new version of NcStudio is installed, the new software will update and overwrite the files of the old version automatically.

## 2.4. Connection of NcStudio and Driver System

Mechanical motion control signals of NcStudio realize the communication between the NcStudio system and driving system of feed motor mounted on the electrical box of a machine tool via the



control card inserted in the PC expansion slot.

Install the machine tool and electrical box in place, and then connect the socket of the motion control card with the corresponding socket on the electrical box by the exclusive cable before connecting the motion control card of NcStudio with the driving system of motor. In this way the connection has been finished.

#### Note:

Please refer to the card manual of the concrete type for wiring diagram between card and electrical system.

The working method may be different with different types, so are the functions.



## 3. The Basic Concepts of NcStudio

NcStudio is a comparably complicated system concerned with many concepts, such as Workpiece Coordinates System (WCS), Machine Coordinates System (MCS), operation mode and operation state, etc. Since these concepts are indispensible in use, so you should figure them out before using the system.

## 3.1. Operation Mode and State

## 3.1.1. Operation Mode

It is necessary for you to understand operation mode correctly. Regarding machining operation, there are totally three modes as below.

#### ◆ Reference Point Mode (Back to the Reference Point)

[Back to the Reference Point] mode is the process to synchronize the internal coordinates with the machine actual coordinates. Therefore, in the process of machine tool start-up, "back to the reference point" is essential. Every time the system is started, the default operation mode is reference point mode. It is very convenient for you to implement [Back to the Reference Point] action.

#### Auto Mode

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

#### Manual mode

Under manual mode, you can control the machine tool via manual operation equipment, such as manual pulse generator and manual operation panel. Manual mode can be subdivided into the following modes: jog, stepping (also-called increment), handwheel, and so on.

- Under [Jog] mode, you can control the machine tool by manual operation devices, such as keyboard, manual pulse generator and hand-held box, etc. when you send motion signals by
  - these devices and keep pressing the button for example, the machine will keep in motion until the manual button is released.

8

2) Under [Stepping] mode, you can also use manual operation device to control machine, e.g. manual pulse generator, keyboard and hand-held box, but the situation is different from [Jog]



mode. Each time you press the key, the machine will move a specified distance until the button is released. The displacement of the machine can be accurately controlled in this mode.

3) Under [Handwheel] mode, you can control the machine tool through handwheel operation. Every time you turn the handwheel a lattice, the machine will move a stepping length.

## 3.1.2. Operation State

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

#### ♦ IDLE State

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

#### **♦** E-STOP State

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

#### Running State

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

### ◆ Pause State

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

#### **♦ LOCK State**

As an internal state, lock state is rarely seen under normal circumstances and only exists during state-switching.



## 3.2. Coordinate System

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

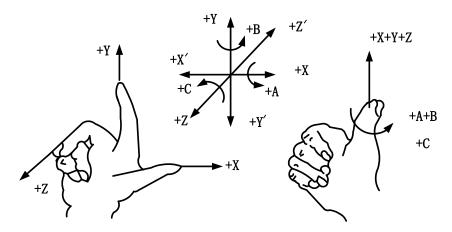


Fig. 3-1 The right hand principle of standard coordinate system

For milling machines, the directions of machine axes are decided by both the type of the 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 is coincidental with spindle axis and the direction of 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 vertical milling machine, if you face the spindle and look in the column direction, right moving direction is the positive direction of X-axis (+ X).

——X-axis, Y-axis and Z-axis constitute the coordinate system following the right-hand rule.

## 3.2.1. Machine Coordinate System (MCS)

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

The machine coordinate system requires the machine with function of [back to reference point], or this term will only appears in the software.

## 3.2.2. Workpiece Coordinate System (WCS)

It is more often to adopt workpiece coordinate system at machining various kinds of workpiece.



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



# 4. Operation Interface of NcStudio

Operation interface of NcStudio is constituted by six major sections. They are title bar, menu bar, numerical control state bar, information display bar, function window, and operation panel. This version witnesses a great change in operation interface compared to the previous editions.

- Picture display is more intuitional and detailed. Tracks can be shown under every operation mode. All processing demonstration is visible in the coordinates screen. Consequently, the accuracy and instantaneity of the processing view is ensured to the highest degree.
- > The system provides powerful visual function, user-friendly interface, distinct style, and rich interface patterns. You can freely select the theme of interface or even design your own style.
- The button tip function is added. When you move the mouse to a button, a hovering tip will appear to tell you the button's function or its shortcut key.
- > The added simulation operation panel provides rich modes for option, which can satisfy different requirements.
- Single shortcut key mode makes the operation easier.

## 4.1. Overall Interface

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

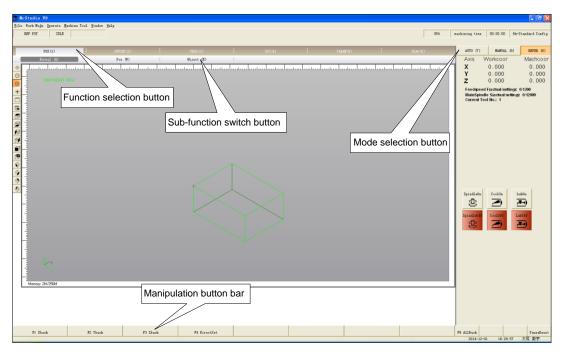


Fig. 4-1 The main operation interface



### 4.2. Title Bar

The first column of NcStudio interface is the title bar, shown in Fig. 4-2. It is used to show the name of the application. Its color indicates whether the corresponding window is activated or not.



Fig. 4-2 Title bar

#### Note:

An active window refers to the window which can accept the input of keyboard at present. Anytime only one window can be active, and all the other windows are inactive.

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

Click the right mouse button or press shortcut "Alt + spacebar" to eject a system menu. This menu can control the position and the size of the window, such as restore, move, close, maximize and minimize, etc. On the right side of the title bar, there are three control buttons, including button for closing, button for maximizing and button for minimizing. These buttons are used for quickly setting the size of the window.

## 4.3. Menu Bar

Under the title bar is the menu bar, see Fig. 4-3.

The menu bar is composed of several pull-down menus including multi-menu items; each menu item can realize a special function, action, or state that is relative to a certain procedure. And the menu item can be selected by mouse or keyboard.

The NcStudio menu bar includes menus of file, mode, operate, machine tool, window and help, through which you can conveniently implement the action of file processing, mode selecting, processing operating, machine-related operating, windows switching, and help information obtaining, etc.

### Mouse operation

Firstly, left click the menu on the menu bar to eject a pull-down menu, and then left click to choose the target item.



#### Keyboard operation

Simultaneously press down 'Alt' key and hot key of the selected menu (For example, there is a line under 'F' in the "File" menu, so the combination key is "ALT+F"), after which a relative pull-down submenu will pop up. Regarding the item in the pull-down submenu, you can also use the keyboard to choose the target item. Take [Open and Load] under [File] menu as an example. "(O)" indicates when the pull-down menu of [File] appears, directly pressing the letter 'O' will open the process file.

#### ♦ Shortcut key operation

In the pull-down submenu, there are relative shortcut keys on the right of some menu options. For example, [File]  $\rightarrow$  [Open and Load (O)...], the shortcut key is 'Ctrl + O'. That is to say, you can directly execute the corresponding action by pressing the relative shortcut key, which can avoid the trouble to enter into multilayer menu.

Behind some menu options, there follow three dots (For example, [File]  $\rightarrow$  [Open and Load (O)...]), which indicates that selecting them will eject a dialog automatically. If an item of the submenu shows in grey, it indicates that the item is unavailable under the current state.

In addition, right clicking in different window positions will eject a context shortcut menu. You can choose the relative item to perform the corresponding command.

## 4.4. NC (Numerical Control) State Bar

Under the menu bar is the NC (Numerical Control) state bar, see Fig. 4-4.



The embedded prompt boxes on the NC state bar display operation mode, operation status, other appended information, processing time and configuration information respectively. It gives you an intuitional way to learn the current machine state and the processing time for further action.

## 4.5. Information Prompt Bar

Under the NC (Numerical Control) state bar is the information prompt bar, see Fig. 4-5.



Fig. 4-5 The information prompt bar

On the prompt bar, three kinds of information can be shown. They are normal prompt information, warning prompt information and error prompt information, which are mainly distinguished by the characters and their color on the information prompt bar.



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

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

#### Note:

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

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

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

## 4.6. Function Window

Function window is the main component of NcStudio V9 interface. It is used to display different kinds of function screens, including six powerful function sections and related manipulation button bar, such as [Position], [Offset], [Program], [System], [Parameter], and [Diagnosis]. And you can press down the [function selection button] (see Fig. 4-6) to enter into the corresponding function section.



Fig. 4-6 Function selection button

Each function section has its relative sub-functions. For example, under the [Offset] function section, there are sub-functions of [Workpiece Coordinates], [Offset], [Save & Load]. Under the [Program] function section, there are screens of [HD List], [FD List], [ProcessWiz] and [History record]. To exchange between function screens, please press the [Subfunction Exchange Button]. See Fig. 4-7.

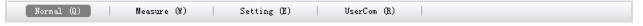


Fig. 4-7 Subfunction exchange button



## 4.7. Manipulation Button Bar

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

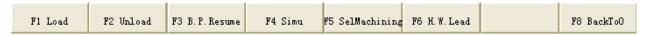


Fig. 4-8 The manipulation button bar

What it usually shows are the shortcut buttons relating to the current function screen or the current mode. For instance, under the function screen of [Position→[Normal], the manipulation button column has buttons of [Load], [Unload], [B.P. Resume], [Simulation], [Sel Machining], [H.W. Lead] and [Back to origin], etc.

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

#### Note:

Unless the conditions are satisfied, some buttons in the manipulation button column are disabled to prevent false operation;

V9 NcStudio can execute operations with a single shortcut key and show the single key hint on [sub-function exchange button], [function selection button], [mode selection button] and other operation buttons. What you need to do is to press down the prompted key, and then the system will respond accordingly;

Take [Object (E)] button as an example, what you need to do is to press down the [E] key of the keyboard, and then the system will jump to the corresponding window;

The default single shortcut key, [Function selection button], ranges from 1 to 6;

The default single shortcut key, [Mode selection button], ranges from 7 to 9;

The default single shortcut key, [sub-function exchange button], ranges from left to right order of Q, W, E, R, T, Y;

The common manipulation shortcut keys of software contain X, Y, Z, H, J, K, L, U, I, O;

On the operation panel, B, N and M are the single shortcut keys of the ratio slider. These three keys can make the cursor transfer to the override slide button, which can be operated by the button of Page Up [PgUp] /Page Down [PgDn];

Mastering these shortcut keys will make machining easier and less time-consuming!

## 4.8. Operation Panel

According to the mode differences, the operation panel can be divided into automatic mode, manual mode and reference point mode, shown in Fig. 4-9. At the upper part of the operation panel, there is coordinates displaying section where you can see the workpiece coordinates and the



machine coordinates, the current feed rate, the current spindle speed and the name of the currently loaded file, etc. Under the coordinates displaying section is the panel button area. Buttons inside this panel button area will be different under different modes.

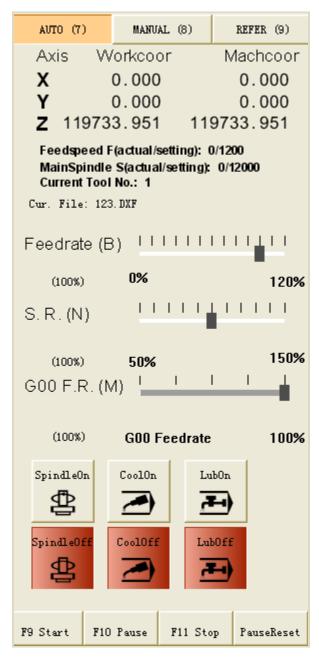


Fig. 4-9 Operation panel



## 5. Auto Mode of NcStudio

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

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

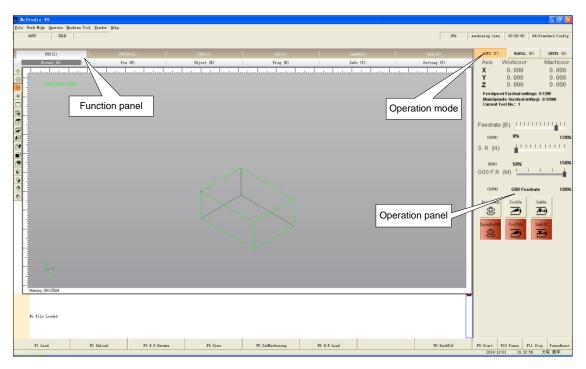


Fig. 5-1 Auto operation mode

## 5.1. Operation Panel Section

This operation panel is composed of several auto manipulation buttons for controlling the machine tool automatically. See Fig. 5-2.



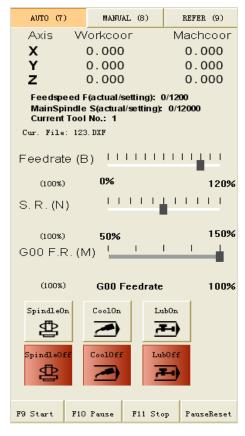


Fig. 5-2 Auto operation panel dialog

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

Under auto mode, button zone includes the following contents:

### ♦ [Feedrate] sliding button

It is shown as Fig. 5-3.



Fig. 5-3 Feedrate sliding button dialogue

You can manipulate the feed speed by controlling current feedrate override. The formula is as following:

Current feed speed = Setting feedrate x Current feedrate override.

The least unit of feedrate override scale is 10% (Each case indicates 10%). Current feedrate override is displayed beneath the word "Feedrate". Three ways can be used to revise the feedrate override. Click the sliding axle; drag the slide block; click the slide block to choose it, then press



[PageUp], [PageDown] key; each time [PageUp] or [PageDown] key is pressed, the block will move one case, and the feedrate override will increase (or decrease) by10%. The range of feedrate override is 0% ~ 120%.

#### Spindle override sliding block

The picture of [Spindle Override Sliding Block] is shown in Fig. 5-4.



Fig. 5-4 Spindle override sliding button dialogue

You can control the spindle speed by adjusting the current spindle override. Its calculating expression is as following:

Current spindle speed = Current spindle override x Spindle speed.

The least unit of the sliding axle of spindle override is 10%, and the axle will give prompt about percentage below the word "S. R. (N)" after adjusted. You can control the sliding axle by mouse or keyboard. Mouse method: click the left key of mouse on the sliding axle, or draggle the block of the sliding axle. Keyboard method: press the Page Up (Page Down) key of keyboard when the sliding axle is selected; each time [PageUp] or [PageDown] key is pressed, the spindle override will increase (decrease) by 10%. The range of spindle override is 50%~ 150%.

### ♦ G00 F. R.(M) sliding button

The picture of [G00 F. R. (M) sliding button] is shown in Fig. 5-5.

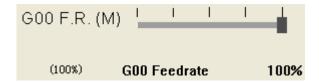


Fig. 5-5 G00 F. R. (M) Button Dialogue

You can manipulate the G00 spindle speed by adjusting the G00 override. When G00 override is adjusted to 0%, the actual G00 override is equal to the feedrate override, and the sliding axle will give the prompt "G00 ratio= Feed rate" below the word "G00". Its calculating expression is:

Current tool traversing speed under G00 command= Feedrate override \* Machining speed.

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

Current tool traversing speed under G00 command = G00 override \* G00 speed.



The least unit of the sliding axle of G00 override is 25% (Each case indicates 25%). And you can control the sliding axle by mouse or keyboard method. Mouse method: click the left key of mouse on the sliding axle, or draggle the block of the sliding axle. Keyboard method: press the Page Up (Page Down) key of keyboard when it is selected; each stroke will increase (or decrease) the override by 25%,G00 overrode should be within the range between 0% and 100%.

### ◆ [Spindle on], [Spindle off], [Coolant on], [Coolant off], [Lubrication on], [Lubrication off]

T buttons on the operation panel are shown in Fig. 5-6:



Fig. 5-6 The Start/Stop button dialog of Spindle, Coolant Valve, and Lubrication Valve on operation panel

These buttons respectively control the switch of spindle, coolant valve, and lubrication valve. On the front row are the start buttons, with character prompts on the top. When one of them is pressed down, the background of the button turns green. On the back row are the close buttons, also with character prompts on the top. When one of them is pressed down, the background of the button turns red.

Under the mode of auto, manual, and reference point, the operation panel displays the start / stop buttons all the same.

#### ◆ [Start]

The button is shown as

The shortcut key is F9, only available under auto mode. If you have not loaded the processing file, the button is invalid; if you load the processing file and accomplish the operation of going back to the machine origin, the button is valid. Pressing down the button will start processing from the first line automatically. If you have not finished the reference point operation ("back to machine origin" or "directly setting") in loading the file for the first time, the system will prompt you to go back to the machine origin after you press this button.

This button has two functions:

If a certain processing file has been loaded, reference point operation has been finished, and the system is in "idle" state the machine tool will carry out the processing procedure automatically



from the first section. Once processing starts, the system will enter into the "Auto/ run" status. If the system is under simulation, the system starts simulation.

➤ If the system is under the "auto| pause" state, pressing down the button will carry out machining continuously from the pause position and make the system enter into "auto| run" state. If the system is in the status of simulation, the system starts simulation.

#### ◆ [Pause]

The button is shown as:

In automatic processing, the "pause" button is valid. This button pressed, the machine tool will pause and raise the cutter and then turn to the state of "Auto/pause". To keep on processing, press down the button "start" or select the relative menu item.

If the system is under simulation status, pressing down the button will suspend simulation and make the system turn to "Auto/pause" state. To keep on simulating, press down the button "start".

Shortcut key: F10, only available for AUTO mode.

#### Note:

Two methods are available for the system to enter into [Pause] state:

- 1. When the system is executing single block function;
- 2. [Pause] function is selected during machining process.

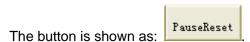
#### ♠ [Stop]

The button is shown as:

This function is enabled under the auto mode or when you input G code and execute it. Pressing down the button will stop machining and turn the system into the state of "Auto/idle". This method is to stop the processing in a normal way.

Shortcut key: F11, only available under AUTO mode or when G code is entered and executed.

