NK300CX Integrated CNC System

Manufacturers' Manual

4th Edition

Weihong Electronic Technology Co., Ltd.

The information contained in this manual is constantly being updated. You can login the official website of Weihong Company www.en.weihong.com.cn to download the latest PDF edition for free.

Preface

About This manual

This manual is intended for manufacturers. If you use Weihong CNC system for the first time, it is suggested to read through this manual. If not, however, you can search for the desired information via the contents.

With 8 chapters, this manual can be divided into 5 parts, as follows:

1) Part 1: preface, introducing the precautions about transportation and storage, installation, wiring, debugging, usage and so on. You need to read them first carefully to ensure safe operations.
2) Part 2: illustration of hardware, including chapter 1, 2 and 6. The former two chapters introduce components of the system, the dimensional sizes as well as pin definition of IO ports of the Lambda controller. Chapter 6 presents parameter settings of servo drivers of various brands as well as their wiring diagrams with Lambda controller.
3) Part 3: introduction to software operation, including chapter 3 and chapter 4. Taking three-axis configuration of integral software and multi-Z axes software as examples respectively, two chapters illustrate detailed operations of single functionality and its corresponding interfaces, which will be an intuitional guidance to users and operators in real practice.
4) Part 4: maintenance, referring to chapter 5. In this chapter, possible problems and their countermeasures are listed, aiming to help users to respond instantly and take effective measures when possible failure occurs.
5) Part 5: last part of this manual, consisting of chapter 7 and chapter 8, corresponding to table of parameters of the system as well as the software license agreement.

Applicable Product Model

This manual is applicable to NK300CX-H and NK300CX-V. Refer to the table below for details.

Product Model	Remarks
NK300CX integrated CNC system	Herein referred to NK300CX as abbreviation, which can be used together with integral software and multi-Z software. Integral software takes three-axis configuration, four-axis configuration and five-axis configuration as a whole. Multi-Z software consists of linkage configuration and alternative configuration. With no further explanation, the system in this manual refers to NK300CX integrated CNC system.

Contact Us

You can contact us by the following info for technical support and pre-sales / after-sales service:

Company Name: Weihong Electronic Technology Co., Ltd.
Headquarters Address: No.1590, Huhang Rd., Fengxian, Shanghai, PRC 201401
Tel: +86-21-33587550
Fax: +86-21-33587519
Website: http://en.weihong.com.cn

Revision History

You can refer to the following table for the revision records of each edition.

Date	Edition	Revision
2016.07	R4	1) Wiring diagrams of Lambda 5S controller removed; 2) Auto tool change of servo tool magazine added, and section 3.20 updated; 3) Section 3.5 and 3.6 updated, returning to the machine origin with absolute encoder function; 4) Other revisions.
2016.02	R3	1) Contact information updated; 2) Wiring diagram of Lambda 5S controller updated; 3) Section 3.14.1 Tool Compensation updated;
2016.01	R2	Main revision contents are as follows: 1) Add section 0 returning to machine origin with absolute encoder function; 2) Add section 5.4.6 software failures because of automatic write number identification; 3) Update change tool flowcharts of linear tool magazine and circular tool magazine in section 0. 4) Other revisions.
2015.12	R1	Released for the first time.

Precautions

Precautions can be divided into caution and warning according to the degree of possible loss or injury in case of negligence or omission of precautions stipulated in this manual.

: general info, mainly for informing, such as supplementary instructions and conditions to enable a function. In case of negligence or omission of this kind of precautions, you may

not activate a function. Note that in some circumstances, negligence or omission of this kind of precautions could cause physical injury or machine damage.

WARNING: warning info requiring special attention. In case of negligence or omission of this kind of precautions, you may suffer physical injury, or even death, machine damage or other losses.

WARNING

1) Precautions Related to Storage and Transportation

- The products should be transported properly in terms of the weight;
- An excess of specified quantity of stacking products is prohibited;
- Climbing, standing or placing heavy loads on the products is prohibited;
- Dragging or carrying the products via cables or devices connected to them is prohibited;

2) Precautions Related to Installation

- Only when this equipment installed in the qualified electricity cabinet can it be used. The construction of the cabinet must reach IP54 grade of protection;
- Paste sealing strips on the joint of the cabinet to seal all the cracks;
- Cable entry should be sealed while easy-to-open on the spot;
- A fan or heat exchanger should be adopted for the heat dissipation and air convection of the cabinet;
- If a fan is adopted, air strainer is a must in air inlet or air outlet;
- Dust or cutting fluids may have access to the CNC device via the tiny cracks and tuyere. Therefore it is necessary to pay attention to the surroundings and air flow direction of the air vent to make sure that the outflow gas is towards pollution source;
- 100 mm space should be preserved between the back of the CNC device and the cabinet wall for plugging cable connected with the device and the ventilation & heat dissipation in the cabinet;
- Space between this device and other equipment should also be preserved according to the requirements;
- The product should be installed firmly and without vibration. During installing, casting, knocking, striking, or loading on the product is forbidden;
- To reduce electromagnetic interference, power-supply components used should be above AC or DC 50V and the space between cable and CNC device should be preserved above 100mm;

WARNING

- It will be better if CNC device is installed at a position facilitating debugging and maintenance.

3) Precautions Related to Wiring

- Only qualified people are allowed to participate in the wiring and checking;
- The CNC device should be grounded reliably and grounding resistance should be less than 4 ohm. Neutral line is absolutely not allowed to replace earth wire. Otherwise, it may result in malfunction of the device due to the interference;
- Wiring should be firm and steady, or mal-operation may occur;
- Voltage values and positive & negative polarity of any connection plug should be in accordance with specifications set forth in the manual, or it may result in breakdowns such as short circuit and permanent damage to the device;
- To guard against electric shock or CNC device damage, fingers should keep dry before plugging or touching switch;
- The connecting wire should not be damaged and squeezed, or the leakage or short circuit may occur;
- It is prohibited to plug or open the chassis of CNC device when power on.

4) Precautions Related to Running & Debugging

- Parameters setting should be checked before running, since wrong setting may lead to accidental movements;
- Modification to parameters should be within the allowable range, or such breakdowns as unsteady running and machine damage will occur.

5) Precautions in Use

- Before power-on, please make sure that the switch is on blackout to avoid occasional start-up;
- Please check the electromagnetic compatibility during electrical design in order to avoid or reduce electromagnetic interference to the CNC device. A low pass filter should be employed to reduce electromagnetic interference if there are other electrical devices nearby;
- It is not allowed to frequently power on and power off. It is recommended to power up the machine again at least one (1) minute later after power failure or blackout.

CAUTION

1) Precautions Related to Product and Manual

- Matters related to restrictions and functions available stipulated in the manuals issued by the machine manufacturer are prior to those in this manual;
- This manual assumes all the optional functions are available, which you must confirm through manuals issued by the machine manufacturer;
- Please refer to manuals issued by the machine manufacturer for the instructions of machine tools;
- Functions, and software interfaces vary with the system and the version of software. Before using the system, you must confirm the specifications.

2) Precautions When Opening the Package

- Please make sure that the products are what you have ordered;
- Check if the products are damaged in transit;
- Check if the components and accessories are damaged or missing in terms of the detailed list;
- Please contact us promptly if product discrepancy, accessory missing or transit damage occurs.

Contents

1 Overview

1.1 Hardware

1.1.1 System Configuration

According to model of the operational panel, NK300CX is divided into NK300CX-H and NK300CX-V, whose pictures are shown as Fig. 1-1 and Fig. 1-2 respectively.

Fig. 1-1 A picture of NK300CX-H

NK300CX-H integrated CNC system consists of the following components:

- One NK300CX host
- One WH106C operation panel
- One WH201C keypad panel
- One Lambda 5S series controller
- Two DB9M/F cable (40cm)
- One DB9M/F cable (length optional)
- One DB9M/F cable (40cm, optional)
- One extended terminal board EX31A1 (optional)
- Handwheel NK-MPG-06 (optional)

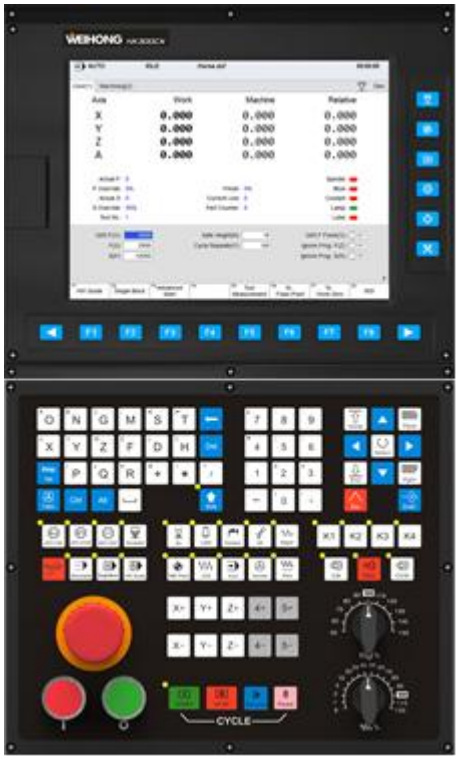


Fig. 1-2 A picture of NK300CX-V

NK300CX-V integrated CNC system consists of the following components:

- One NK300CX host
- One WH108C operation panel
- One Lambda 5S series controller
- One DB9M/F cable (40cm)
- One DB9M/F cable (length optional)
- One DB9M/F cable (40cm, optional)
- One extended terminal board EX31A1 (optional)
- Handwheel NK-MPG-06 (optional)

CAUTION

1) Apart from difference of names and models of specific components, NK300CX-H shares the same with NK300CX-V in aspects of functionalities, size, etc. you can refer to above contents for details of each component.

2) Note that each components of NK300BX is not compatible with that of NK300CX, which is history product of NK300CX.

1.1.1.1 Host

Front view of the host is shown as Fig. 1-3.

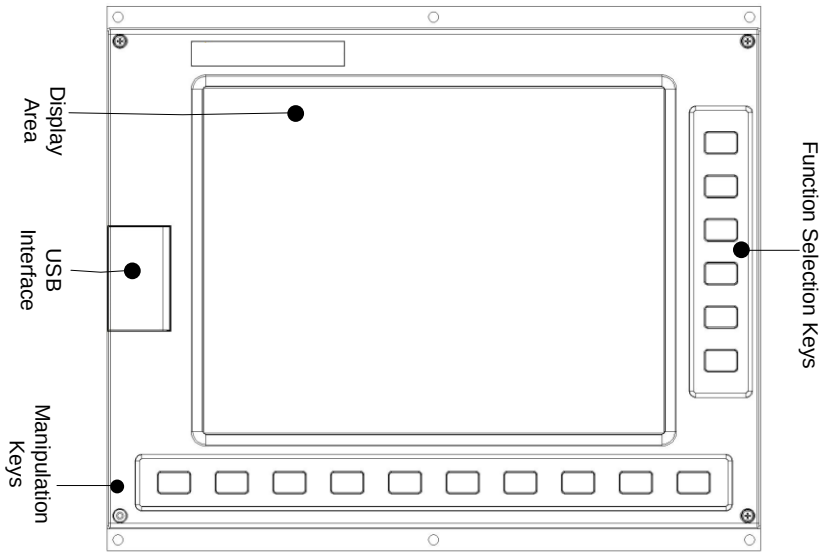


Fig. 1-3 Front view of NK300CX host

1. Display area. It is human-machine interface, or user's interface, which is user-friendly and ease of use.
2. USB interface. It is used for USB removable flash disk connection, protected by a cover.
3. Function selection keys. There are altogether 6 functional areas, including machining, advanced, program, system, parameter and diagnosis. You can access the functional area by pressing the key here directly.
4. Manipulation keys. Including F1~F8, which are used to activate the functions indicated by the soft keys.

See Fig. 1-4 for rear view of the host.