### ◆ [Reset]



The button is valid under all circumstances. Pressing down the button will make the machine tool enter into idle state from warning state (if it is in "alarm" state) and terminate the whole assignment immediately (if the machine tool is under processing).



Shortcut key: Pause, available for every mode.

## 5.2. Function Panel Section

Under auto mode, the function panel buttons are as following, see Fig. 5-7:



Fig. 5-7 Coordinate function section of auto mode

### 5.2.1. Position Function Section

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

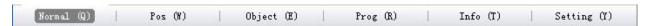


Fig. 5-8 Subfunction exchange button of [position] in Auto mode

#### Normal function screen

Normal function screen is shown as Fig. 5-9.

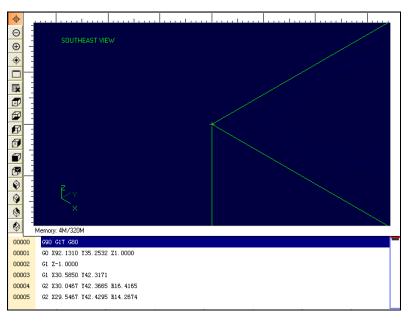


Fig. 5-9 Normal function screen

In the process track window, the path of the tool is tracked in real time. You can directly check three-dimensional tool's track and make sure the machining procedure executed accurately. Besides, in three-dimensional track mode, NcStudio supports abundant operating methods to help you to look over the figure from different positions to a proper scale.

### ♦ [Icon Button]

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





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

V9 NcStudio adds a tip for each of these icon buttons. Moving the cursor above a button will display a tip, informing you of the button's function.

# > [Move]:

Keeping pressing the left mouse key can drag and move the tracking figure; Click the icon [Move], move cursor to the display window, the mouse showing; click the left mouse key, the mouse showing ; drag the mouse slowly, and loose it after completing the moving of the whole track figure.

You can also implement this function by keyboard. When the [move] icon button is pressed down, press the enter key, and then press the direction key to move the whole track figure. When the mini-keyboard is unlocked, the mini-keyboard can be used as direction key.

Note:

Since the operation is very convenient, skillful practice may raise the working efficiency substantially.

> [Zoom In] and [Zoom Out]

Click the [Zoom in] or [Zoom out] button on the left of the process track window, then move the mouse into the display area. When the mouse shows • , click the left key of the mouse to enlarge



the figure; when the mouse shows , click the left key of the mouse to reduce the figure.

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

#### Notice:

When the track view is zoomed in or out by mouse, you can enlarge or reduce part of the track view by clicking mouse on the selected area.

If the track view is enlarged to the ceiling, the view will be automatically switched to [zoom out] mode.

If the track view is reduced to the floor, the view will be automatically switched to [zoom in] mode.

[Show Current Processing Point]:



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

Shortcut key: [End] key on the keyboard.

[Adjust to Window Size]:

Click button [Adjust to Window Size] on the left of the processing track window. The window will adjust to displaying all the paths and making path center in its midpoint. With this method, you can see the whole figure without scrolling the bar.

Shortcut key: [/] key on the keyboard.

[Clear/ Delete]:

After a long time of machining, the simulation figure will become very complex and the temporary folder used to record processing path will become more and more bulky. As it is time-consuming to redraw, move or revolve the simulative graph, you need to clear the machining track regularly. The actual processing track is cleared after you press the button.

After the [Clear] button on the left is clicked, the simulation figure in the track window will be deleted.

Shortcut key: [Delete] key on the keyboard.

View button group, shown in Fig. 5-11 below:





Fig. 5-11 View button group

This group of buttons is used to observe the angle of view of track. When you move the mouse above the group of buttons, a hovering tip will appear to tell you the corresponding angle of view. You can choose a suitable angle of view to observe the processing track to facilitate detection of processing cutter path and processing state.

When you move the mouse to the track window, a hovering tip will pop up—shortcut key: S; when you press down S key, the system will eject the view navigator, shown in Fig. 5-12 below. At this moment, you can operate it according to the single shortcut key on the navigator. "S": front view; "W": top view; "X": bottom view; "A"; left view; "D": right view; "Q": northwest view; "E": northeast view; "Z": southwest view; "C": southeast view.

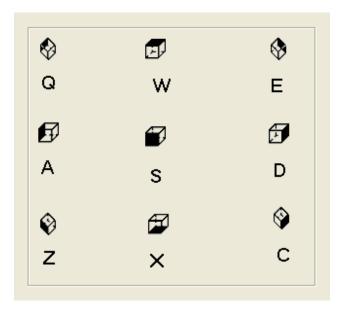


Fig. 5-12 View navigator

#### ♦ Shortcut menu

Click the upper track window of the general window with the right key of mouse, and you can



execute the related operation on the pop-up menu, shown in Fig. 5-13 below:

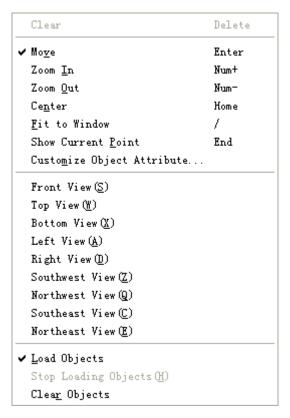


Fig. 5-13 The shortcut menu of the track window

Functions of these quick menus will be illustrated separately as following (introduction to icon functions will be omitted here).

#### ➤ [Clear]

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

- Menu method: choose the menu of "Edit (E)| Clear View (C)".
- 2) Hot key method: press down the key of "CTRL+ DEL" to realize the clearance of track window.
- 3) Mouse method: move the mouse into the manifestation area, click the right key of the mouse, to pop up a menu and choose the option of "Clear".
- 4) Keyboard method: press the key of Delete while the trace window is the current active window.

#### [Center]

It shows current machining scope in the center of the machining track window.

1) Menu method: choose "View | Center View".



- 2) Mouse method: right click in track window to eject a context menu, choose "center" option.
- 3) Keyboard method: activate track window firstly, and then press "Home" key on the keyboard.
- 4) Shortcut key method: press "Home" key on the keyboard, or right-click to eject quick menu and then press "N" key.

## [Property]

Clicking [property (M)] on shortcut menu will eject a [custom parameter] dialogue, shown in Fig. 5-14 below. This window is used to customize track window to set track mode and color:

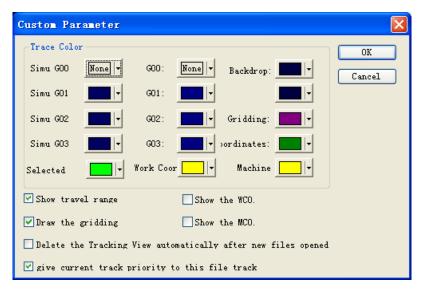


Fig. 5-14 Property dialogue box

Click one of the color buttons, and you will see the following color selection box, as Fig. 5-15.

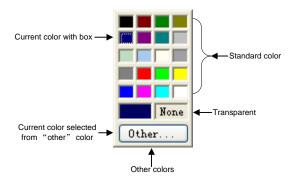


Fig. 5-15 Color selection dialogue

On [Trace Color] board, you can customize the color of backdrop, instruction, grid, coordinate axis, the selected track, etc. In V9 NcStudio, a new function of loading path in advance has been added, so the path colors are divided into simulation path color and real path color. [Simu G00/G01/G02/G03] is corresponding to simulation path color, while [G00/G01/G02/G03] is corresponding to real path color.



#### Prompt:

If the selected color of the command is transparent, the command will be concealed. If the background color is set as transparent, the track window will not be refreshed rightly, so you must be serious in selecting "transparent" color.

## [Travel Range Display]

The function means whether the simulation worktable is shown in the trace window. Selecting this item will show the simulation worktable, otherwise it won't be displayed.

## [Draw Grid]

The function means whether the grid is shown in the track window. Selecting this item will show the grid, otherwise it won't be displayed.

## > [Delete Tracking View Automatically after New files opened]

The function means whether the real track is deleted automatically after you open a new files. Selecting this item will clear the view automatically, otherwise it won't be cleared.

## Give Track Priority to Current File Track

When you load a processing file and start machining, both real processing trace and pre-simulated processing trace will be shown in the trace window. But the [center function] and [adjust to the window function] are only effective for current trace, so you can decide which one is current trace. If this item is selected, it will give priority to the simulated track, or, give priority to the real track.

## [Load Objects]

This function will load figure information of the processed object.

#### [Stop Loading Objects]

During loading machining object information, this function will stop loading the rest.

#### [Clear Objects]

This function will clear figure information of loaded machining object.

On the lower part of the [Normal] function screen, there are related operation buttons of current operating mode, machine status, and running state of processing file, etc.



#### Note:

Under auto mode, the lower part of [Normal] function screen displays the loaded processing file. When auto machining begins, the currently being executed instruction will be highlighted with deep blue as following. The frame in front of the instruction is used to mark the current instruction's position in the whole processing file.



#### Position Function Screen

As Fig. 5-16, this screen is used for looking over the position of tool, including machine coordinate, workpiece coordinate, and "distance to go". When current G code is executed, the absolute difference between current position and destination's position is called "distance to go", only available under auto mode.

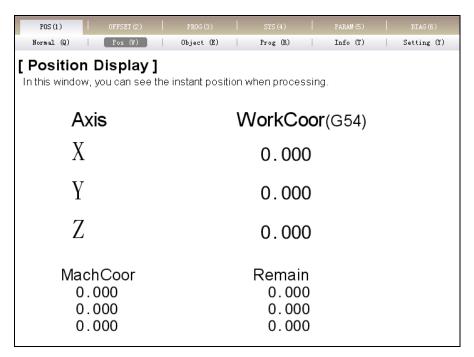


Fig. 5-16 Pos function screen

## **♦** Object Function Screen

This window is mainly used for the machining track display, and the left icon buttons are the same as those of [Normal] function screen, please refer to the introduction of [Normal] function screen for details.



## Program Function Screen

This screen is mainly used for displaying the loaded processing file, and the ID column identifies the related position of current command in the whole machining file, as Fig. 5-17. The [Edit Loading File] button in the lower part of the screen is not available until a processing file is loaded. Pressing the button when it is enabled will eject the NcEdtior, where you can edit the processing file.

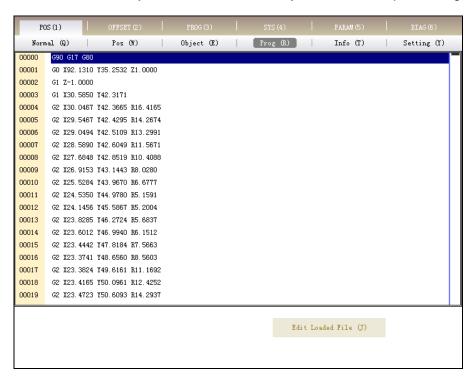


Fig. 5-17 Program function screen

#### **♦** Information Function Screen

As Fig. 5-18, this screen is mainly used for displaying various kinds of information during the processing procedure: such as the maximal processing range, cutting time, and number of machining workpiece, current executed instruction's position, and the length of processing file and the path of processing file, cutting length statistic of cutter. Cutting time only includes time of cutting. Amount of processing workpiece means the times of completing the processing file. Pressing the button will reset the counter. All the information is clear. Cutting length records the total length of cutter path, and the processing time is shown in state column which is convenient for user to look over.



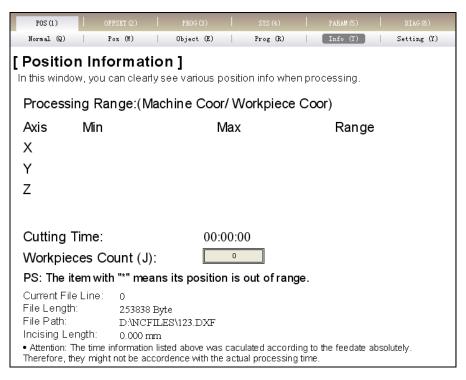


Fig. 5-18 Information function screen

## Setting Function Screen

This function screen is mainly used for setting the parameters in auto mode, such as dry running speed (i.e. G00 speed), processing speed, spindle speed, speed mode specified, ignore spindle speed in program, cycle times of program, time interval between machining file cycle and quality-efficiency regulation, shown in Fig. 5-19.

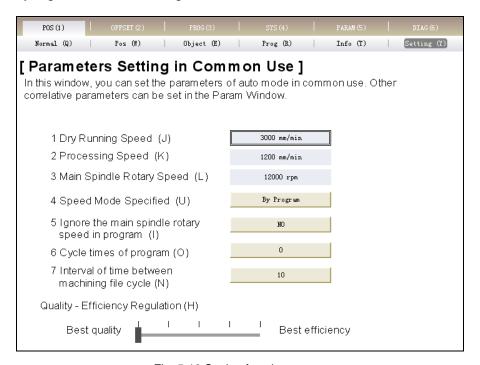


Fig. 5-19 Setting function screen



## [Dry Running Speed]

This item sets the running speed of G00 code, its setting range within 1~100000. If speed is set beyond the range, the system will prompt you. Pressing the button ejects an inputting dialogue box. The dialogue box is commonly used, shown in Fig. 5-20. Supporting the function of "copy", "paste", "calculating expression", and shortcut keys, the input dialogue box brings great convenience in calculating the numeral of multi-digit.

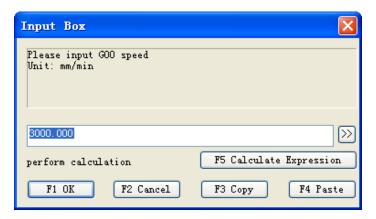


Fig. 5-20 Input box

## [Processing speed]

Processing speed refers to the moving speed under G01, G02 and G03 commands, its range within 1~ dry running speed. And the system will prompt you if the speed is set beyond the range.

#### Spindle Speed]

This item refers to spindle speed in machining.

## Speed Mode Specified]

[Speed mode specified] refers to whether to ignore the feed speed specified by processing file, use the default feed speed in the system and specify speed in proportion. Press the button to eject an input dialog. You can modify the parameter according to the prompt, as Fig. 5-21.



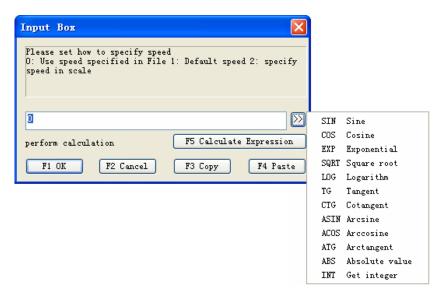


Fig. 5-21 Spindle speed specified input dialogue

## [Ignore Spindle Speed in Program]

This item refers to whether to use default spindle speed.

## [Cycle Times of Program]

This parameter is to specify the cycle times of execution of the loaded file.

## ➤ [Time Interval between Machining Time Cycle]

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

#### [Quality-efficiency Regulation]

The relationship between quality and efficiency is relative without concrete ratio; when the processing efficiency is high, the processing quality is relatively low, and vice versa.

#### Manipulation Button Bar

Under AUTO mode, all the manipulation buttons in the [POS] function section are the same, shown in Fig. 5-22:

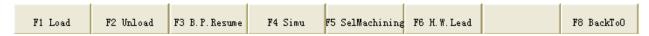


Fig. 5-22 Manipulation buttons of Pos function section in AUTO mode

## > [Load]

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



## [Unload]

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

#### [Breakpoint Resume]

Shortcut key: F3, only available under AUTO mode. This button is not enabled until power off, E-stop, or normal stop in the midst of machining. If you can ensure the accuracy of workpiece coordinate, you can press this button to continue processing. If not, please return to machine origin firstly and then restart the processing. The function also can be used in simulation mode.

## [Simulation]

Shortcut key: F4, only available under AUTO mode. The function of simulating provides a fast but lifelike simulated processing environment for you. If the processing procedure has been loaded, and the current operating mode is "IDLE", pressing down this button and then "start" button will carry out simulating at high speed automatically from the first section. Simulation is just like demo function in CNC system, but it's superior to demo function. Running under the mode of simulating, the system will not drive the machine tool to do the relative actions but show the processing trace of the cutter at high speed in the trace window. Through simulation, you can view the machine tool's movement path in advance to avoid equipment damage caused by programming errors and learn other information. For related information, refer to [Info.] function screen under the [POS] function section.

## [Selected Machining]

Shortcut key: F5, only available under AUTO mode. This function is used for execution of advanced start function. Pressing down the button will eject the [execute (advanced options)] dialogue box, shown in Fig. 5-23. You can execute any block in processing program by inputting the concrete start line number and the end line number for current procedure. When you select [file start] or [file end], the detailed file position will be disabled. This function can also be used for simulating.



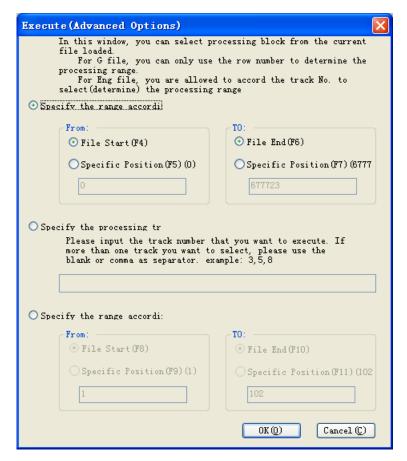


Fig. 5-23 Selected machining

#### [Handwheel Lead]

Shortcut key: F6, only available under AUTO mode. Press down this button to control the motion of each axis of the machine tool only by MPG and disable the buttons on the operating panel.

#### [Back to Origin]

Shortcut key: F8, only available under AUTO mode. Pressing down the button will control the tool to return to the workpiece origin. Please pay attention to the differences among workpiece origin, fixed point and machine origin.

## 5.2.2. Offset Function Section

The function section is used for completing the related operations of workpiece coordinates offset. [Offset] section includes WorkCoor (Coordinate), Offset, S. & L. (Save & Load) function screens, etc.

#### ♦ Coordinate Function Screen

As Fig. 5-24, you can set the current workpiece coordinate under this screen. Click the button to eject the coordinate input dialog, and then enter the value and click "OK", the system to adjust the coordinate automatically (without actual position movement). And then the precise distance and



moving direction will be shown.