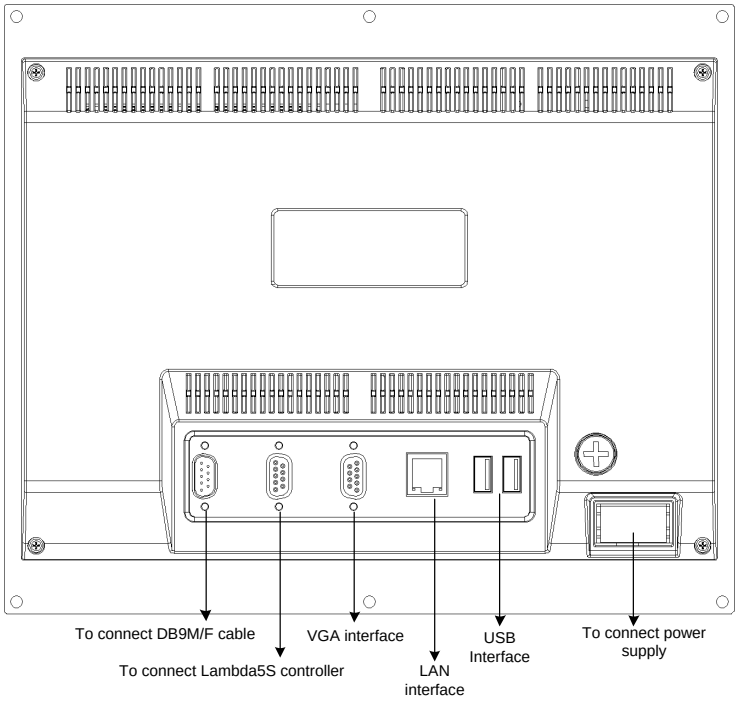


Fig. 1-4 Rear view of the host

1. DB9M/F cable interface. For NK300CX-H, the interface is used to connect with keypad panel; while for NK300CX-V, it is used to connect with operation panel.
2. Controller interface. It is used to connect with Lambda 5S controller.
3. VGA interface. It is used to connect with the monitor or display.
4. LAN interface. It is used to connect with network, with transmission rate of 100Mbps.
5. USB interface. Two USB interfaces, which are used to connect with removable flash disk.
6. Power interface. It is used to connect with 220V power supply.

1.1.1.2 Operation Panel

WH106C operation panel and WH201C keypad panel are adopted for NK300CX-H, while WH108C operation panel is adopted for NK300CX-V.

◆ **Illustration of WH106C Operation Panel**

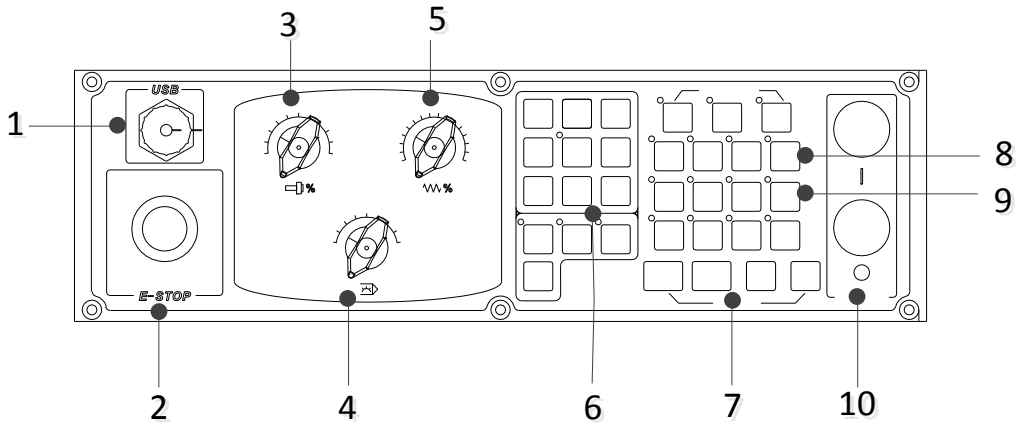


Fig. 1-5 Front view of WH106C operation panel

See below for detailed information of each part.

1. USB interface (with a protection cap), used for connection with removable flash disk.
2. Emergency stop switch. Anytime there is possible danger, operator can press E-stop switch to stop the machine to protect safety of both human and machine, and when danger is cleared, turn the switch in clockwise direction to remove the alarm.
3. Spindle override knob, which is used for spindle speed override adjustment. Refer to section 3.8 for details.
4. Mode selection knob, which is used for mode selection, as shown in Fig. 1-6.

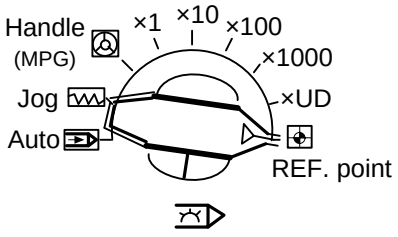


Fig. 1-6 Mode selection knob

5. Feed override knob, which is used for feedrate override adjustment. Refer to section 3.12.1 for details.
6. Axis direction keys, used for manual control of each axis movement in jog mode or jiggle mode. See Fig. 1-7 for concrete keys. How to use [Rapid] key? In manual mode, when any axis direction key and [Rapid] key are together pressed, the axis moves at manual high speed, or called rapid jog speed; when any axis direction key is pressed alone, the axis moves at manual speed, or called jog speed.

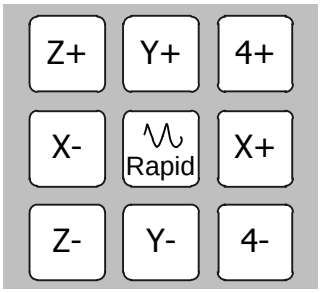


Fig. 1-7 Axis direction keys

7. Motion control keys. In auto mode, you can press [Cycle Start], [Pause] and [Cycle Stop] keys to conduct corresponding functions. The moment power interruption or emergency stop occurs, you can press [Resume] key to resume machining from the interrupted point to save time on condition that the workpiece origin is accurate for sure.
8. Extension keys area. [K1], [K2], [K3] and [K4] keys are included, used for user-defined functions.
9. Auxiliary function keys area. See below for details.

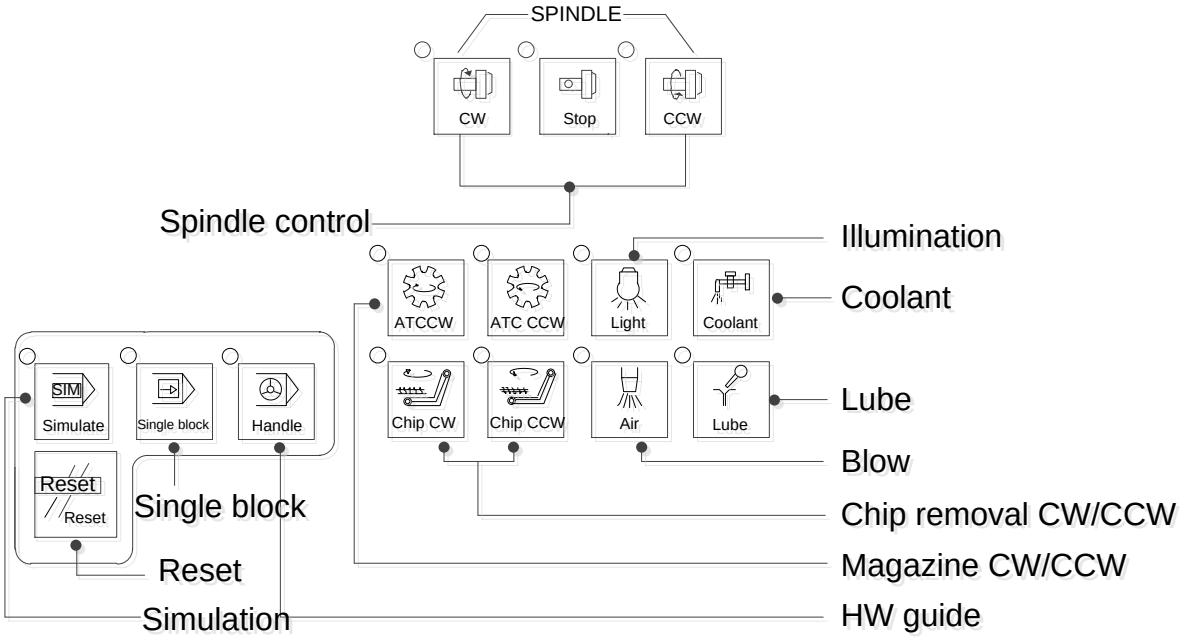


Fig. 1-8 Auxiliary function keys

10. Power ON/OFF switch, used to turn ON/OFF the power supply.

Here is rear view of WH106C operation panel. See below for detailed information of each part.

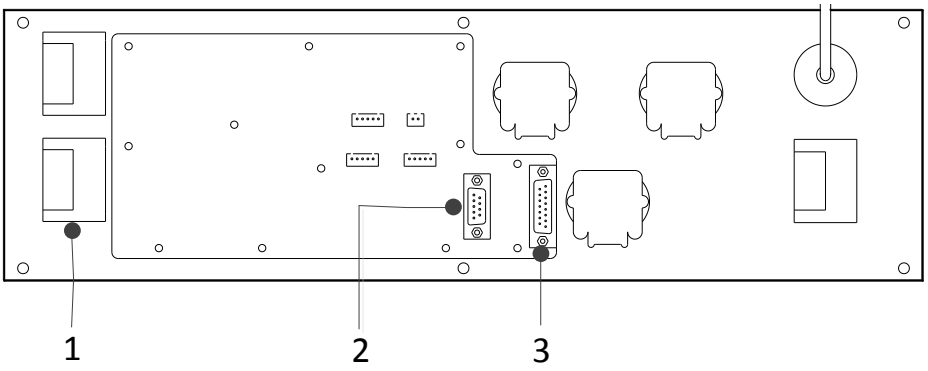


Fig. 1-9 Rear view of WH106C operation panel

1. Panel electrical switch. It is used to control the power supply of operation panel. Refer to section 2.2 for details.
2. DB9M/F cable interface. Connect WH106C operation panel with keypad panel via DB9M/F cable (40cm).
3. MPG interface. It is used for connection with MPG (also called handwheel or handle).

◆ **Illustration of WH201C Keypad Panel**

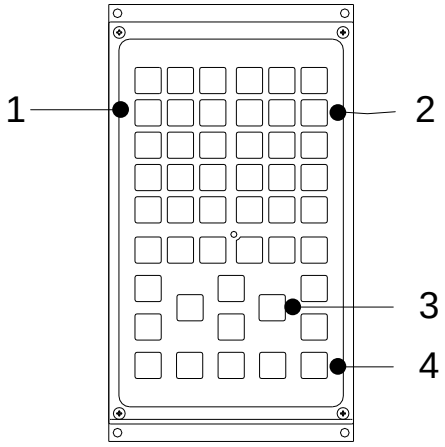


Fig. 1-10 Front view of WH201C keypad panel

1. Alphabet area, where you can enter 26 alphabetic letters. Together pressing [Shift] key and alphabet key or double pressing the alphabet key can input the letter on the upper-left of the key.
2. Numeric keys area, where you can enter number or sign. Together pressing [Shift] key and number key can input the sign on the upper-left of the key.
3. Direction keys area, also called arrow keys, including Up (↑), Down (↓), Left (←) and Right (→) keys. Besides, you can locate the cursor to the beginning or end by pressing [Home] or [End] key directly.
4. System operation keys area. There are 7 keys, including [PgUp], [PgDn], [Enter], [Del], [Select], [Caps], and [Esc], used for jumping to the previous page, jumping to the next page, confirmation, delete, selection, input of letters in upper case and exit respectively.

Next is a picture of rear view of WH201C keypad panel.

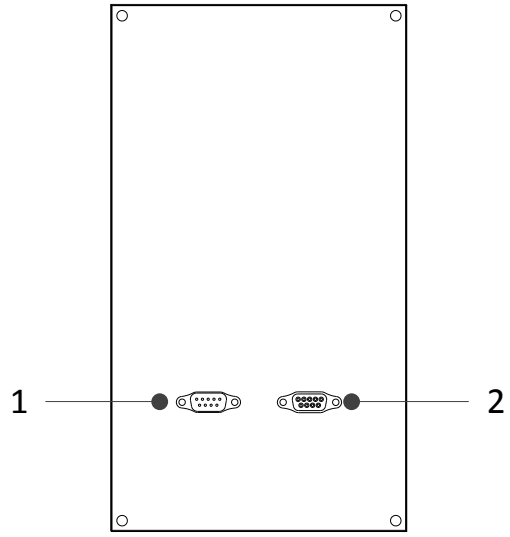


Fig. 1-11 Rear view of WH201C keypad panel

1. DB9M/F cable interface (pin), used to connect with WH106C operation panel.
2. DB9M/F cable interface (hole), used to connect with the host.

◆ **Illustration of WH108C Operation Panel**

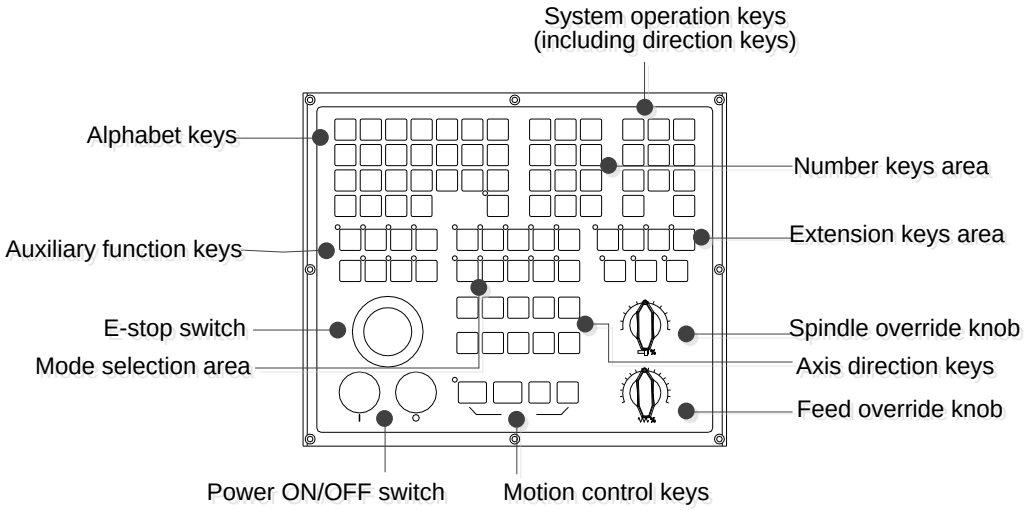


Fig. 1-12 Front view of WH108C operation panel

Most keys on WH108C and WH106C are the same, except for their layout as well as minor difference of auxiliary function keys and mode selection keys. For example, mode selection keys on WH108C correspond to the mode selection knob on WH106C operation panel.

Refer to the following pictures for concrete keys of auxiliary functional area and mode selection area.

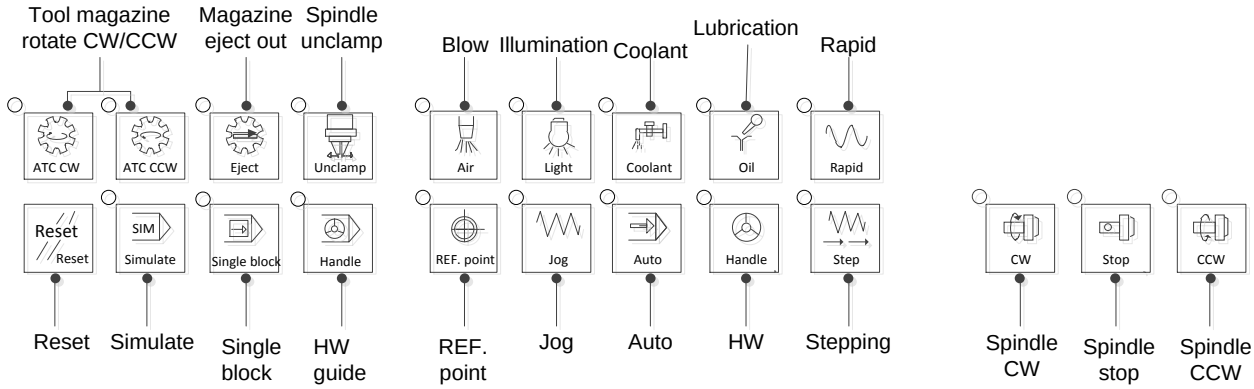


Fig. 1-13 Auxiliary function keys and mode selection keys

Here is picture of rear view of WH108C operation panel.

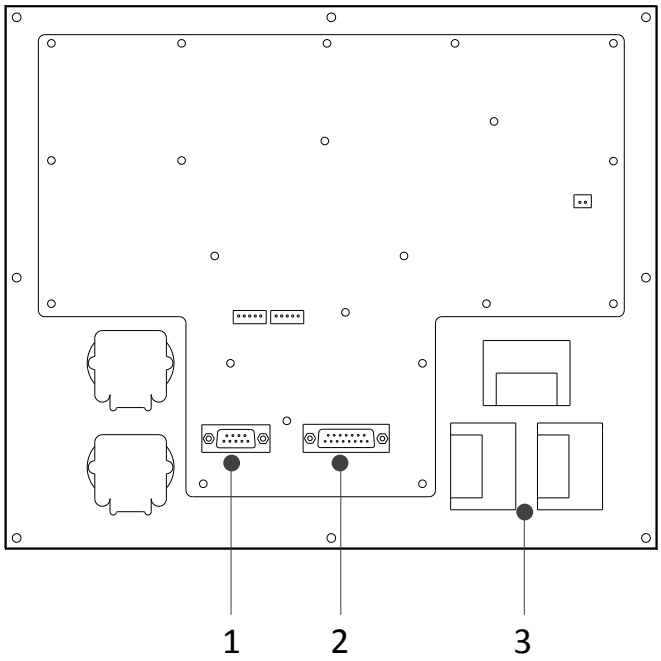


Fig. 1-14 Rear view of WH108C operation panel

1. DB9M/F cable interface, used for connection with the host.
2. MPG interface, used for connection with MPG, or handle, handwheel.
3. Panel electrical switch, used for controlling power supply for the operation panel. See section 2.2 for details.

CAUTION

1) A light on indicator on the upper-left side of a key represents for activation of the function indicated by the key.
2) When a MPG is in need, please connect it to the default interface on the back of operation panel. If a MPG is connected to MPG interface on Lambda 5S controller, you need to set parameter of manufacturer's access

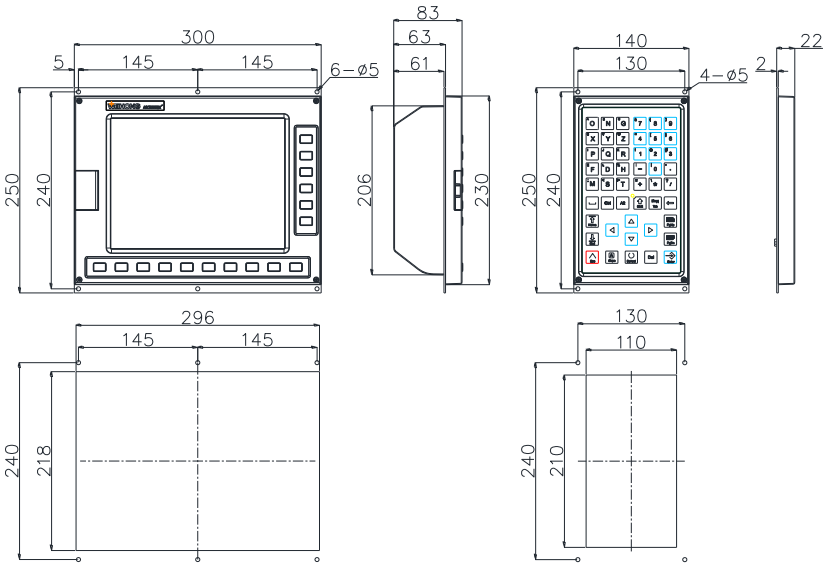
"Handwheel connection mode" to "0" (the parameter is set to "1" by default); otherwise, the handwheel cannot be enabled.

1.1.2 Mounting Dimension

1.1.2.1 Mounting Dimension of NK300CX Host

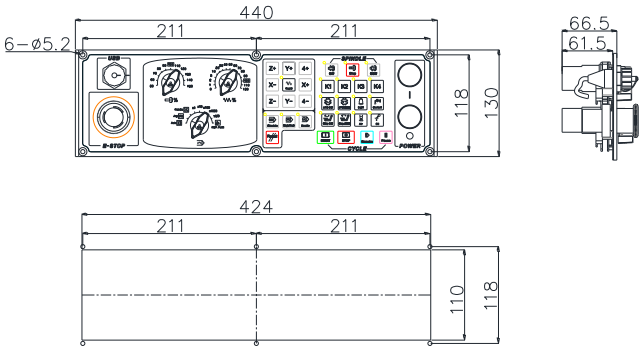
After NK300CX is installed on the machine, 100mm space should be preserved in its surrounding for wiring convenience and ventilation.

The dimensional drawing of NK300CX-H is shown as Fig. 1-15, and that of NK300CX-V is shown as Fig. 1-16.



Dimension and cut-out of the host

Dimension and cut-out of keypad panel



Dimension and cut-out of operation panel

Fig. 1-15 Dimensional drawing of NK300CX-H

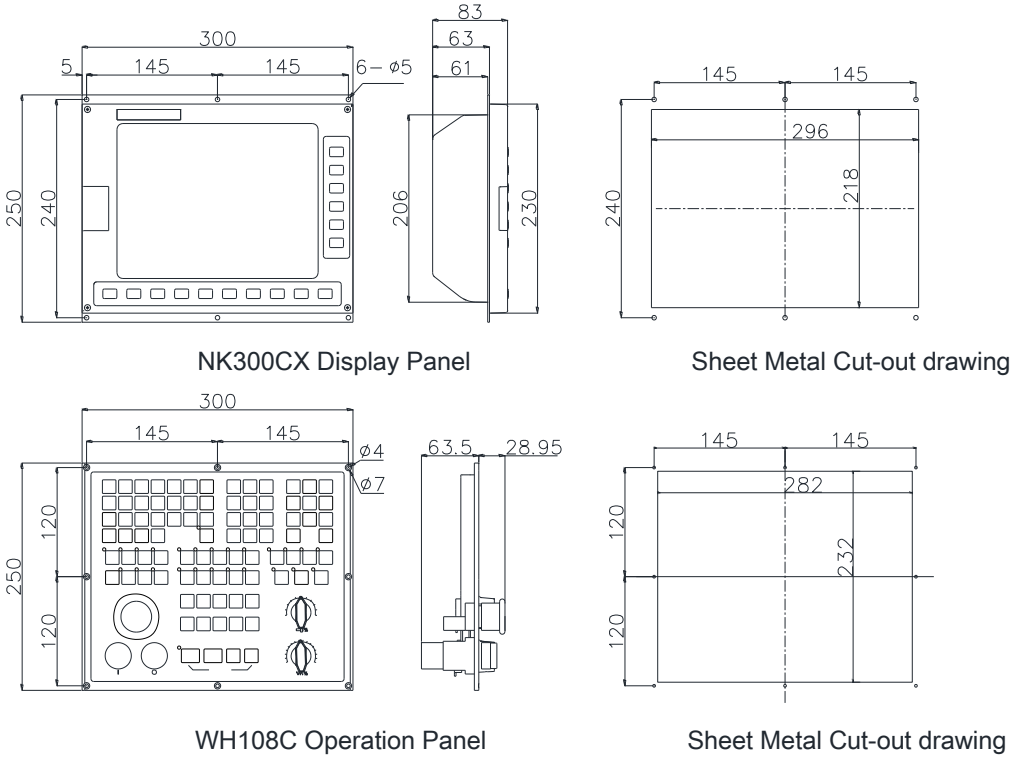


Fig. 1-16 Dimensional drawing of NK300CX-V

1.1.2.2 Mounting Dimension of Lambda Controller 5S

Here is the dimensional drawing of Lambda controller 5S.

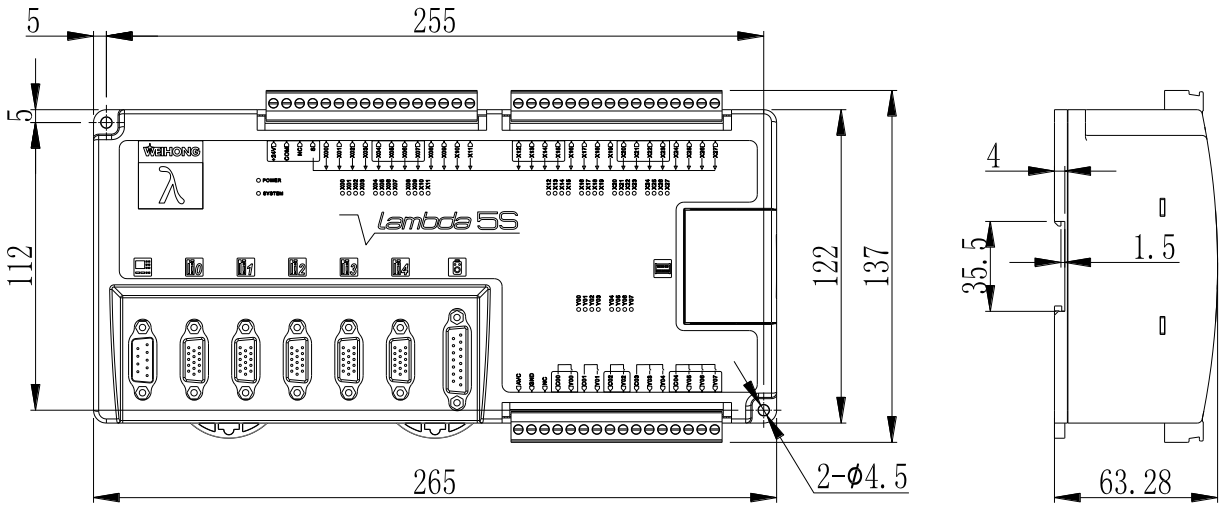


Fig. 1-17 Dimensional drawing of Lambda 5S controller

1.1.2.3 Mounting Dimension of EX31A

Here is dimensional drawing of extended terminal board EX31A1.