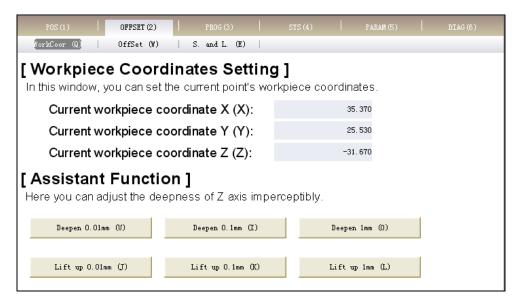


Fig. 5-24 Function screen of workpiece coordinate setting

[Auxiliary function] refers to fine tuning of Z-axis, and you can choose the desired button freely. After the button is pressed, the prompt on the moving direction and the accurate moving distance of Z-axis will be given in the blank region.

> [Deepen 0.01mm], [Deepen 0.1mm], [Deepen 1mm]

Manipulate Z-axis to move downward 0.01mm, 0.1mm and 1mm accordingly.

► [Lift up 0.01mm], [Lift up 0.1mm], [Lift up 1mm]

Manipulate Z-axis to move upward 0.01mm, 0.1mm and 1mm accordingly.

## **♦** Offset Function Section

This picture is used to set public offset and the offset of MCS and WCS (G54, G55, G56, G57, G58, and G59), as Fig. 5-25.



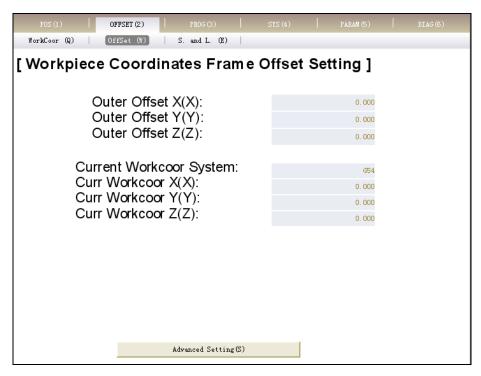


Fig. 5-25 Offset setting function screen of workpiece coordinate system

The X, Y, Z of workpiece offset is the offset of workpiece origin with respect to machine origin, which equals to machine coordinates of X, Y-axis when the tool nose moves to workpiece origin. Only G54 coordinate system is supported.

Public offset: it is used to record the temporary adjustment value of the workpiece origin. The value can only be revised artificially, not automatically. Whether public offset exists or not will not make any difference to fixed measurement or mobile measurement. In other words, if public offset exists, after mobile measurement is finished, the workpiece coordinates of workpiece surface will not change to zero.

The supported workpiece coordinate systems include G54, G55, G56, G57, G58, and G59.

Pressing "advanced settings" button in the picture above will eject a dialog for offset setting. See Fig. 5-26.



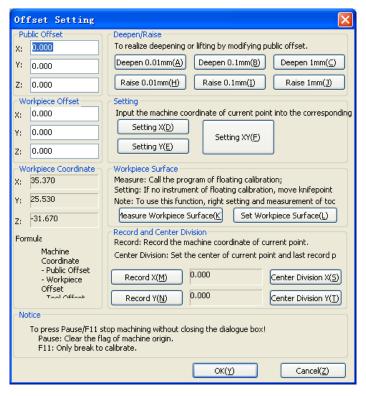


Fig. 5-26 Offset setting picture



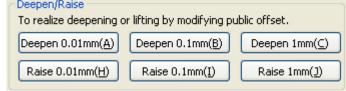
Offset

The functions of [workpiece offset] and [public offset] are the same as those in [Offset] section.



Coordinate Display

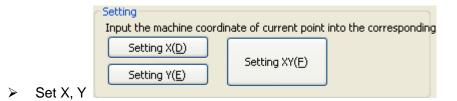
It displays the workpiece coordinate of current point.



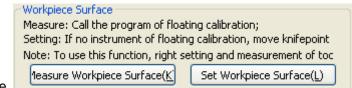
Z-axis Adjustment

Pressing the button will move the workpiece origin of Z-axis up or down by the specified distance to form a new workpiece coordinate system. Through several strokes continuously, the moving distance will be the accumulated value of several adjustments.





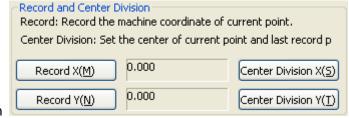
Set X and Y coordinates of the current point, or enable X and Y coordinates as workpiece coordinate origin.



## Workpiece Surface

Since the function of mobile measurement and "measure workpiece surface" are identical, the system will directly call out mobile measurement program.

"Set workpiece surface": press this button when the tool touches workpiece surface to clear the Z workpiece coordinate of the current point.



#### Record and Center Division

Record and center division is used to get the center of two points. When the tool goes to the first point, click button "Record X", the system recording the X machine coordinate of the point; when the tool goes to the second point, click button "Center Division X", the system calculating the X machine coordinate of the two point center. The functions of "Record Y" and "Center Division Y" are similar with "Record X" and "Center Division X" respectively.

#### **♦** Save and Load Function Screen

See Fig. 5-27, you can save and load the offset value of workpiece coordinate in this screen. The saving window is marked with group NO., saved time and file name. When you save or load a record, pressing the related button will eject a dialog box to prompt you.



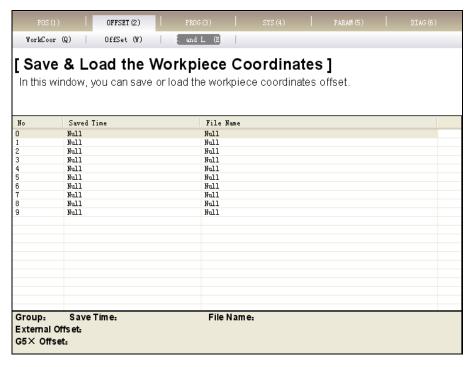


Fig. 5-27 Save & Load workpiece coordinates function screen

## Manipulation Button Bar

Since the manipulation buttons of [offset] section are a little different in different function screens under auto mode, they will be introduced separately.

The picture of manipulation buttons in [workpiece coordinate] is shown in Fig. 5-28:

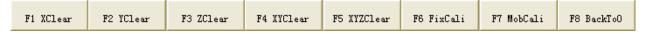


Fig. 5-28 The manipulation button bar in [workpiece coordinate] under [Offset] section

## [X Clear]

The shortcut key is F1. After you press the shortcut key, the system will modify offset to ensure that the X workpiece coordinate of the current point is zero.

## > [Y Clear]

The shortcut key is F2. After you press the shortcut key, the system will modify offset to ensure that the Y workpiece coordinate of the current point is zero.

#### > [Z Clear]

The shortcut key is F3. After you press the shortcut key, the system will modify offset to ensure that the Z workpiece coordinate of the current point is zero.



## [XY Clear]

The shortcut key is F4. After you press the shortcut key, the system will modify offset to ensure that the X and Y workpiece coordinates of the current point are both zero.

#### [XYZ Clear]

The shortcut key is F5. After you press the shortcut key, the system will modify offset to ensure that the X, Y and Z workpiece coordinates of the current point are all zero.

## [Fixed Presetting]

The shortcut key is F6, only available under [Offset] section. Only after the machine tool returns to machine origin is the fixed measurement enabled. This function is used for calibrating Z coordinate of workpiece origin after tool change.

During tool measurement, at first, ensure workpiece surface artificially, set workpiece surface as Z coordinate of workpiece origin, then click button for F6 FixCali , the system automatically recording current Z workpiece coordinate. In case of tool change or tool break, clicking button again will resume previous Z coordinate of workpiece origin.

#### [Mobile Presetting]

The shortcut key is F7, only available under [Offset] section. Pressing the button will start mobile measurement.

F7 MobCali

Mobile measurement brings you the convenience in the procedure of fixing the workpiece origin of Z-axis. Put the tool presetter on workpiece surface firstly, manipulate Z-axis to touch the top of tool presetter, press [begin measurement] button, the system executing tool measurement automatically and confirm Z coordinate of workpiece origin. This operation can be done according to the prompts.

#### [Back To 0]

Shortcut Key: F8. After the button is pressed, the system will control the tool to go back to the workpiece origin. Please pay attention to the differences among workpiece origin, fixed point and machine origin.

The manipulation button picture in [offset] function screen is as shown in Fig. 5-29.



Fig. 5-29 Manipulation button under [Offset] picture



## ➤ [Set\_X]

The shortcut key is F1, only available in [Offset] function screen under [Offset] section. After the shortcut key is pressed, X-axis workpiece coordinate of current point will be cleared with Y-axis, Z-axis unchanged.

#### ▶ [Set Y]

The shortcut key is F2, only available in [Offset] function screen under [Offset] section. After the shortcut key is pressed, Y-axis workpiece coordinate of current point will be cleared with X-axis, Z-axis unchanged.

#### [Set\_Z]

The shortcut key is F3, only available in [Offset] function screen under [Offset] section. After the shortcut key is pressed, Z-axis workpiece coordinate of current point will be cleared with X-axis, Y-axis unchanged.

## > [Set\_XY]

The shortcut key is F4, only available in [Offset] function screen under [offset] section. After the shortcut key is pressed, X-axis and Y-axis workpiece coordinates of current point will be cleared with Z-axis will be constant.

#### [Set\_XYZ]

The shortcut key is F5, only available in [Offset] function screen under [offset] section. After the shortcut key is pressed, X-axis, Y-axis and Z-axis workpiece coordinates of current point will all be cleared.

#### [FixCali]

The shortcut key is F6. Press this button to begin fixed measurement, which is identical with that of [workpiece coordinate].

#### [MobCali]

The shortcut key is F7. Press this button to begin mobile measurement, which is identical with that of [workpiece coordinate].

## ➤ [Back To 0]

The shortcut key is F8. Press this button to return all axes (X, Y and Z axes) to workpiece origin.

The manipulation button picture in [S. and L.] function screen is shown in Fig. 5-30.





Fig. 5-30 Manipulation buttons in [offset] function screen

#### [Save]

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

#### [Load]

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

## 5.2.3. Program Function Section

The function section is used for completing the operations related to processing file. [PROG] section includes HD List, FD list, ProcessWiz, History function screens, etc.

## HD List (Hard Disk List)

See Fig. 5-31, in this screen, you can look up processing files under default file path on the hard disk and execute a series operations of load, edit, delete, rename and etc. Moreover, you can create a new machining file under file path "D:\NCFILES" and edit it.

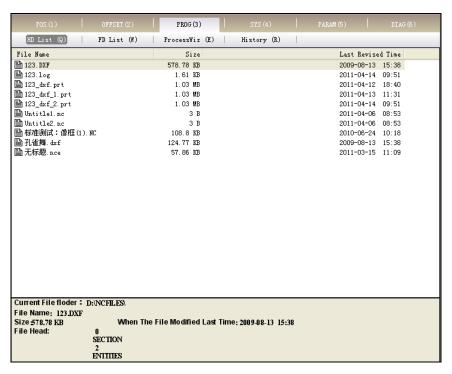


Fig. 5-31 Hard disk list function screen

The upper part is the file list displaying all of the files under the path "D:\ NCFILES", where you can save the processing files for more convenient operation. After a file is selected among the file list, the lower part will prompt some properties of the selected file, such as file (program) name, size, and last revised time and the front lines of the file.



Note:

You can save the processing files to the path of D:\NCFILES, and then directly find them in the HD list and operate them.

## ◆ FD List (Floppy Disk List)

As Fig. 5-32, you can load the files in the floppy disk or flash disk. For instance, the file is under the Naiky folder of flash disk. Series actions of "load", "copy", "select drive" and "safely remove" are available.



Fig. 5-32 Floppy disk list function window

The upper part is a file list; it displays all of the files in floppy disk or external memory storage. And you can execute a series of operations to these files.

After a processing file in the file list is clicked, the lower part will display some properties of the selected file, such as file (program) name, size, and last revised time and the front lines of the file.

#### Processing Wizard Function Screen

NcStudio supplies four basic processing guides aiming at the engraving machine, including circular bottom milling, circular frame milling, rectangular bottom milling, and rectangular frame milling. When using this processing guide, you only need to fill some simple parameters. Then the system will complete the corresponding processing motion automatically. As processing wizard screen varies according to different manipulation buttons, its screenshots will not be given here and its details will be introduced thoroughly later.



#### History Function Screen

If you need to substitute the recent machining file for the current machining file, you can switch to this screen recording the history machining files, shown in Fig. 5-33. To start processing, you only need to load the processing file.

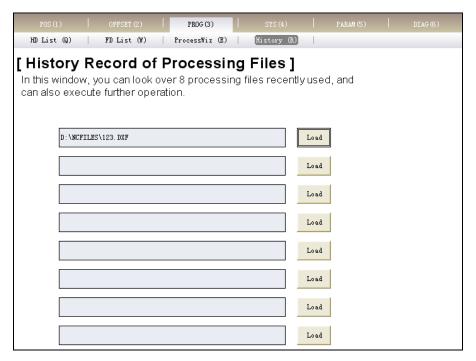


Fig. 5-33 History function screen of processing file

## Manipulation Button Bar

Since manipulation button bar of [PROG] in auto mode varies, it will be introduced separately.

The manipulation button bar of HD list is shown in Fig. 5-34:

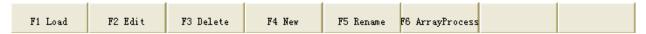


Fig. 5-34 Picture of manipulation button bar in the HD list function screen

## ➤ [Load]

Shortcut key: F1, only available in the HD List function screen of [PROG] section. And you need to select the processing file firstly, then press down the button, the system loading the processing file into the decoder automatically. At the same time, the loading progress bar will be displayed in the information column. After processing file is loaded, the system is ready for processing.

#### ➤ [Edit]

Shortcut key: F2, only available in the HD List function screen of [PROG] section. And you need to select the processing file firstly, then press down the button, the loading progress bar to be displayed in the information column. After the processing file is loaded, the system will switch to



NcCodeEditor for editing.

## [Delete]

Shortcut key: F3, only available in the HD List function screen of [PROG] section. And you need to select the processing file firstly in the file list, then press down the button, the system ejecting a dialog box to prompt you whether to delete the file, shown in Fig. 5-35.



Fig. 5-35 File delete confirmation dialogue

Note:

When the selected processing file is under the state of loading, editing, and processing, the delete function is not allowed.

#### ➤ [New]

Shortcut key: F4, only available in the HD List function screen of [PROG] section. Pressing down the button will create a new processing file named "Untitle1.nc" under the path "D:\NCFILES". And you can also rename the selected file by pressing [rename] button, and edit it by clicking [edit] button.

## > [Rename]

Shortcut key: F5, only available in the HD List function screen of [PROG] section. And you need to select the processing file in the file list, then press down the button, input the file's name into the pop-up dialog box, and then click OK button to complete the function, as Fig. 5-36.

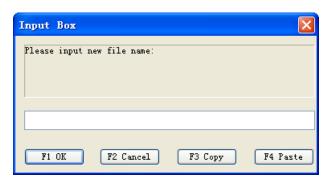


Fig. 5-36 Rename dialogue

## [Array Machining]

The shortcut key is F6, please refer to that of the [Operate] menu in Chapter 8.3 for detail.



In the [PROG] function section, operation buttons of the floppy disk list are shown in Fig. 5-37:



Fig. 5-37 Picture of manipulation button bar in the floppy disk list function screen

#### ➤ [Load]

Shortcut key: F1, only available in the FD List function screen of [PROG] section. After you select a processing program from the file list, you can press this button to load the file into the decoder automatically. At the same time, the loading progress bar will be displayed in the information column. After the processing file is loaded, the system is ready for processing.

## ➤ [Copy to HD]

Shortcut key: F2, only available in the FD List function screen of [PROG] section. After you select the processing program from the file list, you can press this button to copy the file to hard disk.

#### [Select Drive]

Shortcut key: F3, only available in the FD List function screen of [PROG] section. When there are several external memory storages, you can search for the wanted drive by pressing this button.

#### [Safely Remove]

Shortcut key: F4, only available in the FD List function screen of [PROG] section. After using the external memory storages, you can remove them safely by pressing this button. But this function will be unavailable if the file in the device is being used or is loaded into buffer zone.

The manipulation button bar in the [ProcessWiz] function screen of [PROG] section is as Fig. 5-38:



Fig. 5-38 The manipulation button bar in processing wizard function screen

#### [Circular Bottom Milling]

Shortcut key: F1, only available in the [ProcessWiz] function screen of [PROG] section. After the button is pressed, the system will switch to circular bottom milling function picture, shown in Fig. 5-39. You can refer to the processing sketch map in the middle of picture and fill some simple parameters, and then start processing. The system will complete the bottom milling function automatically.



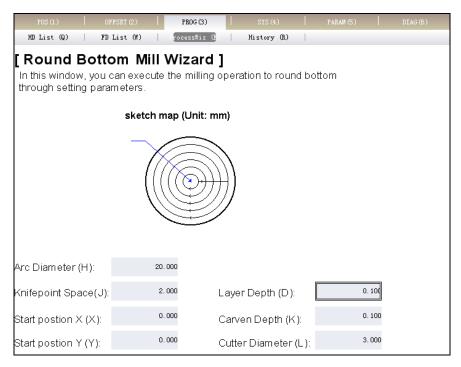


Fig. 5-39 Circular bottom milling function screen

#### [Circular Frame Milling]

Shortcut key: F2, only available in the [ProcessWiz] function screen of [PROG] section. After the button is pressed, the system will switch to circular frame milling function picture, shown in Fig. 5-40. You can refer to the processing sketch map in the middle of picture and select frame milling (internal or external), and fill some simple parameters, and then start processing. The system will complete the circular frame milling function automatically.



Fig. 5-40 Circular frame milling function screen



## [Rectangular Bottom Milling]

Shortcut key: F3, only available in the [ProcessWiz] function screen of [PROG] section. After the button is pressed, the system will switch to rectangular bottom milling function, shown in Fig. 5-41. You can refer to the processing sketch map in the middle of picture and select horizontal or vertical milling, and fill some simple parameters, and then click the start button on the operation panel. The system will complete the rectangular bottom milling function automatically.