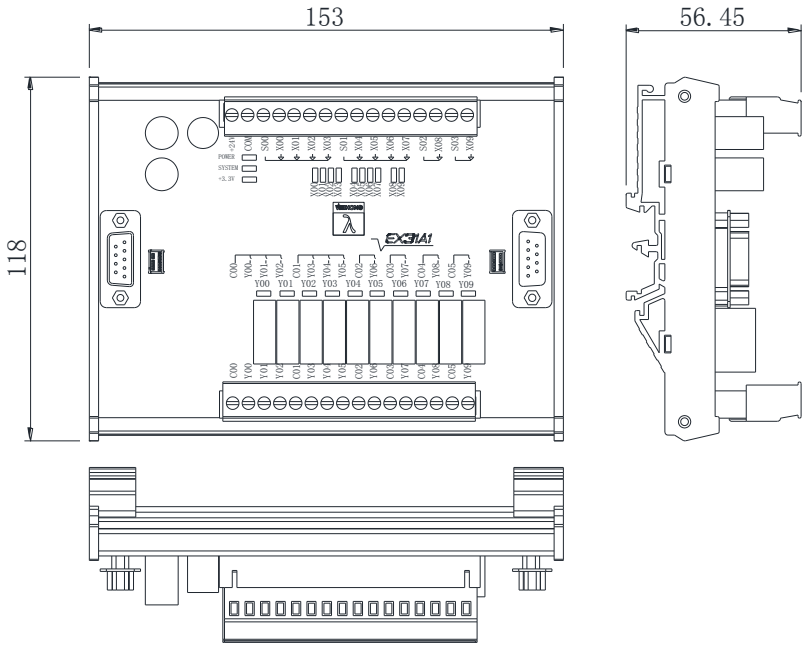


Fig. 1-18 Dimensional drawing of EX31A1

1.1.3 Overall Connection Diagram

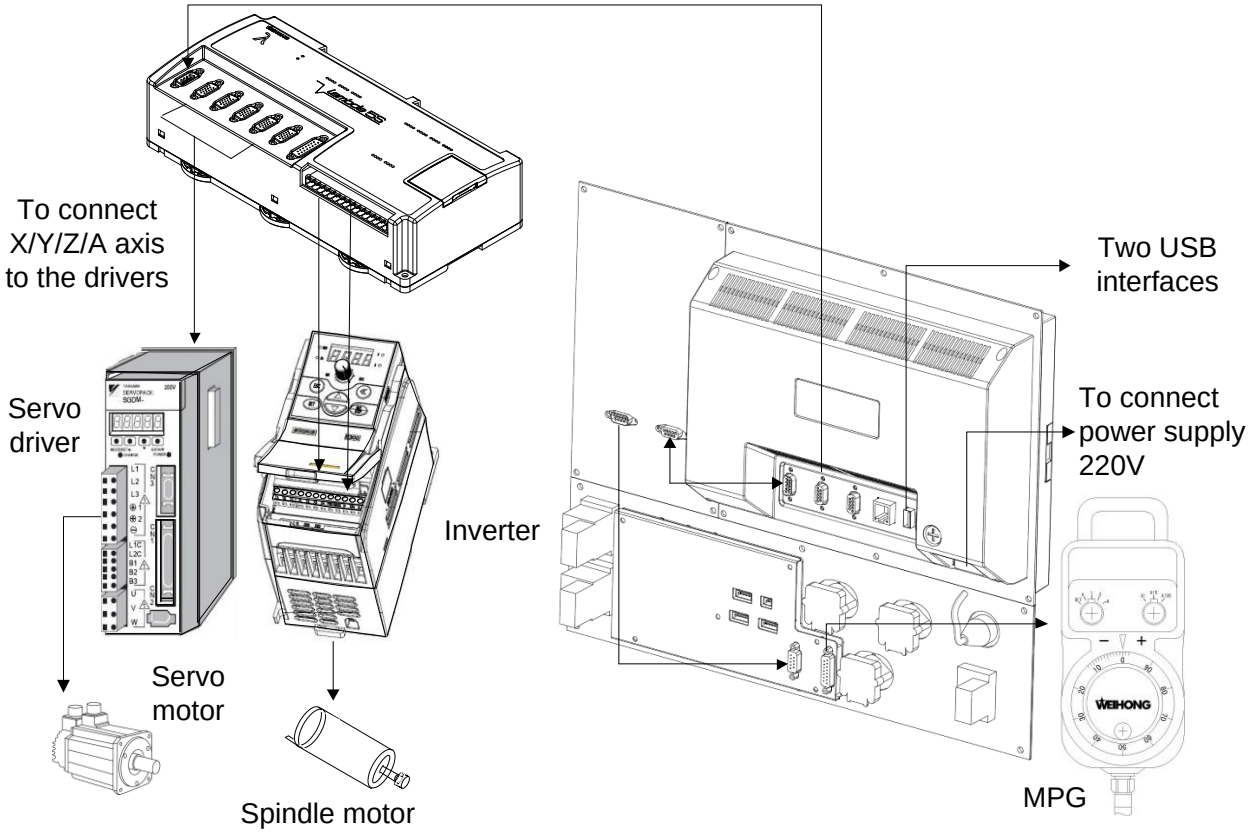


Fig. 1-19 Overall connection diagram of NK300CX-H

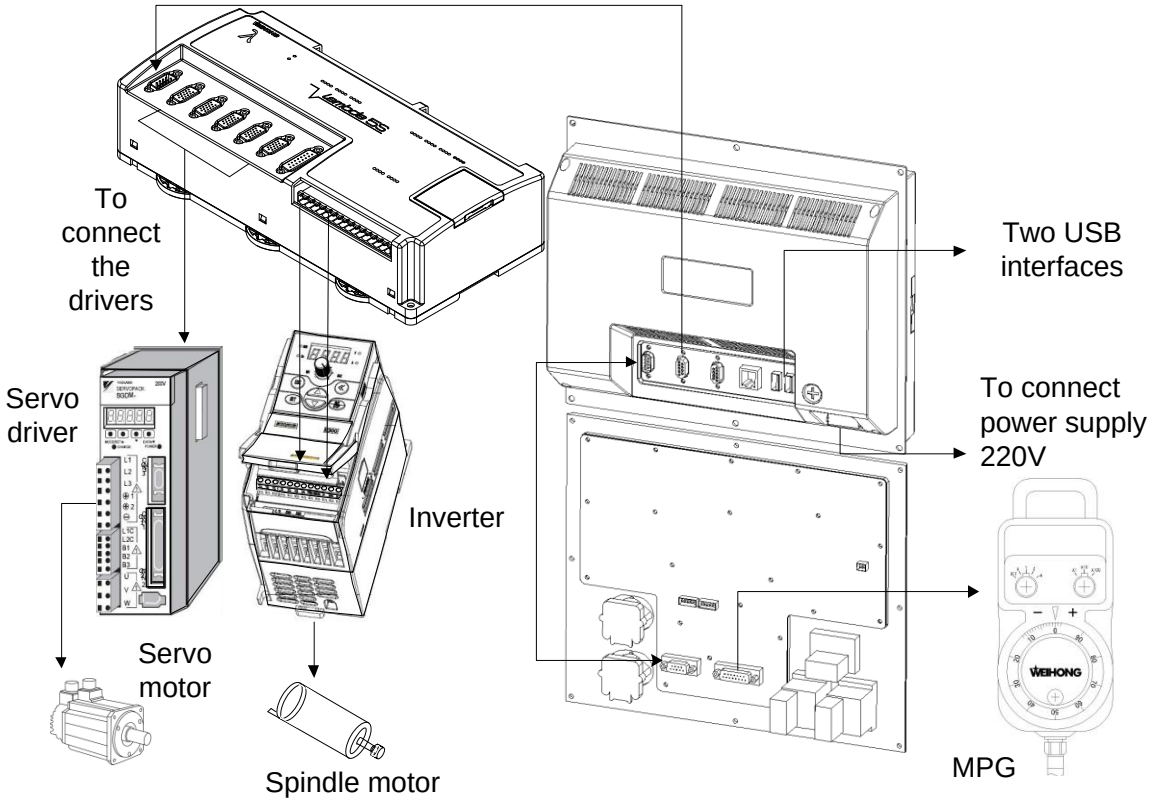


Fig. 1-20 Overall connection diagram of NK300CX-V

1.2 Software

Based on embedded platform, software of NK300CX system consists of integral software and multi-Z axes software. The former combines configurations of 3 axes, 4 axes, 5 axes as a whole. Furthermore, 3 axes configuration can be divided into three types, namely, standard, double Y and rotary table. 4 axes configuration is divided into three types, namely, standard A-type, standard B-type and standard C-type. 5 axes configuration is divided into three types as well, that is, standard AB-type, standard BC-type and standard AC-type. All types under different configurations can be switched in the software, making NK300CX a multi-functional and versatile system. Multi-Z axes software includes linkage configuration and alternative configuration, which is used for motion control of multi-Z axes.

Please note that contents in this manual is the introduction to integral software, taking standard type under 3 axes configuration as examples, if there is no special explanation. Special introduction to multi-Z axes software is presented in exclusive chapters.

Software user interface or HMI is composed of 6 functional areas, which can be switched by 6 functional keys on the right side of the host. Here is the layout of main interface in auto mode, as shown below. Refer to chapter 3 for detailed introduction to operations of each function.

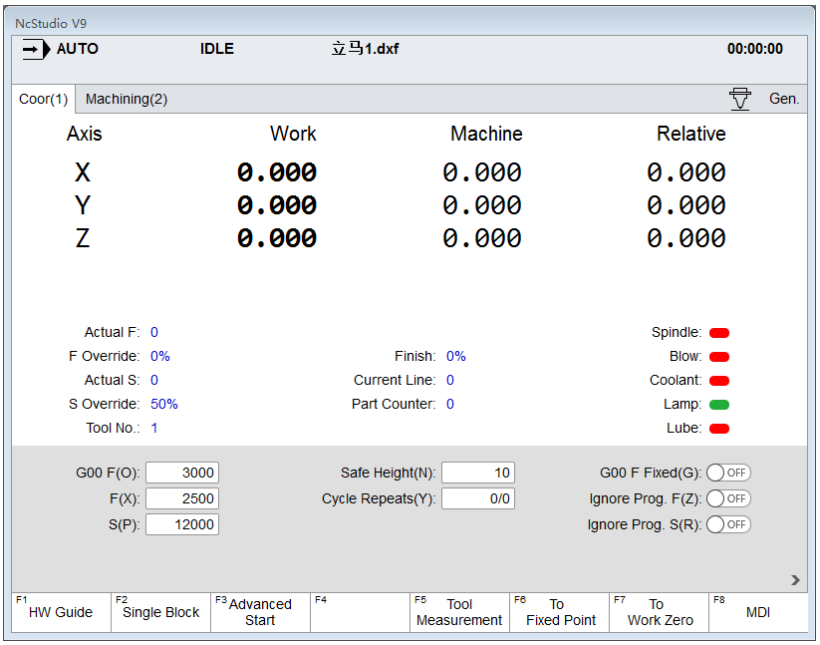


Fig. 1-21 Example of main interface of standard type of 3 axes software

Following is brief introduction to six functional areas.

◆ **Machining functional area**

「Coor」and「Machining」interfaces are folded in this area, where you can set frequently-used parameters, conduct frequently-used operations (e.g. returning to the machine origin, tool calibration, returning to the fixed point, etc.) and auxiliary functions (e.g. HW guide, single block and selective machining, etc.), simulate and obtain machining-related information.

◆ **Advance functional area**

「Coor Manager」and「Tool Manager」interfaces are folded in this area, where you can set workpiece offset and public offsets values, and access tool management.

◆ **Program functional area**

Five interfaces are folded in this area, and they are 「Local」,「USB」, 「Network」, 「Wizard」and「History」interfaces. In this area, you can operate on various program files, both stored in the system and in removable flash disk as well as on the network. In addition, you can load file into the system or track the history.

◆ **System functional area**

「System」and「Computer」interfaces are folded in this area, where you can access functions related to registration, maintenance and network setting. Besides, such information as software version, serial No. of the board card and network setting can be obtained in this area.

◆ **Parameter functional area**

Four interfaces, 「Machine/Controller」,「Axis」,「Personalized」and「Screw Error Comp」, are folded in this area, where you can set and check settings of parameters of different accesses.

◆ Diagnosis functional area

Four interfaces, 「Alarm」「Log」「Port」and「Diagnosis」, are folded here. You can check information of alarm events, warning, logs, ports, feedback pulses and coordinates, etc.

2 Wiring

2.1 Signal Types

The signal types of NK300CX system can be divided into the following 4 types: binary input signal, relay output signal and differential output signal and analog signal.

2.1.1 Binary Input Signal

Binary input signal is active low/high. Conducting to GND (i.e. grounding signal) in NO connection means signal detected, while disconnecting with GND in NC connection means signal detected.

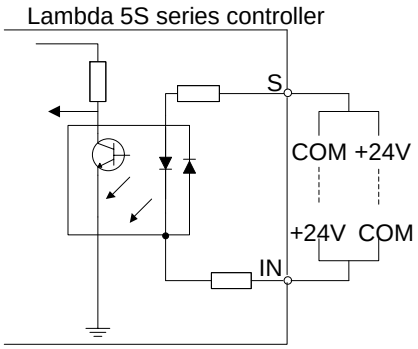


Fig. 2-1 Connection of binary input and mechanical switch

CAUTION

NK300CX system supports inputs active high/low. When the common port S on the Lambda 5S controller is connected to COM, inputs are active high after they are connected to +24V; when connected to +24V, inputs are active low after they are connected to COM.

2.1.2 Relay Output Signal

The outputs on the Lambda 5S controller are relay outputs, and the relay output contact points have load capacity—7A/250VAC and 7A/30VDC, to control 220V AC load of low power. If high power load is needed, a contactor can be used. See Fig. 2-2.

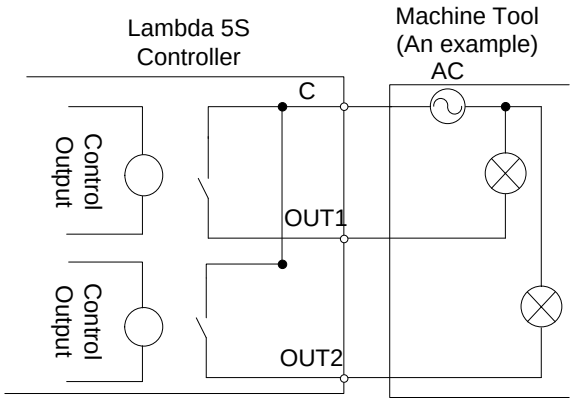


Fig. 2-2 Connection of relay output and contactor

2.1.3 Differential Output Signal

Pulse command format to control driver motion is pulse + direction, negative logic. The maximum pulse frequency is 1MHz. See Fig. 2-3 for pulse mode.

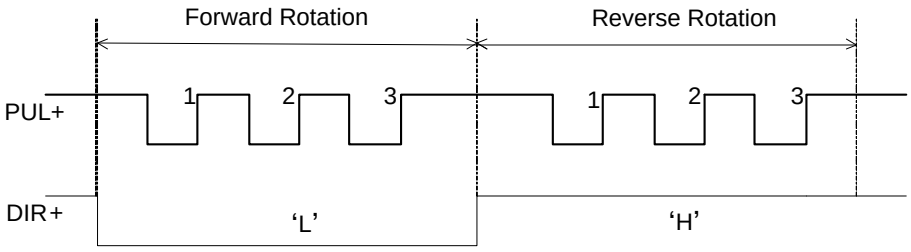


Fig. 2-3 Pulse command output mode

See Fig. 2-4 for differential signal output mode.

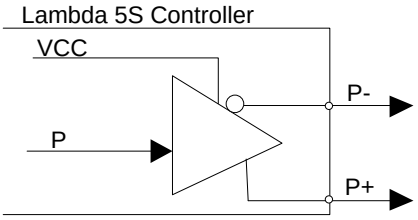


Fig. 2-4 Pulse command output circuit

2.1.4 Analog Output Signal

SVC port, externally connected with the inverter analog voltage frequency command input port, can output voltage controlled from 0V to 10V. And it can control inverter frequency by voltage change in order to master spindle speed.

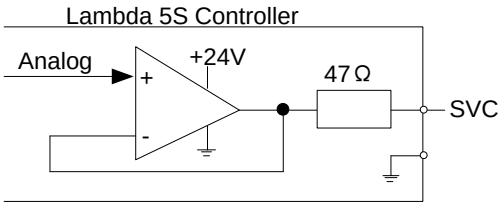


Fig. 2-5 Analog output signal circuit

2.2 Wiring Diagram of Electrical Switch on Operation Panel

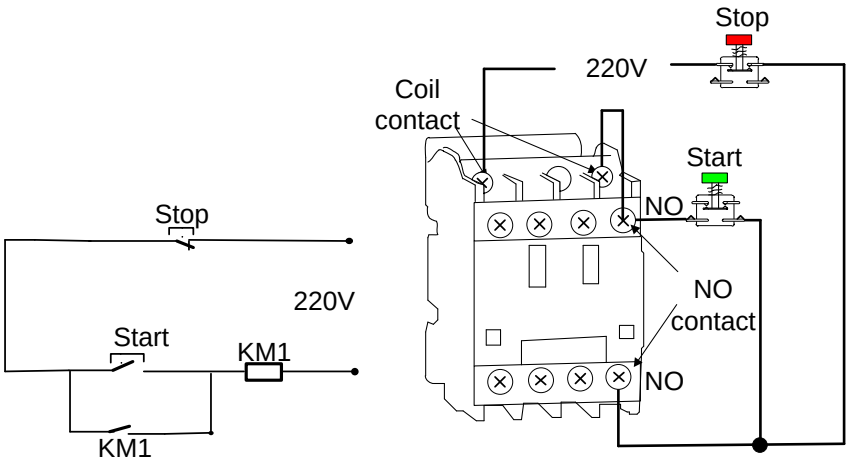


Fig. 2-6 Wiring diagram of electrical switch on operation panel

2.3 Pin Definition and Wiring Specification

2.3.1 Driver Interface Definition

NK300CX system provides 4 pulse feed driver interfaces. The type of the 4 interfaces is 15-pin D-type socket (DB15 pins). The pins definition as follow:

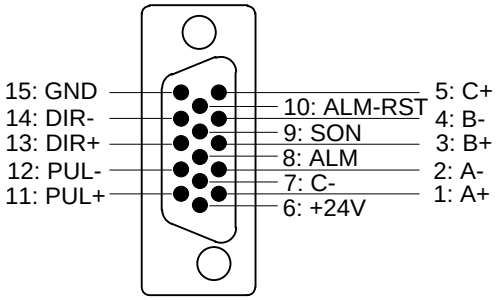


Fig. 2-7 Driver interface definition

Table 2-1 Driver interface definition

Name	Definition	Input /Output	Description
A+, A-	Feedback signal of encoder phase A	Input, differential signal transmission mode	Receive the differential output from encoder signal (phase A, B, C) of driver frequency divider (equaling to RS422).
B+, B-	Feedback signal of encoder phase B	Input, differential signal transmission mode	
C+, C-	Feedback signal of encoder phase C	Input, differential signal transmission mode	
ALM	Driver alarm signal	Input	When breakdown occurs in driver, the output (transistor) will be closed or disconnected.
SON	Servo ON signal	Output	This signal is used for opening (power on) and closing (power off) servo motor. When this signal is connected to COM-, dynamic brake will be released and thus the driver is allowed to work (servo enabled).
ALM-RST	Driver alarm clear signal	Output	This signal is used for alarm/warning status clear, and can only remove the alarms that can be removed.
PUL+, PUL-	Pulse output	Output, differential signal transmission mode	
DIR+, DIR-	Direction output	Output, differential signal transmission mode	
+24V	DC 24V power	Output	Connected to driver

CAUTION

SON signal will be effective in 2 seconds after connecting of power supply. Don't try to drive the motor through the external servo ON or servo OFF drive signal at any time, since the software will control the power-up state of the servo motor.

2.3.2 Handwheel Interface Definition

NK300CX can be externally connected to a manual pulse generator (MPG, or called handwheel). The interface consists of DB15-pins dual-in-line holes, and the pins definition is as shown below.

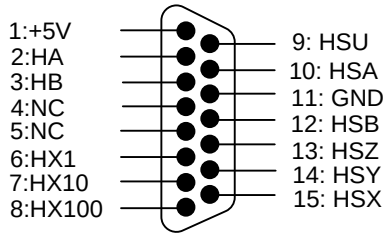


Fig. 2-8 Handwheel interface definition

Table 2-2 Description of handwheel interface

Pin No.	Definition	Description
1	+5V	Power on handwheel
2	HA	Encoder phase A signal
3	HB	Encoder phase B signal
4	NC	
5	NC	
6	HX1	Selection of X1 override
7	HX10	Selection of X10 override
8	HX100	Selection of X100 override
9	HSU	Selection of the 4th axis
10	HSA	Selection of the 5th axis
11	GND	Digital ground
12	HSB	Selection of the 6th axis
13	HSZ	Selection of Z-axis
14	HSY	Selection of Y-axis
15	HSX	Selection of X-axis

2.3.3 USB Interface

There are two USB interfaces at the back of NK300CX host, another one on the front for external connection of an USB device (E.g. U disk).

3 Software Operation

3.1 Debugging Steps

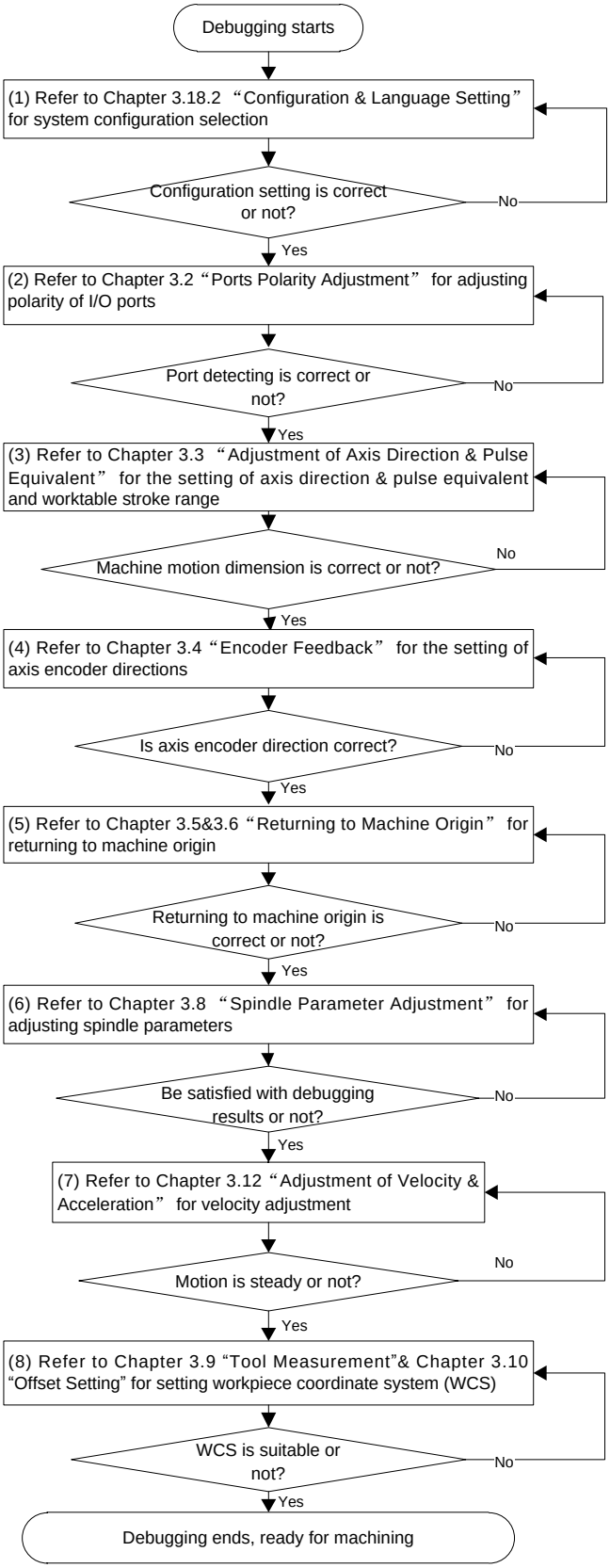


Fig. 3-1 Debugging Steps

3.2 I/O Ports Polarity Adjustment

The polarities of input/ output ports in the software are specified in terms of the switch type: the polarity of normally closed switches should be “NC”; the polarity of normally open switches should be “NO”. On the software interface, the ports with preceding filled dot ● are input ports, while the ones with hollow point ◎ are output ports.