Fig. 5-41 Rectangular bottom milling function screen

## [Rectangular Frame Milling]

Shortcut key: F4, only available in the [ProcessWiz] function screen of [PROG] section. After the button is pressed, the system will switch to rectangular frame milling function picture, shown in Fig. 5-42. You can refer to the processing sketch map in the middle of picture and select operation of internal frame or external frame milling, and fill some simple parameters, and then click the start button on the operation panel. The system will complete the rectangular frame milling function automatically.



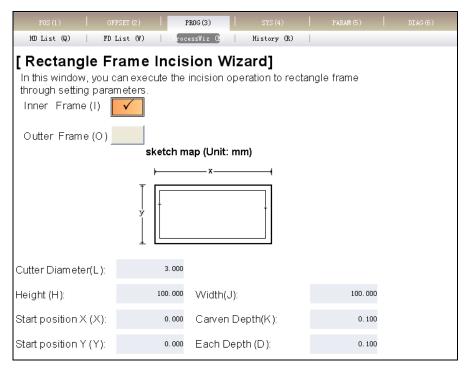


Fig. 5-42 Rectangular frame milling function screen

### [Load]

Shortcut key: F5, only available in the [ProcessWiz] function screen of [PROG] section.

You can select a wizard firstly and then press this button after finishing setting parameters, the system to load the wizard automatically.

#### ➤ [Save]

Shortcut key: F6, only available in the [ProcessWiz] function screen of [PROG] section. If you use a processing wizard with unchanged parameters frequently, then please click the button to save the wizard setting so that you can directly start processing without resetting after loading the wizard next time.

## 5.2.4. System Function Section

The function section is used for related operations of configuration selection, interface selection, language selection, system information, etc. [SYS] section includes "Configuration", "Language", "Interface", and "Information" function screens, etc.

#### Configuration Management Function Screen

The screen displays the configurations supported by NcStudio. You can select the related configuration according to the actual demand by pressing the "select" button, as Fig. 5-43.





Fig. 5-43 Configuration selection function screen

#### **♦** Interface Function Screen

You can select the theme of interface, shown in Fig. 5-44. NcStudio currently supports two themes: Gray and Black. Please press the "select" button to select the function.

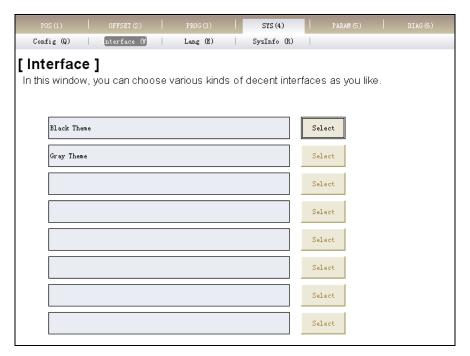


Fig. 5-44 Interface function window

## **♦** Language Function Screen

You can select the language edition in this screen, shown in Fig. 5-45. NcStudio currently supports two language editions: Chinese and English. Please select appropriate language according



to the actual demand. Press down "select" button to accomplish the function.



Fig. 5-45 Language edition function screen

## System Information Function Screen

Under this screen, you can look over the information of NcStudio system, including version, developer, hardware description, card number, remaining time and self test, shown in Fig. 5-46.



Fig. 5-46 System information function screen

#### **♦** Manipulation Button Bar

56

The manipulation button only exists under the [SysInfo] function screen of [SYS] section, shown in Fig. 5-47.





Fig. 5-47 System information manipulation button

## [Hardware Registration]

Shortcut key: F1, only available in the [SysInfo] function screen of [SYS] section. After the button is pressed, the system will eject the "register" dialogue box, shown in Fig. 5-48. You just need to fill the register code into the dialogue box to accomplish the registration.

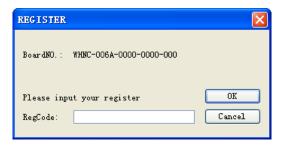


Fig. 5-48 Register dialogue box

## 5.2.5. Parameter Function Section

The [Parameter] function section is used for parameters setting, parameters backup and auto backup, including [General Parameter], [Parameter Backup], [Auto Backup] function screens, etc.

#### General Parameter Function Screen

There are three sub-screens in it. According to the grade of permission, you can enter different sub-screens to set parameters, as Fig. 5-49.

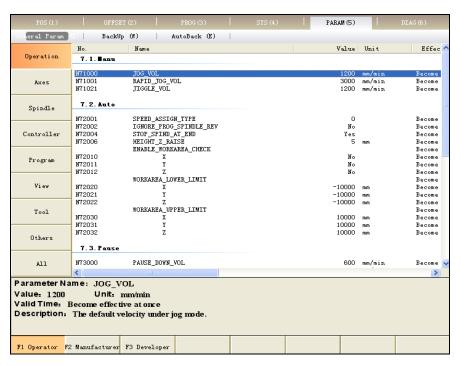


Fig. 5-49 Operator's parameter picture of general parameter function screen



In V9 NcStudio, you can look up parameters based on the type. If you are not familiar with parameter type, please refer to chapter 9 where all the parameters are introduced in numerical order.

V9 NcStudio system adopts a new method of marking the parameters with 5 decimal digits number. While the fist digit stands for main class No. (1-9), the second digit represents subclass No. (0-9) and each subclass contains 1000 numbers (000-999). The type name is given according to the first and the second digit No..

The buttons on the left are the further classification of the parameter types so that it will be more convenient for you to locate the needed parameter and learn its function.

## [Operation Parameter]

Operation parameters include parameters related with actual processing. Clicking the button will display operation parameters list, including N7.1 (manual), N7.2 (auto), N7.3 (pause), N7.4 (reference point), N7.5 (measure), and N7.9 (other operations), etc.

#### [Axis Parameter]

Feed axis parameters include the parameters related with feed axis setting. Clicking the button will display feed axis parameters list; operator's parameters include N1.4 (rotary axis), N7.2 (feed axis related parameter under auto mode), apart from them, manufacturer's parameters also include N1.0 (feed axis normal parameters), N1.2 (compensation), and N1.3 (speed/acceleration limit), etc.

#### Spindle Parameter

Spindle parameters include parameters related with main spindle. Clicking the button will display spindle parameters list, operator's parameters include N7.2 (auto), N7.3 (pause), apart from them, manufacturer's parameters also include N2.0 (spindle normal parameters), etc.

#### [Controller Parameter]

Controller parameters include setting parameters connected to external controller. Clicking the button will display controller parameters list, including N5.2 (handwheel). Only the manufacturer and developer have the right to modify this type of parameters, so the operator column is blank.

#### [Program Parameter]

Program parameters include parameters related to processing procedure setting. Clicking the button will display program parameters list, including N6.1 (Coordinate offset), N6.2 (G code options), N6.4 (speed control), N6.5 (file translation), N6.6 (tool change). Apart from them, manufacturer's parameters also include N6.3 (track control), and N8.7 (show interface).



## [Interface Parameter]

Interface parameters include parameters of interface setting. Clicking the button will display interface parameters list, including N8.1 (position interface), etc.

#### [Tool Parameter]

Tool parameters include parameters related with tool. Clicking the button will display tool parameters list, including N3.1 (tool magazine), N3.2 (tool), N6.6 (tool exchange), etc.

## [Other Parameter]

Regarding this parameter, operators have no right to modify. Manufacturer's parameter provides N4.1 (lubrication), N4.2 (red light when finished), N7.5 (measure) and N7.9 (other operation), etc.

## [All Parameters]

Under current permission, all the parameters and their classification are listed for you to view. You can enter different sub-screens to set parameters according to your level of permission. The relationship between permission and parameters available are shown as below. To look over and modify a certain parameter, you need the permission which has the corresponding right.

Permission	Parameters Available
Operator	operator's parameter
Manufacturer	operator's parameter, manufacturer's parameter
Developer	operator's parameter, manufacturer's parameter, developer's parameter

Table 1 Permission and Parameters Available

Under the screen are the manipulation buttons: [operator], [manufacturer], and [developer] buttons with shortcut keys F1, F2, and F3 respectively. Click them to enter the corresponding [general parameter] sub-screen. Please refer to Chapter 9 for detailed parameter list form.

#### Parameters Backup Function Screen

As Fig. 5-50, you can save or reset the parameters to avoid setting parameters repeatedly. After modifying the parameter, enter this screen, and then press down the "save" button, the system ejecting the inputting dialog box to prompt you for naming the saved item. NcStudio supplies 8 saving items, and you can load the parameter by clicking [Load] button on the right according to saved item name on the left.



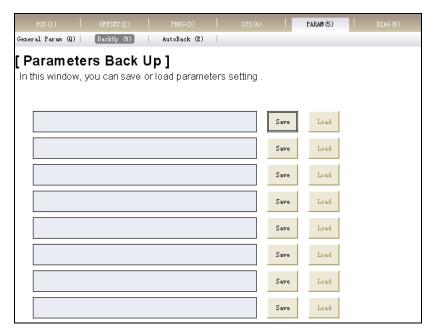


Fig. 5-50 Parameters Backup Function Screen

## ◆ Auto Backup Function Screen

As Fig. 5-51, NcStudio system supports auto backup of parameters. So if you forget to back up parameters, enter this screen including parameter backup settings from ex-factory date to the last saved date and click [load] on the right to recover a parameter setting according to actual needs.

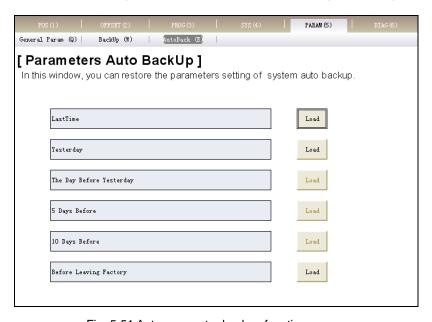


Fig. 5-51 Auto parameter backup function screen

## 5.2.6. Diagnosis Function Section

Identical with that under other four modes, the function of [Diagnosis] section mainly includes log function, polarity modification and port test of hardware, PLC simulation test function, etc.

[Diagnosis] section includes function windows of [Log], [Hardware port], [PLC], and [Others],



which will be introduced separately.

## ◆ [Log] Function Screen

Recording the important operations and system events, this screen provides both current log information to browse and history log information to review, as Fig. 5-52.

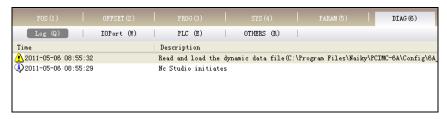


Fig. 5-52 Log function screen

## ◆ [I/O Port] Function Screen

The screen is shown as Fig. 5-53. Used for displaying information about control card and system current input/ output map, this function is very helpful for system supervision and fault diagnoses.

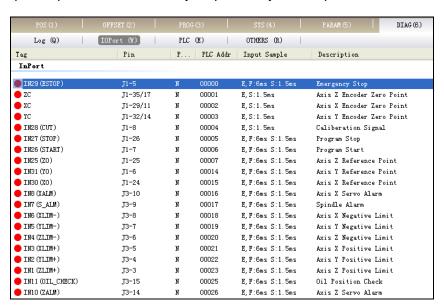


Fig. 5-53 Hardware port function screen

## [PLC] Function Screen

As Fig. 5-54, this screen is used for PLC debugging, which is generally used by system testers and advanced users. For details, please refer to Programming Manual.



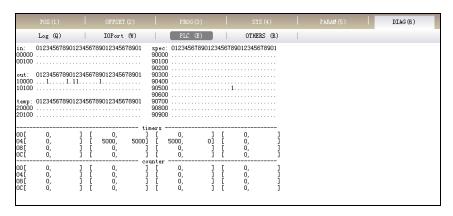


Fig. 5-54 PLC function screen dialogue

#### Other Pictures

This function is used for feeding back port data, shown in Fig. 5-55.



Fig. 5-55 Other function picture

#### Manipulation button bar

The manipulation buttons of [Diagnosis] vary under different sub-functions.

The manipulation buttons of [Log] are shown in Fig. 5-56:

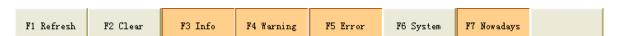


Fig. 5-56 The manipulation buttons of [Log] function screen

#### [Refresh]

Shortcut key: F1, only available in the [Log] function screen of [DIAG] section; pressing down the button will refresh the log list and synchronize the log list with the system.

#### [Clear]

Shortcut key: F2, only available in the [Log] function screen of [DIAG] section; pressing down the button will clear all the log information.

#### [Info], [Warning], [Error], [System], [Recent]

Shortcut key successively: F3, F4, F5, F6, and F7, only available in the [Log] function screen of [DIAG] section; under default situation, these buttons are pressed down, with related information displayed, such as [Normal], [Warning], [Error], [System] and [Current]. If you don't need certain



information, you just need to press that button again to make it pop up. If [Recent] pops up, the system will display all logs since first recorded date.

The manipulation buttons of [IO Port] are shown in Fig. 5-57:

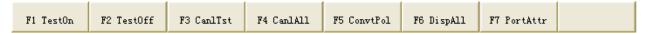
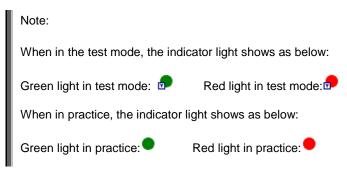


Fig. 5-57 The manipulation button of [IO Port] function screen

## [Test on], [Test off]

Shortcut key in turn: F1, F2, only available in [IO Port] function screen. It is used for simulating signals of hardware, and for simulation test. After the two buttons ([Test on], [Test off]) are pressed, indicator light before the selected port will be shifted between green and red. Green light means the port test starts while red light indicates the opposite action.



#### [Cancel Test]

Shortcut key: F3, only available in [IO Port] function screen. It is used to cancel the simulation test and analog signal. And the real hardware signal will replace the simulating signal.

#### [Cancel All]

Shortcut key: F4, only available in [IO Port] function screen. [CanlAll] is similar to [CanlTst] button, but [CanlTst] button can only cancel one single port's simulation while [CanlAll] button can stop all ports' simulation and replace them with actual hardware signals.

#### [Modify Polarity]

Shortcut key: F5, only available in [IO Port] function screen. You can decide whether the polarity of selected port is N (NO) or P (NC).

## [Show All]

Shortcut key: F6, only available in [IO Port] function screen. The hidden ports will be displayed after this button is pressed down. Pressing the button again will conceal all the vacant ports, with the buttons raised.



## [Port Attribute]

Shortcut key: F7, only available in [IO Port] function screen. Pressing down this button will display the details of port and modify the relative port information as Fig. 5-58. You can set "port sampling time", "filter or not" and "port enabled" function, and modify port name and description.

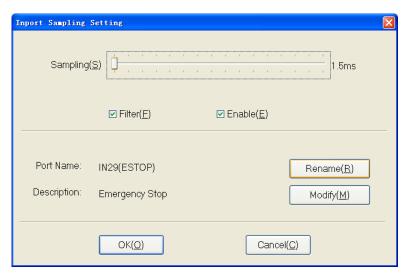


Fig. 5-58 Input ports sampling setting dialogue



## 6. Manual Mode of NcStudio

In manual mode, you can control the machine tool by manual operation facilities, such as manual pulse generator, manual operation panel. The manual mode can be subdivided into jog, stepping (also called incremental) and handwheel and so on.

Manual mode is shown in Fig. 6-1. Similar to auto mode, manual mode also includes one operation panel and one function panel that will be introduced in detail.

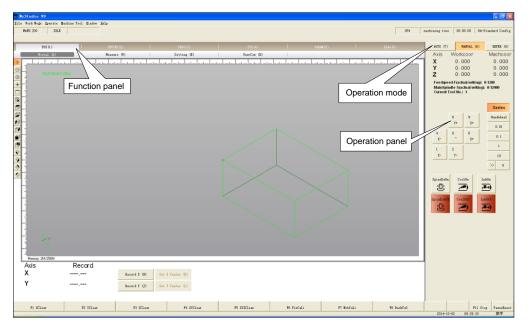


Fig. 6-1 Manual operation mode screen

# 6.1. Operation Panel Section

As Fig. 6-2, on manual operation panel, there are many buttons for controlling the machine manually.



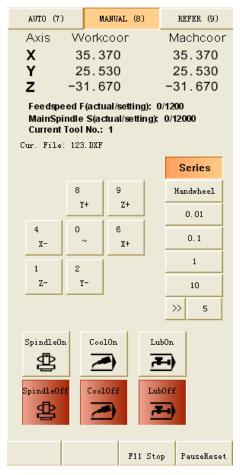


Fig. 6-2 Manual operation panel

On the top of the panel, there is a coordinate display zone, which prompts the current coordinates of the tool, spindle speed, machining cutter No., current feed speed and the name of the current file.

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

> [X+], [X-]manual buttons, shown as:

The buttons respectively correspond to X positive and X negative. Hotkey: number 6 and 4 in mini-keyboard.

> [Y+], [Y-] manual button, shown as:

The buttons respectively correspond to Y positive and Y negative. Hotkey: number 8 and 2 in



mini-keyboard.

The buttons respectively correspond to Z positive and Z negative. Hotkey: number 9 and 1 in mini-keyboard.

> [~] manual button, shown as:

Acceleration button, corresponding to number key "0" in mini-keyboard; it is used together with X, Y, Z direction keys for speeding up cutter, only available under "Series" state. Combination key: "0" + "direction key", for instance, "0" + "8" (accelerating along +Y direction).

➤ [Series], [Handwheel], [0.01], [0.1], [1], [10], [>>] manual buttons, see Fig. 6-3.

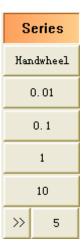


Fig. 6-3 Manual moving buttons

There are totally three manners to manipulate the machine under manual mode. They are series manner, handwheel manner (MPG), and jog manner, which will be introduced separately.

Series (JOG) manner: pressing down the button, you can manually make the machine tool move continuously. When the manual screen is the current active window, clicking the direction key on the panel, or pressing down the corresponding shortcut key will make the machine move continuously. Loosening it will stop the moving.

#### Note:

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

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

Handwheel (MPG): this button is added to support handwheel (MPG—Manual Pulse Generator).