After the connection of a machine tool and power on, the dots should be in red in front of reference point, E-stop, cycle start, cycle stop and tool sensor signal indicating these signals are invalid, or it is necessary to check whether the connection is correct. If there is no problem with the connection, the polarity of the corresponding port should be changed.

The method of modifying polarity: press key  to access functional area [Diagnosis] , then press key “3” to enter interface 「Port(3)」. Select the target I/O port for modification by pressing key “↑” and “↓”, and then press F4 to modify the polarity of the port. Restart to validate the modification.

Interface 「Port (3)」 is as shown in Fig. 3-2, and some function screens need password before operation, such as [Test On], [Test Off], [Cancel All Tests], [Invert Polarity], [Set Port Filter Time] and [Port Protection].

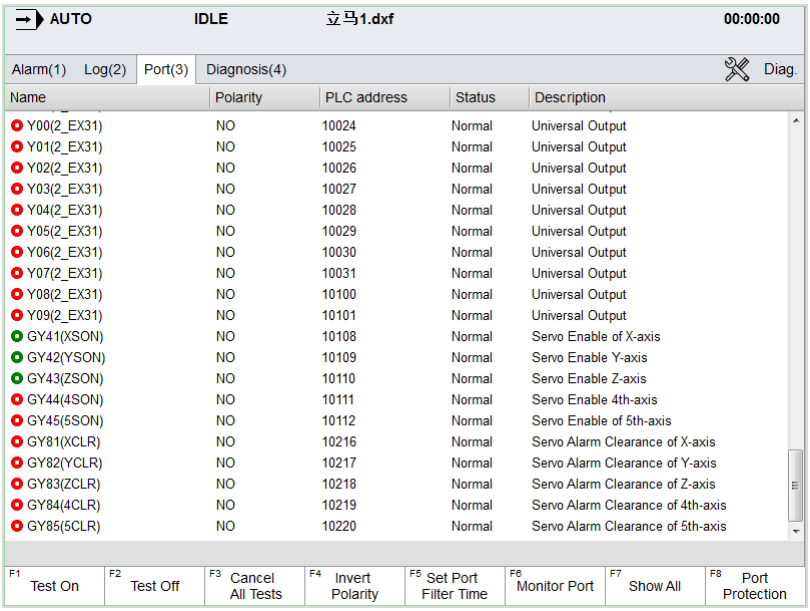


Fig. 3-2 Interface「Port(3)」

◆ Test On/Off

The shortcut keys are “F1” and “F2” respectively, which are only available on interface 「Port(3)」.

Press F1 or F2 to make the indicator light before the port selected shift between green and red. Green light means there is signal in the port; red light means there is no signal in the port.

This group of keys is mainly used for simulating hardware signal, which is for simulation test.

CAUTION

The indicator lights before ports are slightly different in test mode and in practice:

Green light in test mode:  Red light in test mode: 

Green light in practice:  Red light in practice: 

◆ **Cancel All Tests**

Press F3 to cancel simulation test and signals to replace analog signals with real hardware signals.

◆ **Invert Polarity**

Press F4 to change port polarity between NO and NC.

The polarities of feedrate override, spindle override, mode switch, handwheel and encoder zero should be “NO”.

Except for particularly defined ones, the polarities of output ports are generally “NO”.

◆ **Set Port Filter Time**

Pressing F5 can open a dialog box where you can set filter time, with unit of “ms”. A lot interference signal can be ruled out if a reasonable filter time has been set. For example, once occurrence time of the signal is shorter than the filter time, it will be defined as interference signal and be neglected.

◆ **Monitor Port**

Locate cursor to target port by pressing arrow keys, and press F6 to monitor the port. Refer to section 3.15.4 for detail.

◆ **Show All**

Press F7 to display all I/O ports, including those are hidden by default. Press F7 again to hide all I/O ports not in use.

◆ **Port Protection**

In situation that a certain machine state is required even after the software and the controller are disconnected, port protection function is the solution. You can set target ports and select the protection type before disconnection.

Press F8 to open dialog box “Port Protection”, as shown below.

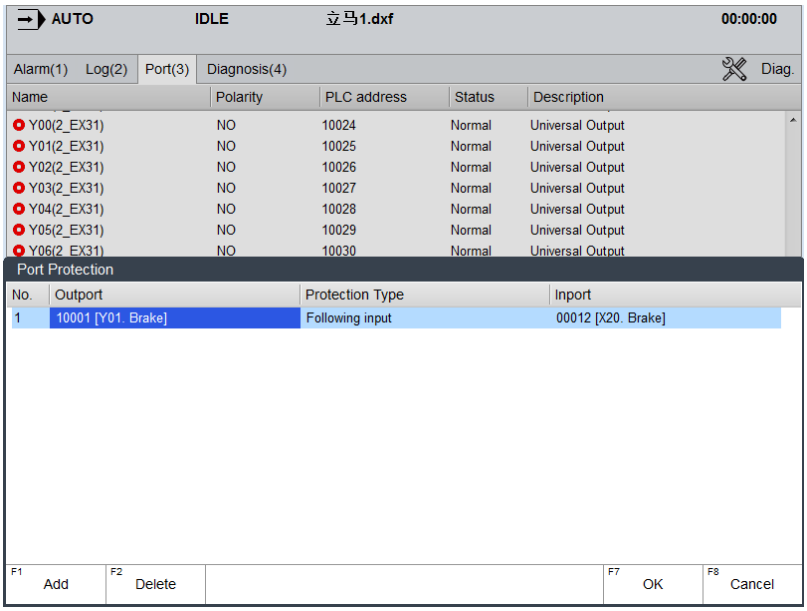


Fig. 3-3 Dialog Box "Port Protection"

◆ **Add**

Press F1 to add a new group of ports to be protected (up to 32 groups of ports can be supported). Press arrow keys to locate cursor onto target port or input box, and press Enter key to open input box, where you can input or modify the content. At last, press F7 to confirm setting or modification. In this process, following contents are required to be set.

➢ Output port (Outport for short) & Input port (Inport for short)

I/O ports on Lambda controller needs to be set. Input the PLC address of the port into input box, e.g. 10001, and press F7 for confirmation. Once set, corresponding information "10001[Y01, Brake]" will appear on interface, as shown in Fig. 3-3. For those undefined I/O ports, their status will remain the same before and after disconnection of the software and the controller.

➢ Protection Type

There are four protection types offered, all taking effect after disconnection of the software and the controller.

(1) "Enable" refers to enabling the output port.

(2) "Disable" refers to disabling the output.

(3) "Following input" refers to making the status of output ports follow those of input ports. If an input port is enabled, the output port will also be enabled; if an input port is disabled, the output port will also be disabled.

(4) "Reverse input" refers to making the status of output ports opposite to those of input ports. If an input port is enabled, the output port will be disabled; if an input port is disabled, the output port will be enabled.

◆ **Delete**

Press F2 to delete the ports which need no port protection.

◆ **OK**

Press F7 to confirm port protection setting.

◆ **Cancel**

Press F8 to back to the interface 「Port (3)」.

CAUTION

1) Addition, modification and deletion of port setting will be effective after the software is restarted.
2) For protection type "Enable" and "Disable", the input port is NA as default and cannot be set; for protection type "Following input" and "Reverse input", the input port should be set.
3) The input port and the output port should be on the same terminal board, and the port address should be valid PLC address.
4) Output port "Brake" must be set as protection port.
5) When output ports change, the protection type of the ports will be reset as "Enable", and the relevant input ports will be reset as NA. When protection type changes, the input port will be reset as NA while the output port remains the same.

3.3 Adjustment of Axis Direction and Pulse Equivalent

3.3.1 Axis Direction Adjustment

The first thing to do in machine debugging is to confirm the positive direction of each axis. The coordinate system of right-hand rule is as shown in Fig. 3-4.

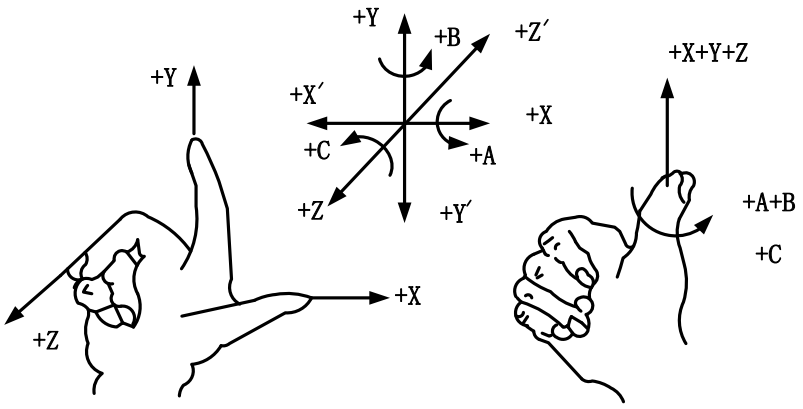


Fig. 3-4 Standard Coordinate System of Right-hand Rule

The axis directions of a machine are decided by both the type of the machine tool and the layout of each component. The basic coordinate axes of engraving & milling machines/ routers are X-, Y-, and Z-axis:

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

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

——The positive direction of Y-axis is the cutter moving away from the operator (+Y).

- **Related Parameters**

Parameter		Details	Setting Range
N10000	Axis Direction (X/Y/Z)	It specifies the motion direction of each axis.	"1" and "-1" represent the two motion directions of each axis.
Fix the positive direction of each axis following the right-hand rule, and then manually operate the machine to check if the axis moves in the correct direction. If the direction is opposite, modify the value of N10000. Taking X-axis as an example, manually move X-axis, just to find it moves oppositely, just change the X value of N10000 from "-1" ("1") to "1" ("-1").			

3.3.2 Pulse Equivalent Adjustment

Pulse equivalent (p): the moving distance of workbench or rotation degree of rotary axis per pulse sent by the CNC device, the minimum available distance controlled by the CNC system as well. Pulse equivalent can be calculated in terms of screw pitch, electronic gear ratio, mechanical deceleration ratio and other relevant info.

The smaller the pulse equivalent is, the higher the machining precision and surface quality will be. The large, the faster feedrate will be. Therefore, lower pulse equivalent should be set under condition of meeting the demand of feedrate. The relationship between maximum feedrate and pulse equivalent is as following:

$$\text{Max. Feedrate} = \text{Pulse Equivalent} \times 60 \times \text{Frequency}$$

For example, the hardware frequency of NK300CX is 1MHz, and provided the pulse equivalent is 0.001mm/p, then:

$$\text{Max. Feedrate} = 0.001 \times 60 \times 1000000 = 60\text{m/min}$$

Mechanical deceleration ratio (m/n): the ratio of reducer input speed to output speed, equal to the ratio of the teeth number of driven wheel to that of driving wheel. When applied in CNC machines, it specifies the ratio of motor speed to screw speed.

$$\text{Mechanical Deceleration Ratio} = \frac{\text{Reducer Input Speed}}{\text{Reducer Output Speed}} = \frac{\text{Teeth No. of Driven Wheel}}{\text{Teeth No. of Driving Wheel}} = \frac{\text{Motor Rotational Speed}}{\text{Screw Roational Speed}}$$

Pitch (d): The axial distance between the corresponding points of two adjacent teeth on the threads.

The calculation of pulse equivalent varies with different motor systems.

- **Stepping Motor**

In general, firstly set the subdivision and then calculate the pulse equivalent. You can also set the pulse equivalent before calculating subdivision. Their relationship can be shown as:

$$\frac{d}{p}=\frac{360}{\theta}\times x\times\frac{m}{n}$$

Hereinto, p stands for pulse equivalent, x represents subdivision of stepping motor while θ refers to stepping angle. Therefore,

$$\text{Pulse Equivalent}=\frac{\text{Screw Pitch}}{\frac{360}{\text{Stepping Angle}}\times\text{Subdivision}\times\text{Mechanical Deceleration Ratio}}$$

For instance, the selected screw lead of X-axis for a certain type of machine tool is 5mm, the stepping angle of stepping motor is 1.8 degree, with “10” subdivision and motor directly connected with screw by coupling. Thus, the pulse equivalent of X-axis is:

$$\text{Pulse Equivalent}=\frac{5\text{mm}}{\frac{360}{1.8}\times10\times1}=0.0025\text{mm/p}$$

- **Servo Motor**

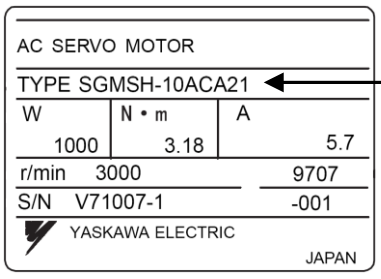
In general, set the default value of pulse equivalent (p) as 0.001mm/p and calculate electronic gear ratio (B/A). Their relationship can be shown as:

$$\text{Electronic Gear Ratio }\frac{B}{A}=\frac{\text{Encoder Resolution}}{\frac{\text{Screw Pitch}}{\text{Pulse Equivalent}}}\times\text{Mechanical Deceleration Ratio}$$

Namely, $\frac{B}{A}=\frac{F\times p}{d}\times\frac{m}{n}$

Electronic gear ratio: if servo motor makes one circle per every 5000 pulse commands sent by the system, setting electronic gear ratio of servo motor can make servo rotate twice with the same amount of pulse commands (please refer to parameters setting of the specific servo).

Please see the servo motor label plate compared to the corresponding manual to confirm its encoder resolution. A label plate of YASKAWA SGMSH type servo is as shown below, and the 4th character in motor type is the serial encoder specification, with resolution of 2^{17}, i.e. 131072.



Motor Type:
TYPE SGMSH-1 0 A C A 2 1
(The 4th Character)

The 4th Character: Serial Encoder Spec.		
Sign	Spec.	Remark
2	17-bit absolute	Standard
C	17-bit incremental	Standard

Fig. 3-5 Servo Motor Brand-encoder Resolution

For instance: (an example of YASKAWA servo) screw pitch of a certain type of machine is 5mm, with 17

bit encoder resolution, “0.0002mm/p” pulse equivalent and “1:1” deceleration ratio.

$$\text{Electronic Gear Ratio} = \frac{PN202}{PN203} = \frac{2^{17}}{5/0.0002} \times 1 = \frac{131072}{5/0.0002} = \frac{16384}{3125}$$

The pulse equivalent of rotary axis refers to the rotation degree of the axis clamping the workpiece corresponding to each pulse. The rotated degree of workpiece per revolution of motor equals to screw pitch.

- **For Stepping Motor**

$$\text{Pulse Equivalent} = \frac{360}{\frac{360}{\text{Stepping Angle}} \times \text{Subdivision} \times \text{Mechanical Deceleration Ratio}}$$

- **For Servo Motor**

$$\text{Electronic Gear Ratio } \frac{B}{A} = \frac{\text{Encoder Resolution} \times \text{ Pulse Equivalent}}{360} \times \text{Mechanical Deceleration Ratio}$$

- **Related Parameters**

Parameter		Details	Setting Range
N10010	Pulse Equivalent (X/Y/Z-axis)	It refers to the displacement or angle generated on the relative feed axis per control pulse.	9e-007~999

CAUTION

The setting of pulse equivalent must be matching with the electronic gear ratio of servo driver or subdivision of stepping driver.

3.3.3 Upper & Lower Limit Setting of Worktable Stroke

Worktable stroke refers to the valid machining stroke range of a machine tool in the X, Y, and Z directions, and the system will carry out soft limit in terms of this range in order to protect the machine.

- **Related Parameters**

Parameter		Details	Setting Range
N10020	Travel Limits-Negative(X/Y/Z)	It sets the machine coordinate of the allowable lower limit of worktable when the parameter N10040 is valid.	-99999~99999
N10030	Travel Limits-Positive(X/Y/Z)	It sets the machine coordinate of the allowable upper limit of worktable when the parameter N10040 is valid.	-99999~99999
N10040	Enable Travel Limits (X/Y/Z)	It sets whether to check the stroke range of worktable.	YES: Valid NO: Invalid

Parameter		Details	Setting Range
N67000 ~N67002	Negative Change Tool Travel Limits (X/Y/Z)	It sets the machine coordinate of the allowable lower limit of travel in tool change when the parameter N10040 is valid.	-99999~99999
N67010 ~N67012	Positive Change Tool Travel Limits (X/Y/Z)	It sets the machine coordinate of the allowable upper limit of travel in tool change when the parameter N10040 is valid.	-99999~99999

CAUTION

In the first setting of the upper & lower limit of worktable stroke, please verify the actually valid range of machine motion in case of accident.

3.4 Encoder Feedback

3.4.1 Encoder Feedback Function

Encoder feedback function is used to measure and give feedback to angular displacement and linear displacement of a screw servo motor. Encoder feedback can be set through modifying parameter N11000 “Encoder Feedback”.

If parameter N11000 is set as “NO”, the machine returns to machine origin without encoder feedback. Please see section 3.5.2 for the principle and process of the returning. If parameter N11000 is set as “YES”, the machine returns to machine origin with encoder feedback. Please see section 3.5.3 for the principle and process of the returning.

3.4.2 Setting Axis Encoder Direction

You can set the axis encoder direction by setting the value of parameter N11110 “Axis Encoder Dir”.

There are two methods to decide and set the axis encoder direction, namely setting via operation and setting via reasoning.

- **Setting via Operation**

Taking X-axis as an example, manually move X-axis towards positive direction, and during the process, press E-stop button. If the coordinate value after adjustment is larger than the value before adjustment, it tells that the current [Axis Encoder Dir] is correct, otherwise, incorrect.

Likewise, manually move X-axis towards negative direction, and during the process, press E-stop button. If the coordinate value after adjustment is smaller than the value before adjustment, it tells that the

current [Axis Encoder Dir] is correct, otherwise, incorrect.

It is the same operation with other axes.

- **Setting via Reasoning**

On condition that the axis direction and pulse equivalent settings are correct:

$$\text{Axis Encoder Direction} = \text{Axis Direction} \times \text{Logical Direction}$$

For example, setting X-axis direction to -1, and pulse to “Pulse + Direction, Negative Logic”. The value of parameter N11110 will be 1= [(-1)*(-1)].

Please refer to section 3.3.1 for axis direction setting, and refer to section 2.3.1 for pulse direction setting.

3.4.3 Encoder Error

Encoder error refers to the absolute difference value of pulse number sent and fed back. (E.g. value of |Un00C – Un00D| in YASKAWA servo) when the detected value is larger than the allowable value set by the parameter, the system will stop emergently and report “(X/Y/Z) Axis dynamic / static error alarm”.

The dynamic encoder error refers to the error in running.

$$\text{Dynamic Error} = \frac{\text{Motion Speed}}{\text{Position Loop Gain}}$$

Assuming that the feedrate of X-axis is 6000mm/min, or 100mm/s, position loop gain of servo driver is $100s^{-1}$, the X-axis dynamic error will be 100mm/s ÷$100s^{-1}$=1mm. If the pulse equivalent is 0.001mm/p, the dynamic of X-axis at 6000mm/min will be 1000p. At this time, if the parameter setting value is lower than 1000p, and X-axis has already returned to the REF point, the system will alarm and prompt X-axis dynamic error exceeding setting value and the X-axis will make relative adjustment.

Static error refers to the encoder error when the system is in idle (with idle time longer than 8s). It can be set through parameter N11140 “Static Tolerance”. Default is 500.

3.4.4 Setting Frequency Division Pulses of PG(X4)

Parameter N11160 “Frequency Division Pulses of PG(X4)” refers to encoder feedback pulse numbers via the frequency division of servo per revolution of motor, or encoder feedback pulse numbers when the linear axis moves a screw pitch. In actual debugging, you can set it according to the parameter setting of servo drivers. See user’s manuals of drivers of other brands.

3.4.5 Encoder Feedback Parameter Specification

- **Related Parameters(modification of all parameters except N80050 in this part needs manufacturer’s access)**

Parameter		Details	Setting Range
N11000	Encoder Feedback	Whether to enable encoder feedback function or not.	Yes: Enable; No: Disable
N11110	Axis Encoder Dir	It specifies the direction of encoder.	1: Positive -1: Negative
N11130	Check Encoder Error	Whether to check the encoder error between feedback value and output value or not.	Yes: Check; No: Not check
N11140	Static Tolerance	When the axis is steady, if the difference between the feedback value and output value is bigger than this value, alarm will occur.	1~999999
N11150	Dynamic Tolerance	When the axis is dynamic, if the difference between the feedback value and output value is bigger than this value, alarm will occur.	1~999999
N11160	Frequency Division Pulses of PG (*4)	It specifies the encoder feedback pulse number via frequency division of servo per revolution of motor.	1~999999
N80050	Print Info	It shows debug info about the process of returning to the REF point (only used for machine with encoder at present).	Yes: Show No: Not to show

3.5 Returning to Machine Origin

Origin of Machine Coordinate System (inherent coordinate system of a machine tool), also called mechanical origin, and home, is a fixed point assigned by design, manufacturing and debugging before the machine tool leaving factory. After startup of the CNC system, it is necessary to back to machine origin (home all axes) manually or automatically.

CAUTION

The below functions will not be activated until backing to machine origin completed: soft limit, setting fixed point and tool change.

3.5.1 Software Operation

This section focuses on introduction to operation procedure and software interface of returning the machine origin under three axes configuration. With four/five axes configuration activated, returning order can be customized.

3.5.1.1 The Process of Returning to Machine Origin

The processes of returning to machine origin of X, Y, Z-axis are included and identical, as shown in Fig. 3-6 (an example of X-axis).

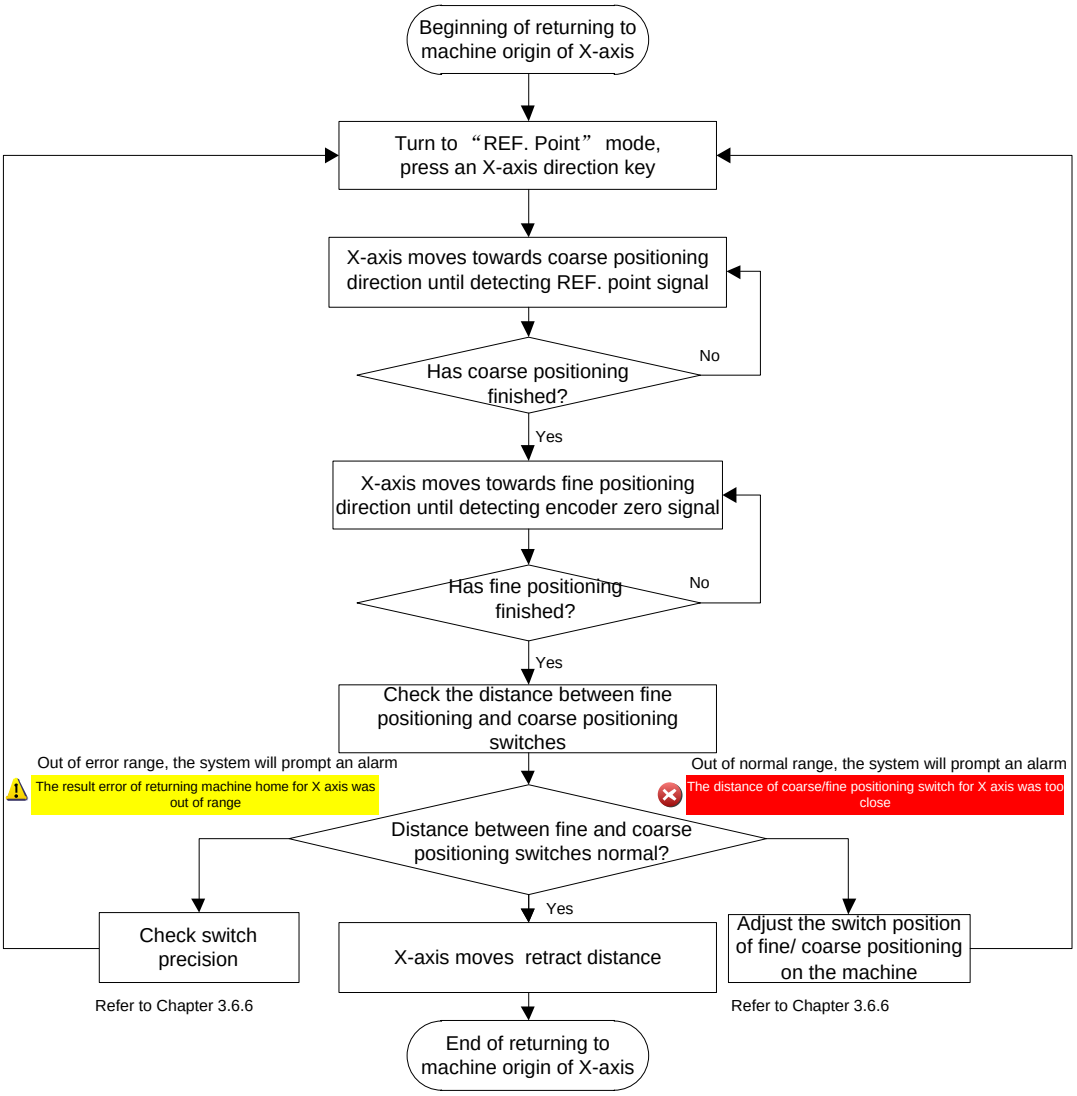


Fig. 3-6 The Process of Returning to Machine Origin (X-axis)

3.5.1.2 Returning to Machine Origin under Three Axes Configuration

Under three axes standard configuration, turn the system into REF point mode, press key  to enter interface 「Coor (1)」, as shown in Fig. 3-7.