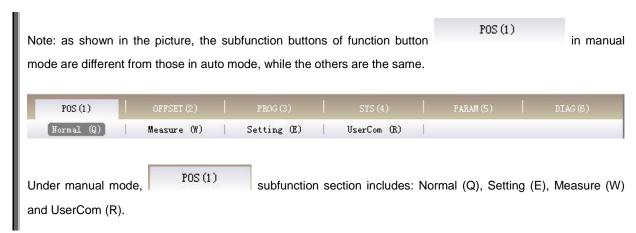
After installing a handwheel and pressing this button under manual mode, you can manipulate the machine tool by handwheel. At this time, the X, Y, Z direction keys are invalid.

Stepping manner (0.01, 0.1, 1, 10, >>): also called step-manner or increment-manner. Different from jog manner, stepping-manner can control the machine's feed distance more accurately. Every time the direction key is triggered, the machine will move a distance of a specified step-size. After one button of [0.01], [0.1], [1], [10] is pressed, stepping manner is activated. And increment stepping length equals to selected step-length. For instance, if [0.1] button is selected, increment stepping length will be 0.1 mm. Moreover, V9 NcStudio also supports custom step-size. Pressing down [>>] button will eject an "Input Box" dialogue. Input a step-size that you want, and then click [OK], the value of the custom step-size displayed behind [>>] button. For example, press the [>>] button and input "500.0" into the "Input Box", and then click [OK], "500" to be displayed behind [>>] button. When [500] button is pressed, every time triggering one of X and Y manual buttons will make the machine tool move 500 mm accordingly.

## [Stop], [Reset]

The two functions have already been introduced under auto mode, which will be omitted here.

## 6.2. Function Panel Section



## Normal Function Screen

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



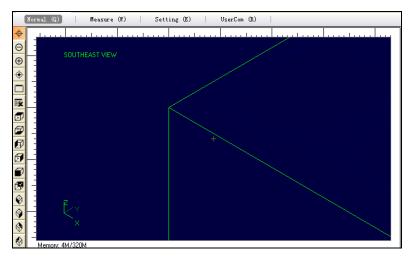
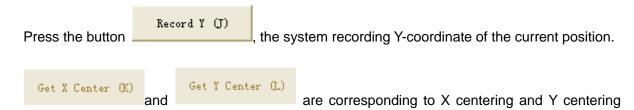


Fig. 6-4 Normal function screen under Manual mode

## [Record X]

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

## [Record Y]



functions.

#### ♦ Measurement Screen

"Measurement" screen under manual mode is as following, see Fig. 6-5.

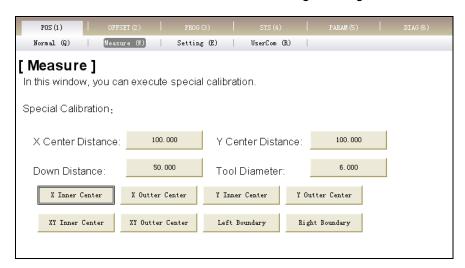


Fig. 6-5 Measurement screen under manual mode

Measurement screen is used for tool measurement, including normal measurement and special



measurement. Normal measurement can be divided to fixed measurement and mobile measurement (illustrated in manipulation button bar); special measurement includes 8 special measurements of X inner / outer center, Y inner / outer center, XY inner / outer center, left / right boundary.

Special measurement is used for moving cutter to workpiece center. And inner center measurement is for cavity shape workpiece center. Before center measurement, the tool nose must be in the predicted center position, and the value of center distance, down distance and tool diameter must be specified. In inner center measurement, the tool down distance should be less than distance from tool nose to workpiece surface, while larger in outer center measurement. Besides, you can set operation parameters according to the buttons in picture.

- X center distance: the pre-estimated distance from workpiece midpoint to X boundary, depending on concrete situation.
- Y center distance: the pre-estimated distance from workpiece midpoint to Y boundary, depending on concrete situation.
- Tool down distance: tool down / up distance
- Tool diameter: tool's actual diameter.
- 1) X inner center measurement is shown in Fig. 6-6. (Its operation method is the same as that of inner measurement of Y center and XY center)

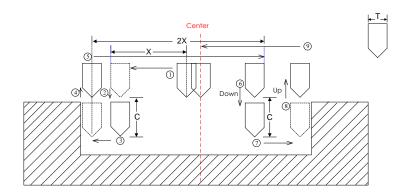


Fig. 6-6 Measurement process of X inner center

2) X outer center measurement is shown in Fig. 6-7. (Its operation method is the same as that of outer measurement of Y center and XY center).



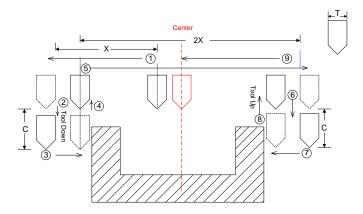


Fig. 6-7 Measurement process of X outer center

## 3) Boundary measurement

Boundary measurement includes left / right boundary measurement, used to set boundary as workpiece origin, shown in Fig. 6-8.

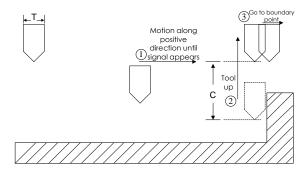


Fig. 6-8 Measurement process of +X boundary

## **♦** Setting Function Screen

Setting function screen under manual mode is shown in Fig. 6-9. In this screen, you can set the common parameters under manual mode, such as manual low speed, manual high speed, etc. Manual low speed refers to the moving speed when you don't press the manual high speed key, within range between 1 mm/min and Manual high speed. Manual high speed refers to the moving speed when you press the manual high speed key, within range between 1 mm/min and G00 speed.



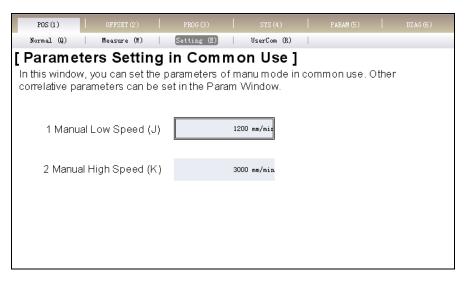


Fig. 6-9 Setting function screen of POS function section under Manual mode

## ◆ Manual Data Input

Manual Data Input screen is shown in Fig. 6-10. In this screen, you can input G instructions manually & repeatedly and execute them. The screen supplies ten instruction input items. If no instruction is input, the [Execute] button is disabled. Click [Edit] button to eject a dialog "Input Box", input instruction into the dialog, click [Ok], and finally press the activated [Execute] button to execute this instruction. Moreover, the instructions input will be remembered in the screen, which provides you with a very convenient way to look over the instructions.



Fig. 6-10 User command function screen under Manual mode

## Manipulation Button Bar

Under MANUAL mode, manipulation button columns of each subfunction screen under POS



function section are the same, shown in Fig. 6-11.



Fig. 6-11 Manipulation button column of Manual mode

## [FixCali]

Fixed measurement is used to revise Z coordinate of workpiece origin after tool change.

During tool measurement, at first, ensure workpiece surface artificially, set workpiece surface as

Z coordinate of workpiece origin, then click button

F6 FixCali the syste

the system automatically recording

current Z workpiece coordinate. In case of tool change or tool break, clicking button again will resume previous Z coordinate of workpiece origin.

F6 FixCali

#### Attention:

Fixed measurement will be invalid after NcStudio is closed. After restart of NcStudio system, you need to reset fixed measurement.

Operate as the following prompt:

Click F6 FixCali for the first time:

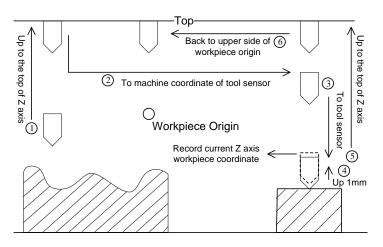


Fig. 6-12 First measurement

Measurement after tool change or tool break, click again

F6 FixCali



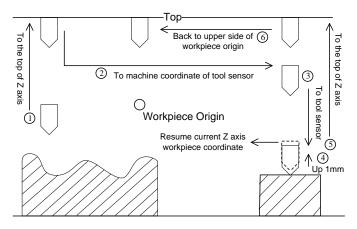


Fig. 6-13 Measurement after tool change

## [MobCali]

Press the button F7 MobCali, the system executing mobile measurement.

With mobile measurement, it is more convenient for you to ensure the workpiece origin of Z-axis. Put the tool presetter on workpiece surface, manipulate the tool to the top of Z-axis to touch the tool presetter, press [begin measurement] button. Then tool measurement will be executed automatically to confirm Z coordinate of workpiece origin. You can operate according to prompts.

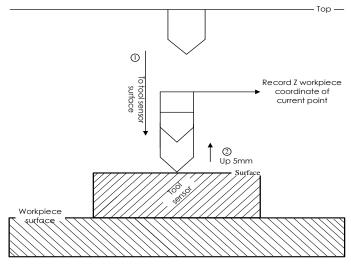


Fig. 6-14 Mobile measurement

## [Back To0]

[Back to0] is the same as that under auto mode, which will not be introduced any more.



# 7. Reference Point Mode of NcStudio

[Back to the Reference Point] mode is the process to unify the internal coordinates and the actual coordinates. Therefore, during system start-up, it is essential to return to the reference point. After the system is booted each time, the default operation mode is the reference point mode, which is convenient for you to perform the operation of "back to machine origin".

The software picture under reference point is as Fig. 7-1.

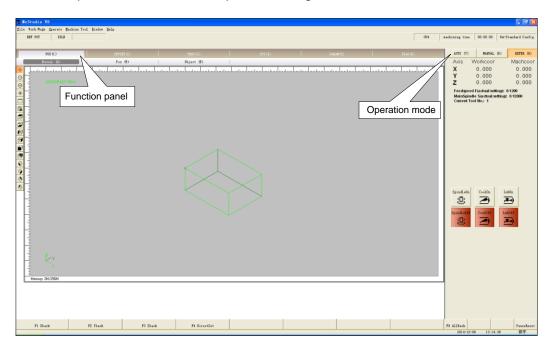


Fig. 7-1 Reference operation panel dialogue

The operation panel of reference point displays the operation buttons related with machine origin point, supporting an interactive interface for manipulating the machine to return to reference point manually. Its function selection buttons and sub-function switch buttons are the same as those in manual and auto operation panels.

## 7.1. Operation Panel Section

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





Fig. 7-2 Operation panel dialogue

The operation panel of reference point displays the operation buttons related with machine origin point, supporting an interactive interface for manipulating machine to back to the reference point manually.

## ◆ [All Back]



As the MCS origin, machine origin (the reference point) is a fixed position on the machine, positioned by both mechanical switch and electrical system. Regarding the MCS, please refer to chapter 3.2.1. Operation of backing to the reference point requires the support from machine hardware, so some types of machine don't have this function. As the benchmark of the whole machine, backing to reference point calibrates the coordinates of the current point.

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

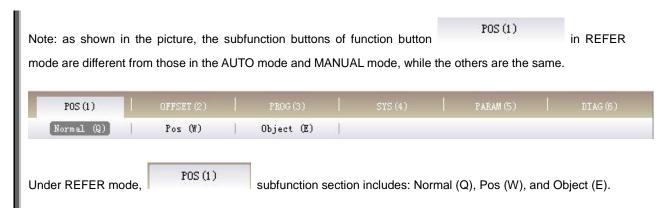
[All Back] button calls both X-axis and Y-axis simultaneously to back to the reference point. V9 NcStudio also supports single-axis back and direct setting. Please refer to chapter 7.2 for details.



## ◆ [Reset]

This button has already been introduced in AUTO mode and no more details here.

## 7.2. Function Panel Section



#### **♦** Normal Function Screen

The normal function screen is the same as those in auto and manual modes except for manipulation button bar in the lower part. Therefore, this function screen will not be introduced here. Picture is shown in Fig. 7-3:

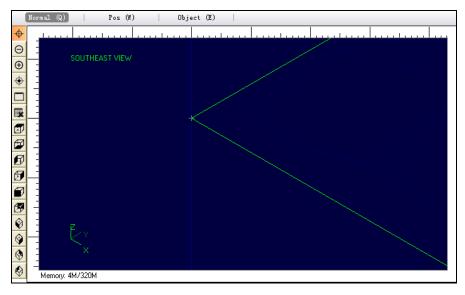


Fig. 7-3 Normal function screen

#### **♦** Position Function Screen

The POS function under REFER mode is the same as that under auto mode except for manipulation button bar in the lower part. This is convenient for you to look for the position of cutter under these two modes, including machine coordinate, workpiece coordinate and current workpiece coordinate and remaining distance. The picture will not be listed any more.



## **♦** Object Section

The upper part of object section is the same as that of normal section, except for the missing of program displaying section, so the picture of this section will be omitted

## ♦ Manipulation Button Bar

The manipulation buttons in different function screens under this mode are all the same, shown in Fig. 7-4:

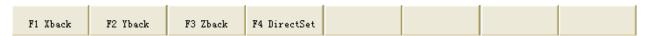


Fig. 7-4 Manipulation button bar under REFER mode

## [X Back]

Hotkey: F1, it is only available under REFER mode. Press the button to drive the tool back to the machine origin along X-axis. When the operation finished, there will be a mark ( ) of completing backing to the machine origin behind [X-axis] in operation panel.

## [Y Back]

Hotkey: F2, only available under REFER mode. Just like [X Back], Press the button to drive the tool to go back to the machine origin along Y-axis. When the operation is finished, there will be a mark ( ) of completing backing to the machine origin behind [Y-axis] in operation panel.

#### > [Z Back]

Hotkey: F3, only available under REFER mode. Just like [X Back], press the button to drive the tool to go back to the machine origin along Z-axis. When the operation is finished, there will be a mark ( ) of completing backing to the machine origin behind [Z-axis] in operation panel.

## [DirectSet / Direct Setting]

Hotkey: F4, it is only available under REFER mode. If you can make sure the current position is consistent with its machine coordinate without abnormal state, such as "E-stop", you can click the button to directly set the current machine coordinate as the accurate machine coordinate. When the operation finished, there will be a mark ( ) of completing backing to mechanical origin before [X-axis], [Y-axis] and [Z-axis].



# 8. Menu System of NcStudio

Menu operation, along with keyboard operation and shortcut key operation, plays an important role in the entire operation framework of NcStudio. NcStudio furnishes you with the abundant menu commands that will impress you a lot in this chapter.

## 8.1. File Menu

[File] menu includes the operation commands related to processing files, shown in Fig. 8-1:

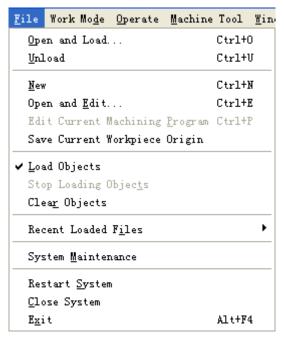


Fig. 8-1 File menu dialogue

#### [Open and load]

Shortcut key: Ctrl + O. The menu is used for opening the processing file saved in hard disk and loading it into the decoder of the system. Then the parse engine will parse the file into G instructions, and transfer them to NC control card. Choose the menu item, the system popping up the "Open and Load" dialog box, shown in Fig. 8-2. After the processing file is loaded, the title bar and the mini-coordinate display zoom will display the name of loaded processing file.

Once a processing file is loaded into NcStudio system, it can not be modified (but it can be read). So if you want to modify the loaded processing file, you must unload it first in NcStudio system.



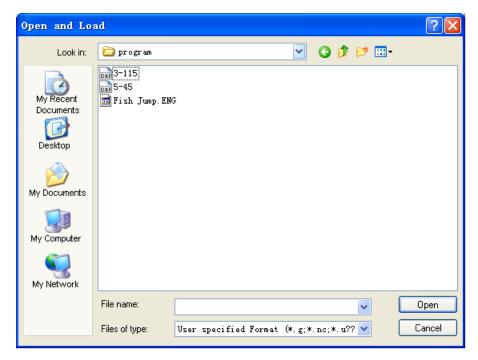


Fig. 8-2 Open and load dialogue

## Note:

When the menu item with symbol "..." behind is chosen, the system will eject a dialog box or an input box.

## ➤ [Unload]

This menu item is the same as the "unload" button in [Pos] function section under AUTO mode. After loading a processing file in the system, you can choose "Unload" to close the currently loaded file. Then the parse engine buffer will release the loaded file.

#### Note:

If the loaded file is being processed, the file loaded in parse engine buffer zone can't be closed.

## > [New]

You can create a processing file and edit it instantaneously. Choose this menu item; the system will create a new file and switch to editing window for code editing. Besides, the file can be saved instantaneously in editing window.

## [Open and edit]

This menu item is used for opening a processing file saved in disk and loading it into editing window. You can edit the processing file in it. Clicking the menu option will eject an [open and edit] dialogue box. You can operate it according to the prompt of the dialogue box.



## [Edit current machine program]

This function can load the procedure file that has already been in parse engine buffer into the editing window. This menu item is valid only when there are procedure files in parse engine buffer.

## [Save current workpiece origin]

Write current workpiece coordinate origin to the end-of-file. When it is loaded again, the system will automatically check whether there is a saved workpiece coordinate origin, and prompt you whether to use the existing one as current workpiece coordinate origin. If you select "Yes", the saved coordinate origin in the file will be current workpiece coordinate origin; selecting "No" will remain the current workpiece coordinate. However, if there is no saved origin, the system will load the file normally without any further dealing.

During write-in, if there has already been a saved workpiece coordinate origin, the previous one will be overwritten. But if there is no saved one, it will be written directly to the end-of-file.

Note:

It is not recommended to edit files with coordinate origin saved, including editing using NcStudio and other tools.

## [Load objects]

This menu item is used to parse the machining file and load the whole entire track with one shot. In actual machining process, [Pos] function screen will also display the simulated processing trace, but it proceeds stepwise in accordance with machining process. The two kinds of the tracks can be distinguished by color; selecting [Load Objects] will load the track in advance, and its opposite option is [Clear Objects].

## [Stop loading objects]

This item is used for suspending loading processing track into system. The excessively large track of the loaded file results in system stagnation. If so, you can choose [Stop Loading Tracks] to terminate the track loading or choose [Clear Tracks] to stop track loading and clear the loaded track.

#### [Clear objects]

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

## [Recent loaded files]

This item will open a new submenu, which can display the recently loaded processing files. If you need to load these files, you can click the file name directly.



## [System maintenance]

This item is mainly used for making a backup, recovering, installing and repackaging, etc. Selecting this menu item will close the NcStudio for a while and eject an option selecting dialogue box, shown in Fig. 8-3. And the dialogue includes 4 items: [key data backup], [Key data resume], [system installation], and [generate installation packet].



Fig. 8-3 System maintenance dialogue

- Under "Backup Key Data" option card, you can back up key files in default directory, including controlling parameters, sub-program of common-G code, screw error data, PLC program, etc. If there are backup files under the default directory, they will be displayed in the window below. You can delete or preserve them, and also check the backup file property in the window below by clicking the file name. The backup button is on the lower part of panel. You can select whether to save the files under the default directory. If not, you can appoint another directory for files backup.
- 2) Under the "Key data resume" option card, you can resume the key files according to the backup. If there are backup files under the default directory, they will be displayed in the window below.

  Clicking the file in the window will display its attribute below the window. You can execute [delete]



and [start resume] function. You can choose the other backup files by direction key. If the former backup file is saved in other position, you can click [Browse] to search for it. Click the "Start Resume" button to resume the system automatically.

- 3) Under the "System installation" option card, you can install NcStudio system again. But the installation will delete all former files under the installation directory and the new version won't keep any data of the old version. Therefore it's recommended to do backup before the installation. If there are installation packages under the default directory, they will be displayed in the window below. Clicking the package in the window will display its attribute below the window. You can execute [delete] and [start installation] or select other package by direction key. Click the "Start Installation" button to start setup automatically
- 4) Under the "Generate installation packet" option card, you can create a new installation package on the basis of current system data, which is useful to save a system of stable version. If there are packages under the default directory, they will be displayed in the window below. Clicking the "generate to default path…" below will create the package under the default path. After the package in the window is clicked, its attribute will be displayed below the window. You can execute [delete] and [start installation] or check the attribute of other packages by direction key. Click the "generate to appointed path" to create the package under the defined path and then click the "Generate installation packet" to start generation automatically.
- [Restart system]

Exit and restart the system.

[Close system]

Exit and close current system.

➤ [Exit]

The menu item is used for closing NcStudio system. If you do not save the processing file under edit, the system will prompt you to save it. You can operate as the prompts.

## 8.2. Work Mode Menu

[Work Mode] menu includes the related operation commands of mode selection, as Fig. 8-4:



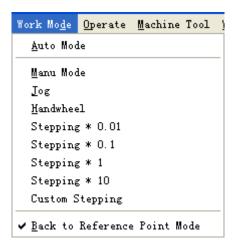


Fig. 8-4 Mode menu dialogue

## [Auto Mode]

It is corresponding to the AUTO button on the operation panel. Selecting the menu item will shift the mode to AUTO. Shortcut key: Ctrl + F1.

## [Manu Mode]

It is corresponding to the Manual button on the operation panel. Selecting the menu item will shift the mode to Manual mode. Shortcut key: Ctrl + F2.

#### [Jog]

It is corresponding to the Series (Jog) button on the operating panel. Selecting the menu item will appoint the series (Jog) as current manual manner.

## [Handwheel]

It is corresponding to the Handwheel button on the operating panel. Selecting the menu item will appoint handwheel as the current manual manner.

## [Stepping\* 0.01], [Stepping\*0.1], [Stepping\*1], [Stepping\*10]

They are corresponding to the stepping length selection buttons on the operating panel under manual mode. Selecting this item will appoint stepping as the current manual manner, and you can select the relative stepping length.

## [Custom stepping]

It is corresponding to button on manual operation panel. Selecting the menu item will enable you to customize the stepping length.



## [Back to reference point mode]

It is corresponding to the REFER button on the operating panel. Selecting the menu item will shift the mode to REFER mode. Shortcut key: Ctrl + F3.

## 8.3. Operate Menu

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

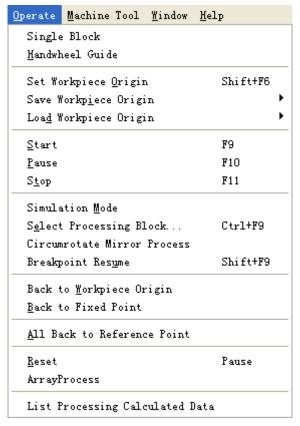


Fig. 8-5 Operation menu dialogue

## [Single block]

The menu item is only available under AUTO mode.

You can set the processing assignment to be executed in single step mode, which provides a good support for mistake diagnosing and malfunction restoring. Once in the single block mode, when resultant velocity of each axis is zero each time, the system will stop.

## [Handwheel guide]

If you need to use handwheel in machining, you can connect handwheel first before selecting this option. And then the system will execute machining by handwheel.



## > [Set workpiece origin]

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

## [Save workpiece origin]

This function is used for saving the current workpiece origin into the processing system file. You can save the commonly used workpiece origin as pre-set value and the system can at most save 10 groups of coordinate data. If you want to rapidly position to the workpiece origin set at the first time in the future work, you can use this function to save it.

## [Load workpiece origin]

This function is used for loading the preset coordinate value of workpiece origin and rapidly returning to it. After loading the workpiece origin, you can use [back to workpiece origin] command to go back to the preset workpiece origin.

## [Start] [Pause] [Stop]

They are corresponding to the [Start], [Pause], and [Stop] buttons on the operating panel in the auto mode. This menu option is only valid in the auto mode, and can be used for the simulation function.

### [Simulation mode]

The menu option is corresponding to [simu] button of [POS] function section under the [auto] mode. If the processing procedure has been loaded, and the current operating mode is "IDLE", pressing down this button will carry out simulating at high speed automatically from the first section. Simulation function is similar to the demonstration function, but superior. Simulation function provides you with a fast and vivid simulating machining environment.

Running under the mode of simulating, the system will not drive the machine tool to do the relative actions but show the processing trace of the cutter at high speed in the trace window. Through simulation, you can view the machine tool's movement path in advance to avoid equipment damage caused by programming errors and learn other information. Once simulation starts, this item will change into "Stop simulation and withdraw the simulation mode". Executing this item will suspend simulation immediately.



## [Select Processing Block]

This item corresponds to button [SelMachining] of [POS] function section under [AUTO] mode, only available under AUTO mode. The shortcut key is Ctrl + F9. It's used for advanced start and simulation. After the menu item is selected, the system will eject an "execute (advanced options)" dialogue, as Fig. 8-6. You can handle it following the prompt.

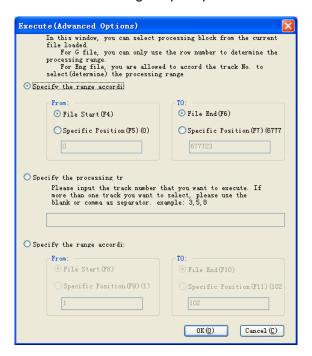


Fig. 8-6 Executing (advanced options) dialogue box

#### [Breakpoint resume]

The item is corresponding to the "B.P. Resume" button of [POS] function section under [AUTO] mode, only available under AUTO mode. The shortcut key is Shift + F9. After this item is selected, the system will execute the processing file from the breakpoint of last time. It can also be used in simulation.

#### [Circumrotate Mirror Process]

This function is used for executing rotation and mirror machining centered on workpiece origin. And selecting this item will eject the following dialogue, as Fig. 8-7.



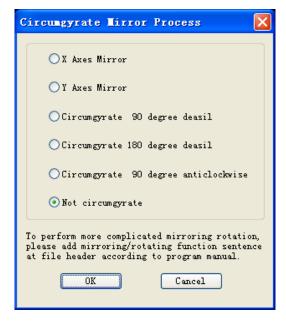


Fig. 8-7 Mirror and rotation machining dialogue

## [Back to Workpiece Origin]

Workpiece origin is the origin of workpiece coordinate and the origin of workpiece dimension programming; generally speaking, it is a start-point of workpiece machining. Programmer can set workpiece origin freely. The position relation between workpiece origin and reference point can be fixed by [Set Workpiece Origin] item.

This item selected, tool nose will return to the workpiece origin automatically from current position.

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

## [Back to Fixed Point]

You can set a fixed point in advance. In case of tool damage or tool change, selecting this item will uplift the cutter and move it to the fixed point in order to change tool.

## [All Back to Reference Point]

This item is used to call all axes back to reference point. It is strongly suggested to use this function before machining or when E-stop occurs.

#### > [Reset]

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



## [Array Process]

This function is used for array processing in the same machining program. After you select this item, an "Array process" dialog will pop up, as Fig. 8-8.

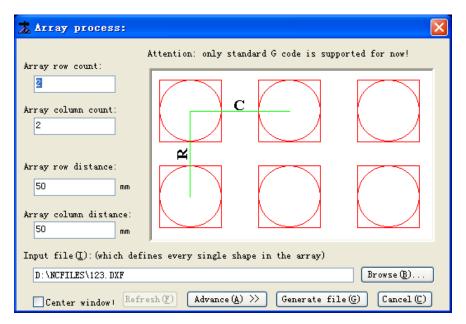


Fig. 8-8 Array processing

Click button Browse (B)..., and select the program for array processing.

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

If you need to set different array row distances or column distances, click button "Advance", a dialog as Fig. 8-9 popping up.



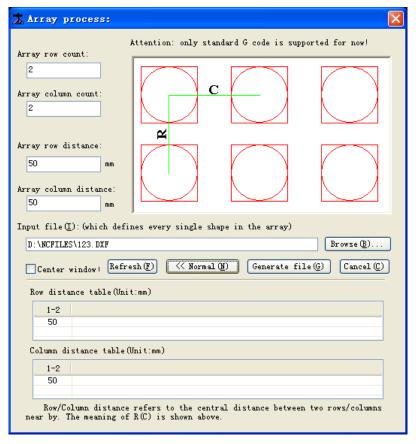


Fig. 8-9 Array process (advanced mode)

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

## > [List Processing Calculated Data]

In machining, selecting this item will display the calculated info. of the machining file.

## 8.4. Machine Tool Menu

[Machine Tool] menu includes the relative menu options of spindle start/stop, lubrication start/stop, and coolant start/stop. It can implement the start and stop functions for spindle, lubrication valve, and cooling valve, shown in Fig. 8-10.



Fig. 8-10 Machine tool menu



## [Main spindle]

This menu option can set the startup of spindle.

## [Lubricate]

This menu option can set the startup or stop of lubrication pump.

## [Coolant]

This menu option can set the startup or stop of coolant pump.

## [Light]

This menu is used to control the startup of lighting.

## 8.5. Window Menu

Window menu is used for interface navigation, which marks all visible windows under current mode. You can select an item to enter the desired window. Under different modes, the menu will vary, as Fig. 8-11.

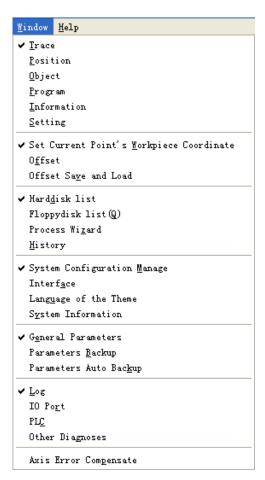


Fig. 8-11 Window menu dialogue



## 8.6. Help Menu

The menu is used for displaying system info. and registration info., shown in Fig. 8-12 below.

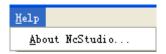


Fig. 8-12 Help menu dialogue

## [About NcStudio]

Selecting the menu item will eject a [About NcStudio] dialog box, as Fig. 8-13.

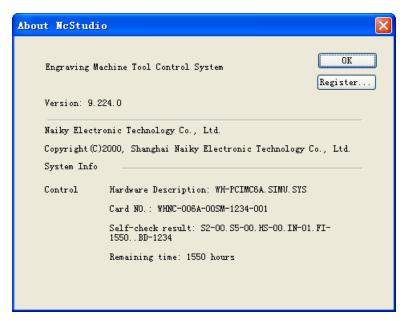


Fig. 8-13 "About NcStudio" dialogue

You can look up version, control card and manufacturer information in this dialogue. Clicking the "Register..." button under [Ok] button of the dialogue box will eject a register dialog box; you can accomplish the function after entering the register code, shown in Fig. 8-14 below.



Fig. 8-14 REGISTER dialogue



## 9. Parameters of NcStudio

With abundant parameters, NcStudio is competent for various processing tasks. This chapter will specialize in introducing the parameters involved in NcStudio system. According to different levels of permission, general parameters of NcStudio system can be subdivided into operator parameters, manufacturer parameters and developer parameters. To look over or modify some types of parameters requires equivalent permission.

## 9.1. Methods of Parameters Modification

Use mouse, or press 'Tab/ Shift + Tab' to move the cursor onto the parameter list box, and press the 'direction key' (up and down arrow key) to choose the parameter that you want to modify, at last press 'enter key' and input value into the parameters input box; or you can also double click the parameter line, and then input value into the parameters input box.

#### Note:

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

In the parameter list box, only part of the parameters can be displayed. The following two methods will teach you how to look over the complete content of the parameters: look at the description window of the parameters information under the low part of the parameter interface (recommended). This window will show you the complete information about the parameter you choose. Another method is to press down the 'direction key' (left or right direction key) in the parameter list box and look over the whole information through the scroll bar.

All modification to the parameters must be done under the "idle" state of system.

## 9.2. Manufacturer (Including Operator) Parameter List

Please look at operator parameters first, shown in Fig. 9-1.



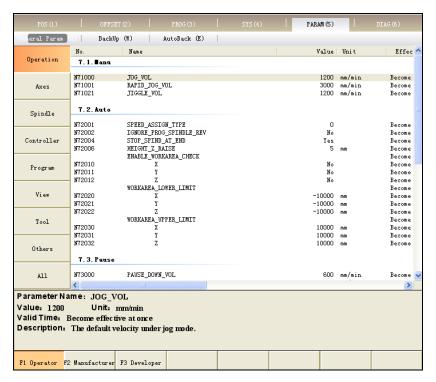


Fig. 9-1 Operator parameters

Manufacturer parameters include collection of operator parameters and some factory-setting parameters, shown in Fig. 9-2:

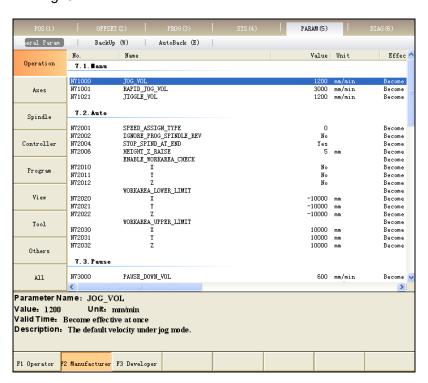


Fig. 9-2 Manufacturer parameters

The parameters in V9 NcStudio adopt 5-bit decimal numerical coding. Bit 1 is the main category (1~9), and bit 2 is the subcategory (0~9). Each subcategory has 1000 numbers (000~999). The type names are offered according to the numbers of bit 1 and bit 2.



In the following, the entire manufacturer parameters are listed according to the code numbers of the parameters. You can look up the parameters via their code numbers.

No.	Name	Setting Range	Default	When Enabled		
1.0 Axes	1.0 Axes General					
N1000	AXIS-DIR	positive direction     regative direction	-1	After restart		
N1001 N1002		remental direction of machine comment, and "-1" indicates negative		specified axis: "1"		
	AXIS_PULSE_FACT OR(X\Y\Z)	0.0000009~999.0	0.001	After restart		
N10010 N10011 N10012	Pulse equivalent is the displacement or angle produced on the corresponding axis by each control pulse. And it refers to the least displacement which the motion control card can deal with in detail. In stepping system, it is corresponding to a stepping pulse, then according to transmission connection; the angle displacement of the stepping pulse will be converted into linear value.					
N10020 N10021	WORKBENCH_LOW ER_LIMIT(X\Y\Z)	-99999.0~99999.0	X: 0.0 Y: 0.0 Z: -350.0	After restart		
N10022	It sets the machine coordinate of the allowable lower limit of the worktable stroke wher [check_ workbench_ range] is valid.					
N10030 N10031	WORKBENCH_UPP ER_LIMIT(X\Y\Z)	WORKBENCH_LOWER_LIMI T (Machine Coor.)~99999.0	X: 800.0 Y: 600.0 Z: 0.0	After restart		
N10032	It sets the machine coordinate of the allowable upper limit of the worktable stroke when [check_ workbench_ range] is valid.					
N10040 N10041	CHECK_WORKBEN CH_RANGE(X\Y\Z)	0 (False): Disabled 1 (True): Enabled	1 (True)	After restart		
N10042	It sets whether to check	k the worktable stroke.				
1.1 Comp	pensations		T			
N12000	ENABLE_LEADSCR EW_CMPN	0 (False): Disabled 1 (True): Enabled	1 (True)	After restart		
1412000	It specifies whether screw error compensation is valid, including backlash and screw pitch compensation.					
N12001	ONLY_BACKLASH_ CMPN	0 (False): Disabled 1 (True): Enabled	1 (True)	After restart		
	It specifies whether to execute error compensation only based on screw backlash.					
N12010	BACKLASH(X\Y\Z)	0.0~1.0	0.0	After restart		
N12011 N12012	It specifies the screw backlash of each axis. It is only effective when parameter "N12001" is set as true.					
N12020	ENABLE_QE_CMPN	0 (False): Disabled 1 (True): Enabled	0 (False)	After restart		



No.	Name	Setting Range	Default	When Enabled		
	It specifies whether arc across quadrant error compensation is valid.  QE_DATA (Group 0\ Group 1\ Group 2\ Group 3\ Group 4\ Group 5)  QE_DATA is compensation parameter for removing spikes when an arc across quadrant.  Settings of the parameters are the same as those in X-axis, Y-axis and Z-axis.					
N12030	TIME (QE_DATA)	0.0~10.0	0.0	After restart		
N12080		is, the larger the range influence				
etc	00	second is recommended.	od by the comp	oriodaeri wiii be.		
	DISTANCE (QE_DATA)	0.0~10.0	0.0	After restart		
N12031 N12081 etc	The larger the value is, the more obvious the effect of compensation is. But note that if the value is too large, the curve will be sinking inward, and if too small, it won't reduce the height of spikes effectively. It's recommended to measure the actual height of sharp-angle by the laser interferometer in debugging the machine. The parameter is usually set as 0.3~3 times of the sharp-angle height (The effect is related to compensation time and compensation intension).					
	DELAY (QE_DATA)	0.0~10.0	0.0	After restart		
N12032	Owing to the different	mechanical properties of differe	nt machines, the	e position of the		
N12082	spikes may not just be	spikes may not just be at the quadrant conversion point, but some distance away from the				
etc	quadrant conversion p	oint. Please estimate the related	time of this dista	nce and set it as		
	this parameter value.					
N12033	INTENSITY(QE_DAT A)	0.0~1.0	0.75	After restart		
N12083 etc		The value of the parameter affects the compensation effect. The larger the value is, the				
		t of compensation will be.				
	Acc Limits			T		
N13000 N13001	AX_MAX_VOL (X/Y/Z)	0.001~100000.0	48000.0	Immediately		
N13002	It specifies the maximu	m speed of each axis.				
1.4. Rota	tive Axes(Rotary Table)					
N14002	ROT_AXIS_WP_DIA	1.0~3000.0	10.0	Immediately		
1114002	It is the compiling diam	eter of the workpiece to be proces	ssed under the tu	ırntable mode.		
	ROT_AXIS_MAX_VE LO	0.001~9999.0	600.0	Immediately		
N14003	It specifies the max. rotary speed of rotary axis. In any case, the rotate speed of rotation axis should never be above the maximum speed of the specified rotation axis. This value is determined by the property of machinery and electrical characteristics.					
	ROT_AXIS_ANG_AC C	None	500.0	After restart		
N14004	It specifies the max. angular acceleration of rotary axis. In any case, the angle acceleration					
	of rotation axis should never be above this value, which is determined by the property of					
	machinery and electrical characteristics.					



No.	Name	Setting Range	Default	When Enabled		
2.0. Spino	2.0. Spindle General					
N20001	SPIND_VELO_MAX	0.0~999999.0	24000.0	After restart		
		le max. spindle speed which shoul	ld accord with the	e setting speed of		
	the inverter.	• •		5 1		
	PROG_SPINDLE_R					
Nonno	EV	0.0~999999.0	12000.0	Immediately		
N20002	If the spindle speed ha	as not been clearly specified by the	ne processing pro	ogram, it will use		
	this setting value.					
	SPINDLE_DELAY_TI	0.0~60.0	5.0	Immodiately		
	ME	0.0/ 960.0	3.0	Immediately		
N20003	It means the delay time	after the spindle receives the den	nand of start or st	top, which can let		
	. •	ng rotary speed at the time of rec	J	emand or let the		
		at the time of receiving the stop	demand.			
3.1 Tool L			T	T		
N31000	TOOI_LIB_SIZE	0~256	10	Immediately		
	•	m capacity of tool library.				
3.2. Tool	Setting		1	ı		
N32020	Name	No more than 127 letters or 63	""	Immediately		
N32160	characters					
etc	It specifies the tool's na			L P. C.I		
N32021 N32161	Diameter	None	0.0	Immediately		
etc	It specifies the tool's diameter.					
N32022	Length	None	0.0	Immediately		
N32162 etc	It specifies the tool's length.					
N32023	DIAMETER_WEAR	None	0.0	Immediately		
N32163 etc	It specifies the estimated abraded quantity of tool diameter during processing.					
N32024	LENGTH_WEAR	None	0.0	Immediately		
N32164 etc	It specifies the estimate	ed abraded quantity of tool length	during processin	g.		
N32030	Offset (X\Y\Z)	None	0.0	Immediately		
N32031 N32032	It specifies the tool offset along X-axis, Y-axis and Z-axis.					
4.1. Lube						
	ALITO LUB	0 (False): Disabled	0 (False)	Immediately		
N41000	AUTO_LUB	1 (True): Enabled	U (Faise)	пппешасету		
	It specifies whether to start lubrication pump automatically and periodically.					
N41001	LUB_INTERVAL	3.6∼3.6 e +006	18000.0	Immediately		
114 100 1	It indicates time interva	l between a certain startup of lubr	icating pump and	the next startup.		



No.	Name	Setting Range	Default	When Enabled	
N144000	LUB_DURATION	1.0~100.0	5.0	Immediately	
N41002	It indicates the startup time of the lubricating pump.				
N41004	MachTaskEndInform Type	0: Red lamp is out 1: Red lamp lights for 3 seconds	0 (False)	After restart	
	It specifies how to infor	m operators after the end of mach	nining task.	ı	
5.2. Hand	lwheel				
	HW_COUNT_STRIC TLY	0 (False): Disabled 1 (True): Enabled	0 (False)	After restart	
N52001	If this parameter is set as enabled, the moving distance of the machine will abide by to count of the handwheel pulse strictly. But if the handwheel rotates too fast, the machine would likely keep moving quite a long time even if the handwheel has stopped. If the parameter is set as "invalid", the system may have rapid response time to the rotation handwheel. But when the handwheel moves too fast, the moving distance of the machine tool may not be consistent with the rotation distance of handwheel.				
N52002	HW_DIRECTION	1: Maintain the original moving direction of the machine tool during the motion of handwheel1: Maintain the original moving direction of the machine tool during the reverse motion of handwheel.	-1	After restart	
	It sets the positive and negative moving direction of the machine when the handwheel rotates. Specifically, if the parameter is set '1' and the handwheel runs CW, the machine will move in the same direction as the original direction. If the parameter is set '-1' and the handwheel runs CCW, the machine will move in opposite direction to the original direction.				
N52003	HW_MULTIPLE_1 0.001~10.0 0.001 After restart  Under the circumstance of X1 ratio, the system will take the pulse quantity produced by the handwheel as the actual pulse quantity that has been received.				
N52004	HW_MULTIPLE_10 0.001~10.0 0.01 After restart  Under the circumstance of X10 ratio, the system will take 10 times of the pulse quantity produced by the handwheel as the actual pulse quantity that has been received.				
N52005		0.001~10.0 e of X100 ratio, the system will tal			
N52006 N52007	HW_GEAR_FACTOR 1~1000 1 After restart  When the machine tool is controlled by handwheel, the parameter specifies the feed rate's magnification. You need to set the numerator and the denominator based on this proportion. If the numerator is small and the denominator large, the proportion will be small, and the relative feed rate of the machine tool will be small too. On the contrary, if the				



No.	Name	Setting Range	Default	When Enabled		
	numerator is big as the denominator small, the proportion will be large, and the relative feed rate of the machine tool will be fast too.					
	HW_ACC	1.0~1000.0	200.0	After restart		
N52010		It sets the acceleration when handwheel is selected. The smaller the value is, the more stable the motion will be.				
N52020	Support NKEX Keyset	0: Null 1: Mini keyboard 2: Naiky extension keyboard	0	After restart		
	It specifies the type of	•	T	1		
N52021	Install Extend Keyboard Hook	1 (True): Install 0 (False): Not install	1 (True)	After restart		
	It specifies whether to i	nstall extended keyboard hook.				
6.1 Coord	dinate					
N61200	PROCESS_ONCE_ WHEN_REPEATED	0 (False): Multi-processing 1 (True): Processing once	0 (False)	Immediately		
	It specifies whether to	It specifies whether to process a line or arc only once if it is repeated.				
N61300	Enable_LocatedCoor	0 (False): Disabled 1 (True): Enabled	0 (False)	Immediately		
	It specifies whether to	enable the located coordinate.				
6.2 G Co	de Options					
	RT_DEC_LEN	0.0~999.0	5.0	Immediately		
N62000	at approaching speed)	ecelerate when it is close to a targeto to protect the tool. The parameter machine tool begins to decelerate	r indicates the dis	stance away from		
N62001	SPEED_WHILE_AP PROACHING_TO_W ORKPIECE	0.001~99999.0	300.0	Immediately		
		es the feed rate when the tool app ne workpiece is less than decelera		kpiece. (Distance		
	IJK_INC	0 (False): Disabled 1 (True): Enabled	1 (True)	Immediately		
N62020						
	RADIUS_TOL	0.0~9999.0	1.0	Immediately		
N62021	In the IJK incremental manner of G02 and G03, it will count the radius twice. Generally, the two values of the radius are not equal with their difference called tolerance of arc. Usually, the tolerance of arc instruction is not very large. Recommended tolerance is 10 micron.					
N62090	G09_TOL(X\Y\Z)	0.0~99.0	0.001	Immediately		
	_	l .	l .	,		



No.	Name	Setting Range	Default	When Enabled
N62091 N62092	The parameter can be used together with the accurate stop command of G09 and with the parameters of pulse equivalent. G09 Accurate Stop, non-modal.  The instruction is only effective for the specified procedure block. To execute this command, the tool will decelerate at the end of the procedure block, and implement the arriving check, and then execute the next block.  If user is critical of the accuracy and the machine is equipped with the device of grating measure or encoder, please connect the device of grating measure (encoder) with the corresponding port of the NcStudio motion control card. When the system implements the accurate stop, the tool will decelerate to stop at the end point of the procedure block. In the meantime, the system will compare the tool actual position transmitted back by the device of grating measure (encoder) with the target position. If the warp is larger than the tolerance, the system will make fine adjustment and check again the warp until it is within the tolerance. If the serial motions have not been completed within the limited time, an error will be reported. To this point, user needs to regulate the following parameters or properly increase the tolerance: N11302 [ACPOS_SAMPLE_INTERVAL], N11320 ~ N11321 [ACPOS _ENCODER_DIR], N11330 ~ N11331 [ACPOS			
N62410	ENABLE_CUTTER_ COMPENSATION	FACTOR], N11340 ~ N11341 [ACI 0 (False): Disabled 1 (True): Enabled enable cutter compensation.	0 (False)	Immediately
N62730	•	-99999.0~99999.0	0.0 n the G73_G83	Immediately circular drilling
N62760	command.  DRILL_STOP_DIR			
6.3 Track	Control			
	TRACK_CORNER_T	0.0~0.1	0.0	Immediately
N63001	For the fine finish of the whole workpiece, at the connection point between two procedure blocks, the cutter may not move to the specified position accurately. When the difference between the actual tool position and the specified position is equal to the value of this parameter, the system will regard it as the completion of this procedure block.			
N63002	EXACT_STOP_TIME   0.0~999.0   0.0   Immediately   During the processing, because of the different inertia of each axis, the servo system may meet lag phenomenon. This parameter is used to overcome the lag phenomenon produced by the servo system.			
N63006	SMOOTH_time	0~0.064	0.0	Immediately



No.	Name	Setting Range	Default	When Enabled	
	The bigger the value of the parameter is, the smoother the surface of the workpiece will be. But if the value of the parameter is too large, it will affect the dimension of the workpiece. Setting the value as "0.05" is recommended.				
6.4 Spee	d/ACC		T		
	AX_STARTUP_VOL	0.0~100000.0	0.0	Immediately	
N64000	It is corresponding to the start-up frequency of stepping motor, which is the minir rate of machine motion.				
N64020	RAPID_TRAVERSE_ FEEDRATE	0.0~100000.0	6000.0	Immediately	
1104020	It is the default running (under G00 command)	speed (not the processing speed)	) of the machine t	tool in positioning	
N64040	DEFAULT_FEEDRAT E	0.0~100000.0	1200.0	Immediately	
1004040	It is the default processing speed (not the positioning speed) of the machine tool in positioning.				
	AX_LINEAR_MACH_ ACC	0.001~100000.0	1200.0	Immediately	
N64100	The acceleration of a single axis is used to describe the ability of a single feed axis to decelerate or accelerate. It depends on the physical properties of the machine tool. The larger the value is, the higher the efficiency is. But if the value is too large, abnormal condition may occur.				
N64101	AX_LINEAR_POST_ ACC	0.001~100000.0	1200.0	Immediately	
	It specifies the maximu	m acceleration of each feed axis	when the machin	e is positioning.	
	Z_LINEAR_ACC	0.001~100000.0	400.0	Immediately	
N64102		m acceleration of Z-axis.		,	
	AX_CON_ACC	0.001~100000.0	3000.0	Immediately	
N64120	It is used to describe the maximum acceleration of the feed motion occurring on the adjacent axes. It is recommended to set the value as 1~2 times of the acceleration of the single axis.				
NC44FO	AX_ACC_ACC	None	10000.0	Immediately	
N64150	It specifies the change	rate of single axis acceleration (a	cceleration of acc	celeration).	
N64203	INTERP_ALGO	0: Echelon Algorithm 1: Type S Algorithm 2: LEP Algorithm	1	Immediately	
	To decrease error, choose the most suitable interpolation algorithm after debugging.				
N64207	LIMIT_ARC_VELO	0 (False): Disabled 1 (True): Enabled	1 (True)	Immediately	
	It specifies whether arc velocity limit is valid.				
N64208	REF_CIRCLE_MAX_ VELO	0.001~100000.0	1800.0	Immediately	



No.	Name	Setting Range	Default	When Enabled	
	The reference circle is a sample when the machine tool makes a workpiece of arc type. The max. speed of the reference circle refers to the biggest tolerant speed when the machine tool is processing a circle of 10mm diameter without producing vibration. When processing a circle, the system will automatically count out the maximum centripetal acceleration on the arc according to the value of this parameter to make sure no strenuous vibration produced by the machine tool. The larger the value of this parameter is, the larger the following error of the circle is. Please fill in here the measured accepted speed.				
	CIRCLE_MIN_VELO	0.001~100000.0	180.0	Immediately	
N64209	When machining a circle of a small radius, even if a very small arc feed speed can produce a very big centripetal acceleration, while the arc speed limit will result in a lower processing speed. To ensure the processing efficiency, when the speed counted out by the system is lower than the set value of the parameter, the system will take this set value as the processing speed.				
N64210	ENABLE_MCM	0 (False): Disabled 1 (True): Enabled	0 (False)	Immediately	
1104210	•	e system will process steadily. V table in processing, but processin	•		
N64211	CUR_EFF_SCHEME _INDEX	0-4	0	Immediately	
	It is used to specify an	efficiency strategy group.			
N64212 N64217 N64222 N64227	ANGLE ACC GENE	None	Group 0: 1 Group 1: 1.5 Group 2: 2 Group 3: 2.5 Group 4: 3	Immediately	
N64232	It specifies regulatory factor of corner acceleration.				
N64213 N64218 N64223 N64228	ACCGENE	None	Group 0: 1 Group 1: 1.1 Group 2: 1.2 Group 3: 1.3 Group 4: 1.5	Immediately	
N64233	It specifies regulatory fa	actor of corner acceleration in sing	gle axis machinin	g.	
N64214 N64219 N64224 N64229	CIRCLELIMITSPEED GENE	None	Group 0: 1 Group 1: 1.2 Group 2: 1.6 Group 3: 2 Group 4: 999	Immediately	
N64234	It specifies regulatory factor of circle speed limit.				
N64240	SLIDE_TIME_GENE It is used to specify the	0.01-10.0 ratio of unit processing time to con	1.0 ntrol periodic time	Immediately when the speed	
	is smooth.				



No.	Name	Setting Range	Default	When Enabled
6.5 File Translation N65000 ~ N65003 PLT File Translation Parameter PLT file translation parameters are used for the translation of PLT files. PLT is a 2D processing file format defined by America HP company. It is usually used on the embossment or advertising carving. N65100~N65106 DXF File Translation Parameter DXF file translation parameters are used for the translation of DXF files. N65200~N65205 ENG File Translation Parameter ENG file translation parameters are used for the translation of the ENG files.				
N65000	TOOL_UP_HEIGHT( PLT) 0.0~99999.0 1.0 After reloading PLT is a 2D processing file format defined by America HP company, usually used on the embossment or advertising carving. This parameter specifies the cutter lifting height in rapid traverse.			
N65001		0.001~99999.0 s a 2D file format defined by Amerovertising carving. Normally, 1plt=		·
N65002	TOOL_DISTANCE_A T_AREA(PLT)  O.0001~99999.0  The space between cutters needs to be confirmed by the diameters of cutters. In order to fulfill the processing, part of the adjacent cutters must be overlapped, which is the space between cutters in PLT machining.			
N65003	CUTTING_DEPTH(P LT)	-99999.0~0.0	-1.0	After reloading
N65100	It specifies the cutting of TOOL_UP_HEIGHT( DXF)	0.0~99999.0	1.0	After reloading
N65101	CUTTING DEPTH(DXF) It specifies the cutting of	-99999.0~0.0	-1.0	After reloading
N65102	DEPTH_LAYER(DXF	-99999.0~0.0  f each cutting during 2D processing	0.0	After reloading
N65103	FIRST_POINT_AS_ ORIGIN(DXF)	O (False): Not take the starting point as the zero point     1 (True): Take the starting point as the zero point     set the first coordinate point as zero.	1 (True)	After reloading



No.	Name	Setting Range	Default	When Enabled
	ENABLE_MACHINE _DIVIDUALLY(DXF)	0 (False): Disabled 1 (True): Enabled	0 (False)	After reloading
N65104	When processing of DXF file, the system will deem the action of cutter lifting as the separating mark between the adjacent shapes. If no cutter lifting, the system will deem there is only one shape being processed. Once the cutter lifting is implemented, it indicates that the processing has been finished. For example, process several adjacent circles on one workpiece, the depth of the circle is 10 mm, each feed depth of Z-axis is 2 mm. If the parameter is set to be valid, the cutter will lift after machining 5 times, then one circle is completed, the machine will process the next circle. If the parameter is set to be invalid, the machine will process current circle only once and then go to the next circle. When all circles have been processed once, the system will re-implement the previous process four times until all the processing has been finished.			
N65105	ENABLE_MACHINE _BOTTOM(DXF)	0 (False): Disabled 1 (True): Enabled	0 (False)	After reloading
N65106	It only enables thresho FORCE_TO_RECOG NIZE_DXF_FILE_AS _METRIC_SYS(DXF)	Id manipulation on the surface of v 0 (False): Invalid 1 (True): Valid	workpiece (2D cu 0 (False)	After reloading
	It sets whether to forcibly recognize dxf file in metric size.			
N65200	TOOL_UP_HEGHT( ENG)	0.0~99999.0	1.0	After reloading
	It specifies the cutter lif	ting height in rapid traverse.		•
	ENABLE_CHANGE_ TOOL_PROMPT(EN G)	0 (False): Invalid 1 (True): Valid	1 (True)	After reloading
N65201	If it is set valid, the machine will suspend when meeting command of tool change. Z-axi will uplift, and the system will remind user to change tool in the prompt column and user can implement tool change now; otherwise, if it is set invalid, the machine will not suspen when meeting command of tool change, but the system will still remind user to do too change in the prompt column.			column and user will not suspend
N65202	RECYLE_MACHININ G_TIMES(ENG)	1~9999	1	After reloading
	It indicates the cycling processing times in ENG file machining.			
N65203	MACHINING_ENG_F ILE_BY_TOOL_NUM BER(ENG)	0 (False): Not Use 1 (True): Use	0 (False)	After reloading
1153203	When using the function of tool selection, once user opens an ENG file, a dialog will pop up and require user to select a tool (specified in the Eng file instead of the tool set in the system default setting) to do the processing.			



No.	Name	Setting Range	Default	When Enabled
N65204	DEEP_HOLE_MACH INING_WAY(ENG)	Reciprocating chip removal     High-speed reciprocating chip removal	0	After reloading
	It indicates the manner	of processing deep hole in ENG t	ile machining.	
Nosoos	RETRACT_VALUE(E NG)	0.0~99999.0	1.0	After reloading
N65205	It indicates retracting a removal manner.	amount when drilling deep holes	in high speed r	eciprocating chip
6.6 Chan	ge Tool			
N66000	PROMPT_CT_CMD	0 (False): Invalid 1 (True): Valid	0 (False)	Immediately
	It decides whether to pa	ause and prompt tool change afte	r tool change cor	nmand pops up.
N66001	CTUP	-99999.0~99999.0	0.0	Immediately
1100001	It specifies the upper pe	osition in tool change.		
N66002	CTDOWN	-99999.0~99999.0	0.0	Immediately
1100002	It specifies the lower po	osition in tool change.		
N66003 N66004	SPINDLE_POSITION _IN_TOOL_CHANGE (X\Y\Z)	-99999.0~99999.0	X: 10000.0 Y: 10000.0 Z: 10000.0	Immediately
N66005	It specifies the machine	e coordinate of spindle in tool char	nge.	
N66013 N66014	CHANGE_TOOL_AH EAD_POS (X\Y\Z)	-99999.0~100000.0	X: 0.0 Y: 0.0 Z: 0.0	Immediately
N66015 It specifies the machine coordinate of the pre-point during tool ch		g tool change.		
N66023	CHANGE_TOOL_SP EED	0.0~100000.0	3000.0	Immediately
	It specifies the moving	speed of the spindle during tool cl	hange.	
N66024	CHANGE_TOOL_ZA XIS_SPEED	0.0~9560.0	1800.0	Immediately
	It specifies the up and down speed of Z-axis during tool change.			
7.1 Manu				
	JOG_VOL	0.0~100000.0	1200.0	Immediately
N71000				ystem when user
	RAPID_JOG_VOL	0.0~100000.0	3000.0	Immediately
N71001	The speed specified by this parameter is the default running speed of system when user presses down both direction keys and acceleration key under manual mode.			
N74004	JIGGLE_VOL	0.0~9560.0	1200.0	Immediately
N71021	It specifies the jiggle ve	elocity under auto mode.		



No.	Name	Setting Range	Default	When Enabled	
7.2 AUTC	)				
N72001	SPEED_ASSIGN_TY PE	<ul><li>0: Select file speed</li><li>1: Select default speed</li><li>2: Specify speed in proportion</li></ul>	0	Immediately	
	It specifies the style se	ection of feed rate.			
N72002	IGNORE_PROG_SPI NDLE_REV	0 (False): Valid 1 (True): Invalid	0 (False)	Immediately	
	If this parameter is set	as "True", the specified rotary spe	ed in file will be i	nvalid.	
N72004	STOP_SPIND_AT_E ND	0 (False): Don't stop 1 (True): Stop	1 (True)	Immediately	
1172004	It specifies whether the stopped.	e spindle will stop rotating autor	matically when t	he processing is	
N72010 ~	ENABLE_WORKAR EA_CHECK (X\Y\Z)	0 (False): Invalid 1 (True): Valid	0 (False)	Immediately	
N72012	•	It specifies whether to enable workpiece coordinates range check. The range is constituted by the upper limit and the lower limit of the workpiece coordinates.			
N72020 ∼	WORKAREA_LOWE R_LIMIT ( $XYZ$ )	-99999.0~99999.0	-10000.0	Immediately	
N72022	It specifies the lower lin	nit of the workpiece coordinates in	each direction.		
N72030 ∼	WORKAREA_UPPE R_LIMIT (X\Y\Z)	Lower limit of worktable stroke (machine coor.)~99999.0	10000.0	Immediately	
N72032	It specifies the upper lin	mit of coordinates of workpiece in	each direction.		
7.3 Pause	е				
	PAUSE_DOWN_VOL	0.0~100000.0	600.0	Immediately	
N73000	It sets the speed (retrocontinues after pause.	urning to cutting point before pa	ause) of Z-axis	when processing	
NI72004	PAUSE_UP_VOL	0.0~100000.0	600.0	Immediately	
N73001	It sets the lifting speed	of Z axis when pausing occurs.			
N73002	PAUSE_OPTION	Specify cutter lifting amount     Specify workpiece     coordinate     Specify machine coordinate	0	Immediately	
	It sets positioning mode	e of Z-axis in pausing.			
N73003	Z_WPCOOR_ON_PA USE	0.0~9999.0	10.0	Immediately	
147 3003	It is valid when parameter N73002 is set "1", appointing the workpiece coordinates of the position of Z-axis when pausing occurs.				
N73004	Z_OFFSET_ON_PA USE	0.0~500.0	10.0	Immediately	
147 3004	•	eter N73002 is set "0", appointing oefore pause when pausing occur	•	height of Z-axis	



No.	Name	Setting Range	Default	When Enabled
N73005	STOP_SPINDLE_AT _PAUSE	0 (False): Don't stop 1 (True): Stop	1 (True)	Immediately
	It specifies whether spi	ndle stops rotation in pausing.		
N73006	Z_MACHCOOR_ON _PAUSE	-350.0~0.0	0.0	Immediately
147 3000	It is valid when parame when pausing occurs.	eter N73002 is set "2", appointing	the machine cod	ordinate of Z axis
7.4 Bkref				
N74000	CANCEL_REFPT_A FTER_ESTOP	0 (False): Don't cancel 1 (True): Cancel	1 (True)	Immediately
1174000	If this parameter is set will be cleared.	"true", once there is ESTOP, the	current symbol c	of reference point
	NEED_REFPT_BEF ORE_MACHINING	0 (False): Don't need to back to the reference point 1 (True): Need to back to the reference point	1 (True)	Immediately
N74001	Servo system adopts incremental coding machine, but the system can only recognic position relative to the previous one. To confirm an arbitrary point with fixed machine, user needs to select a reference point (machine origin). During the processif an ESTOP or power off occurs, an error of displacement may appear when machine origin, the system can retrieve the coordinates workpiece to ensure the processing accuracy.		th fixed machine g the processing, ar when machine	
N74010 ~ N74012	REFPT_COOR(X\Y\ Z)	Lower limit of worktable stroke (machine coordinate)~ upper limit of worktable stroke (machine coordinate) ordinates of the machine origin.	0.0	Immediately
N74020 N74021 N74022	COARSE_LOCATIN G_DIR(X\Y\Z)	1: Positive direction -1: Negative direction	X: -1 Y: -1 Z: 1	Immediately
11/4022	reference point.	ection of coarse positioning durin	ig the process c	n backing to the
N74030 N74031 N74032	COARSE_LOCATIN G_VELO(X\Y\Z)	0.001~10000.0	X: 1800.0 Y: 1800.0 Z: 1800.0	Immediately
1174032	It sets the feed rate of	coarse positioning during the proc	ess of backing to	zero point.
N74050 N74051	FINE_LOCATING_DI R(X\Y)	1: Positive direction -1: Negative direction	X: 1 Y: 1 Z: -1	Immediately
N74052 It sets the motion direction of fine positioning during the process of backing h		ig home.		



No.	Name	Setting Range	Default	When Enabled
N74060 N74061	FINE_LOCATING_V ELO(X\Y\Z)	0.001~10000.0	X: 60.0 Y: 60.0 Z: 60.0	Immediately
N74062	It sets the feed rate of f	ine positioning during the process	of backing to the	e reference point.
N74080 N74081	BACK_DISTANCE(X\Y\Z)	-1000.0~1000.0	X: 2.0 Y: 2.0 Z: -2.0	Immediately
N74082	It refers to the addition out of zero sensitive zo	al moving distance after the comp	oletion of fine po	sitioning to move
N74090	Location Times  It is used for counting t	None he times of fine positioning during	1 backing to the re	Immediately eference point.
7.5 Meas	ure			_
N75001	CALI_SPEED  It is the speed when measurement.	None tool is approaching the surfa	60.0 ce of tool prese	Immediately etter during tool
	CALI_TIMES	None	1	Immediately
N75002	It counts the times of fine positioning for tool approaching the surface of tool presetter during tool measurement.			
NIZEO20	CALI_TOLERANCE	0.0~10.0	0.1	Immediately
N75020	It specifies the maximu	m allowable error during several t	ool measuremen	t.
	LIGHT_CONTROL_ OUTPORT_INDEX	1: Without disconnection with measurement line -1: With disconnection with measurement line	-1	Immediately
N75023	line. If the function is	ware involves the function of disc s included, the port will give a machining and another signal to nt.	signal to disc	onnect with tool
N75100	MOB_CALI_THICKN ESS	-1000.0~1000.0	0.0	Immediately
N75100	It is used to specify the thickness from the surface of the tool presetter to origin in floating measurement.		to the workpiece	
NZESS	FIXED_CALI_HEIGH T	None	10.0	Immediately
N75201	This parameter specifies the height from the surface of tool presetter to the surface of worktable in fixed measurement.			
N75203	FIXED_CALI_QUICK _SPEED	None	300.0	Immediately
1975203	It specifies the trave measurement in fixed r	rsing speed from the highest measurement.	point to initial	position of tool



No.	Name	Setting Range	Default	When Enabled
N75210 N75211 N75212	FIXED_CALI_POS (X\Y\Z)	-99999.0~99999.0	X: 0 Y: 0 Z: -1	Immediately
1175212	It specifies the machine	e coordinate of tool measurement	position in fixed	measurement.
7.9 Opera	ation others			
N79000	Z_DOWN_VELO_OP TION	<ul> <li>0: Don't dispose</li> <li>1: Only Z downwards</li> <li>processing motion available</li> <li>2: Including Z downwards</li> <li>processing motion available</li> </ul>	0	Immediately
	It specifies the feeding	speed of Z-axis.		
N79001	Z_DOWN_VELO	0.0~100000.0	480.0	Immediately
147 900 1	It specifies the down fe	ed rate of Z-axis.		
	SAFE_HEIGHT	0.0~1000.0	10.0	Immediately
N79003	It calculates the height relative to workpiece coordinate origin. The system deems the horizontal movement at the specified height is safe, generally used to perform the operations of backing to the origin and breakpoint resuming.			
	ACTION_AFTER_PR OGRAMMING	O: Don't move Back to fixed point Back to workpiece origin	0	Immediately
N79100	To improve efficiency, you may need to move the machine tool to a fixed position after the machining is finished. E.g., raise Z-axis to a fixed position after finishing workpiece processing to replace another workpiece. If you want the machine tool to automatically back to a fixed point after processing, please select this parameter and set the fixed point.			
N79110	FIXPT_POS (X\Y\Z)	-99999.0~99999.0	0.0	Immediately
N79111 N79112	It is only valid when the parameters are set "1", indicating the machine coordinates of the fixed position.			
N79200	DISPLAY_SIMU_OU T_RANGE	0 (False): Don't clew 1 (True): Clew	1 (True)	Immediately
	mode.	prompt under the situation of out o	or mechanicai str	oke in simulation
N79400	AUTO_BACK_TO_Z ERO	<ul><li>0 (False): Don't back to machine origin</li><li>1 (True): Back to machine origin</li></ul>	0 (False)	After restart
	It specifies whether to automatically back to the machine origin.			
8.1 Positi	on View			
N194000	AUTO_LOAD_TRAC K	0 (False): Invalid 1 (True): Valid	1 (True)	Immediately
N81000		ystem will analyze the processi I, which will help user load the trac	_	atically after the



No.	Name	Setting Range	Default	When Enabled
	AUTO_LOAD_TRAC K_LIMIT	0~100000	1000	Immediately
N81001	When the size of the pr	ocessing file is smaller than this v	alue, the system	will automatically
	analyze the processing track; but when the size of the processing file is equal to or la			equal to or larger
	than this value (the processing file is too large), the track will not be analyzed.			
8.3 Progr	3.3 Program View			
	Y_Axis Work Range	1: Positive interval	1	Immediately
N83000	1_Axis Work Range	2: Negative interval	·	Illinediately
	It sets the work space of Y-axis of machine tool.			
8.7 Show	Show Interface			
	Show Interface	0/1/2	2	After restart
N87000	It selects the interface display type. 0: Solely open NcEditor; 1: Solely open NcStudio; 2:			
	Open both NcEditor an	d NcStudio		



# 10. NcStudio Appendix

## 10.1. Appendix of Shortcut Keys

Most of the following shortcut keys are global, while the local shortcut keys will be identified with " $\star$ ", with the precondition of enabling offered.

♦ Num buttons can be used to select function section and current mode, sorting from left to right in the interface:

1	Position function section
2	Offset function section
3	Program function section
4	System function section
5	Parameter function section
6	Diagnosis function section
7	Auto mode
8	Manual mode
9	Reference point mode

Select current function picture under the function section by key Q, W, E, R, T,Y, sorting from left to right:

Q	The First function picture
W	The Second function picture
Е	The Third function picture
R	The Forth function picture
Т	The Fifth function picture
Υ	The Sixth function picture

◆ Ctrl + F\* (\* represents a figure), selecting the mode by combination keys:

Ctrl + F2 Manual mode

Ctrl + F3 Reference point mode



### Other short-cut keys:

F1~ F7 Operation buttons (valid in function picture)

★F9 Start (valid in auto mode)

★F11 Stop (valid in auto mode)

Pause Reset

Ctrl + O Open and load

Ctrl + U Unload

Ctrl + N New

★Ctrl + F9 Selective processing (valid in auto mode)

Shift + F6 Set current point as workpiece origin

Shift + F9 Breakpoint resume



## 11. Software License Agreement

#### Important—Read Carefully before Using This Product:

The term "Software Product" includes all copies of the licensed software and its documentation. This license agreement is a legal agreement between You (either an individual, a legal entity or any affiliated companies or other entities) and Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). By installing, copying, or otherwise using the Software Product, you agree to be bound by the terms and conditions of this license. Unless otherwise stated in this agreement, you shall not use, copy, revise, rent, or transfer the Software product or any part of the Software Product for any other purposes.

#### **Description of Further Rights and Restrictions:**

- 1. You may install for use one copy of the Software Product on a single machine;
- 2. You may make a copy of the Software Product for archive or backup purposes and the copy is to be used on this machine only;
- You may transfer the Software Product and the license agreement to a third party, provided that the third party accepts the terms and conditions stated in this agreement, with prior express permission from Weihong Company;
- 4. When transfer confirmed, you shall transfer all the copies of the original documents and the supplementary documents to the third party or destroy all the copies un-transferred.
- 5. You can use the Software Product on a network server or intranet server only if it is stipulated in explicit terms that you are allowed to use the Software Product on a network server or intranet server, or you have purchased license for each node and terminal using the Software Product:
- 6. You may NOT sublicense, assign or transfer the license agreement;
- 7. You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
- 8. You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
- 9. The license agreement shall be terminated automatically upon you transfer the Software Product or copies of all or part of the Software Product to a third party.

### **Intellectual Property Rights Notice**

The Software Product and all intellectual property rights therein (including but not limited to any all copyrights, patents, trademarks, and publicity rights) are owned by Weihong Company. The Software Product is protected for Weihong Company on the basis of copyright law and international treaty provisions as well as on the basis of other laws and agreements regarding intellectual property. You are not allowed to remove the copyright statement made in the Software Product, and guarantee that you shall copy the copyright statement in all copies of the Software Product or of any part of the



Software Product. You are obliged to stop any form of illegal copying of the Software Product and accompanying materials.

#### After-sales Guarantee:

Weihong Company guarantees that for 90 days from the date of shipment the software carrier will be free from defects in materials and workmanship. When such a defect has been confirmed, our only responsibility is to replace the software carrier. This remedy is your exclusive remedy. This after-sales guarantee is invalid for any carrier defect caused by accidents, abuses or maloperation. The replaced software carrier enjoys the remaining guarantee time of the original software carrier or of a 30-day guarantee time, whichever is longer.

Except as the after-sales guarantee made above, the Software Product does not enjoy any other form of after-sale guarantee.

#### **Limitation of Liability:**

The above guarantee, whether made explicitly or by implication, constitutes the entire contents of the guarantee, including guarantee for the commerciality and applicability of special application aim. Whether you follow other terms in this agreement or not, Weihong Company, as well as its agents and sales staff, will not be responsible for any profits loss, availability loss, business break-off or any forms of indirect, special, accidental or inevitable damage or claim made by any third party, generated from the using of the Software Product, even if Weihong Company has been informed of the possible occurrence of such events in advance.

#### Termination:

This license may be terminated by Weihong Company at any time once you violate any terms or conditions made in this agreement. Once the license is terminated, you are obliged to destroy all the copies of the Software Product or return them to Weihong Company.

#### **Applicable Law:**

Copyright Law, Regulations for the Protection of Computer Software, Patent Law and other relevant laws and regulations.

Now, we affirm that you have already read through this agreement and understood it thoroughly and agreed to comply with all of the terms and conditions of this agreement strictly.

Weihong Electronic Technology Co., Ltd.