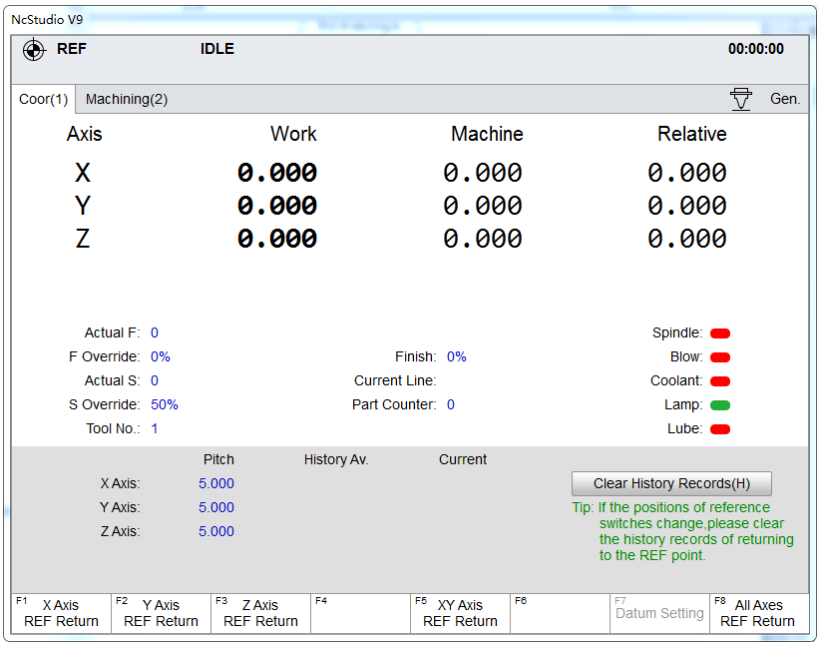


Fig. 3-7 Three Axes Standard Configuration—Returning to REF Origin

- **X-axis/Y-axis/Z-axis/XY axes Returning to Machine Origin**

Press F1/F2/F3 to return a single axis to the REF point at a time. Press F5 to return X-axis and Y-axis to the REF point at the same time.

The system entitles Z-axis the highest priority in returning REF point by default. If Z-axis is returned first, a prompt box will pop up, as shown in Fig. 3-8. Select “No” to exit the operation and “Yes” to make the selected axis return to the REF point.

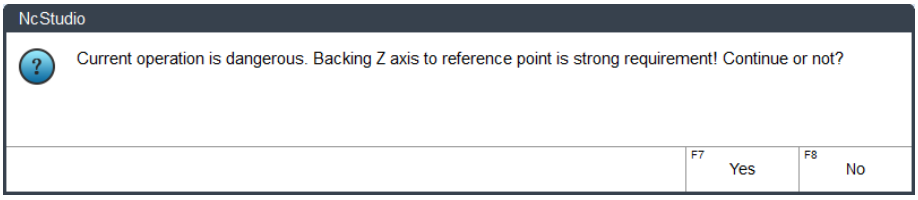


Fig. 3-8 Dangerous Operation Prompt

- **All Axes Returning to the Machine Origin**

Press F8 to execute operation “All axes REF Return”. For safety, Z-axis will return to REF point before other axes return to REF point.

3.5.1.3 Personalized Setting of Axes Order in Returning to REF Point

Compared to three axis configuration, four axis configuration and five axis configuration have one more function--personalized setting of axes order in returning to REF point. There are two flexible ways to return to the REF point, “Set Single REF Return” and “Set All REF Return”. You can customize the axes order in returning to REF point according to your needs. An introduction to the function under four-axis A model configuration is as follows.

In reference point mode, press key  and key “1” to enter interface 「Coor(1)」 under functional area [Machining], as shown in Fig. 3-9.

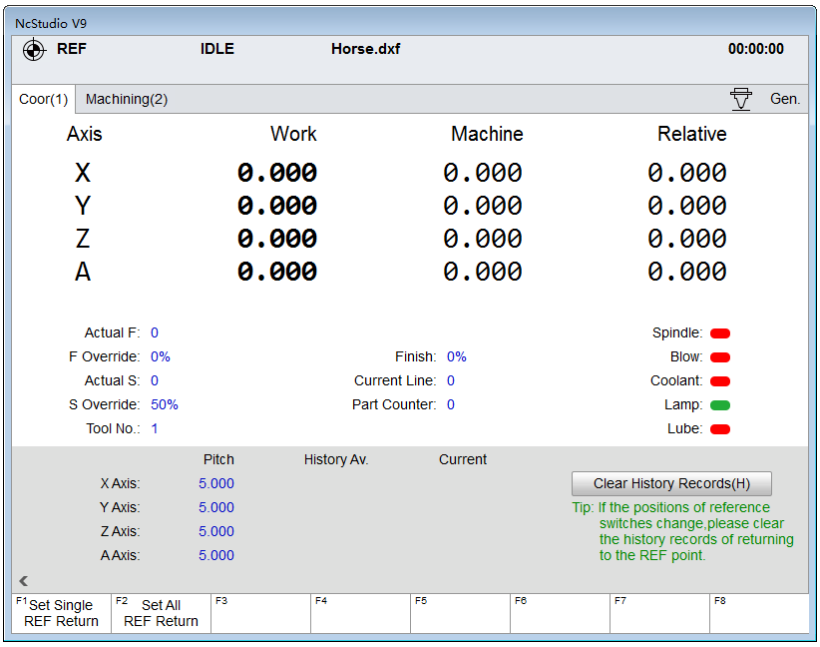


Fig. 3-9 Interface 「Coor(1)」 under Four Axis A model configuration

- **Set Single REF Return**

Press F1 to open dialog box “Set Single REF Return”, as shown in Fig. 3-10. 8 lines of order rules are supported in this box. And symbol “>” represents that axis at the left of the symbol obtains higher priority than that at the right in returning to REF point. For example, Z>X means that Z-axis has higher priority than X-axis have in returning to REF point.

Press the direction keys or key “Select” to choose an option from options “X”, “Y”, “Z”, “A “ and “NA” in the input boxes in a rule. Please note that axis options are different under different configurations.

If you do not need any order rule, change the axis in the rule into “NA” and press F7 to cancel the rule.

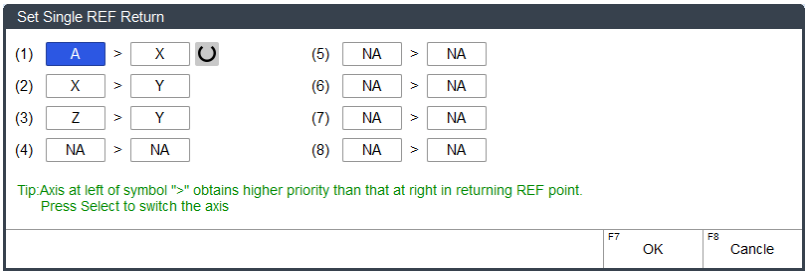


Fig. 3-10 Dialog Box “Set Single REF Return”

- **Set All REF Return**

Press F2 to open dialog box “Set All REF Return”, as shown in Fig. 3-11. Different from dialog box “Set Single REF Return”, there is only one line of order rule. Options “XY”, “Z”, and “A “ are available to be entered in the input box. Press the direction keys or key “Select” to choose an option in the input box. All the starting axes must be inputted in the order rule. Please note axis options and number of axes are different under different configurations.

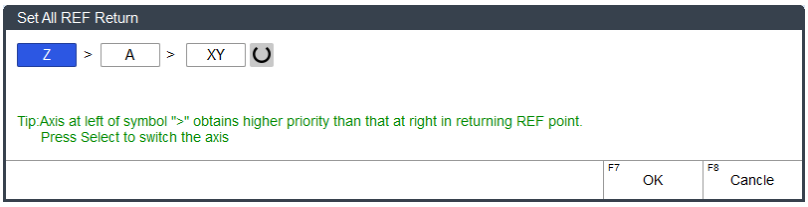


Fig. 3-11 Dialog Box "Set All REF Return"

CAUTION

1) There is no priority between the 8 order rules in "Set Single REF Return".

2) Invalid rules will be filtered automatically by the system during setting oder rules. For example, rules such as NA>A and A>NA will be regard as invalid rules.

3) Repeated or contradictory axis in setting oder rules are not allowed. For example, X>A and A>X is contradictory , and X>X is reapted setting. Otherwise, prompt "There exists repeated or contradictory axis when setting single axis REF returning order. Please check." or "There exists repeated axis when setting all axis REF returning order. Please check." will pop up.

3.5.2 Principle of Returning to Machine Origin (without Encoder Feedback)

The encoder feedback function is involved in the system, specified by parameter N11000. The sketch map of returning to machine origin with servo motor is as below (without encoder feedback):

- **Coarse Positioning Stage**

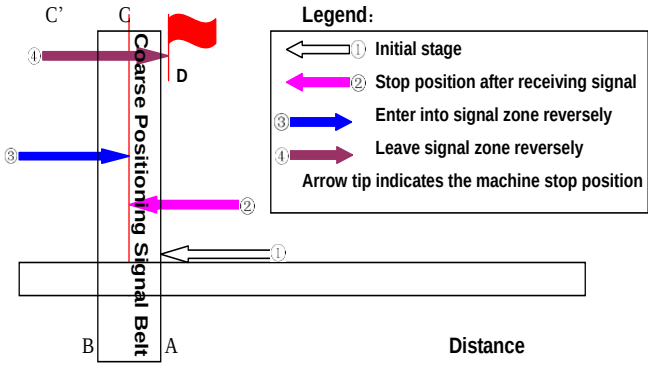


Fig. 3-12 Sketch Map of Coarse Positioning (Stopping within the Signal Belt after Receiving Coarse Positioning Signal)

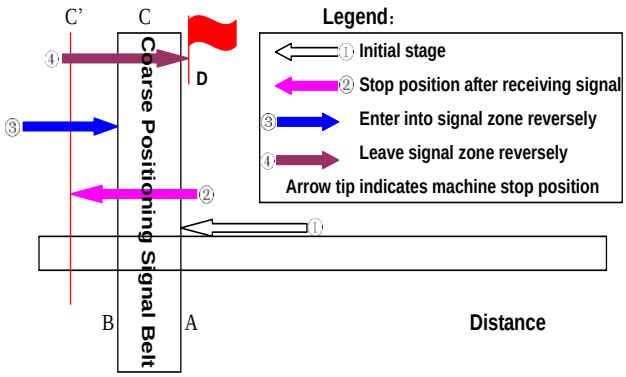


Fig. 3-13 Sketch Map of Coarse Positioning (Stopping Out of the Signal Belt after Receiving Coarse Positioning Signal)

1) When the machine keeps moving until receiving REF. point signal at place A, it should stop immediately, but it may stop at place C or C' due to time lag and inertia.
2) The machine keeps moving reversely at one third of coarse positioning speed until receiving REF. point signal (if the machine has stayed in the signal belt in the above step 1, it will make no motion in this step).
3) The machine keeps moving reversely at one-tenth of coarse positioning speed until the REF. point signal disappears (across the signal belt).
4) The machine halts at the red flag place D after the end of this stage.

- **Fine positioning Stage**

The process of fine positioning stage is identical with that of coarse positioning stage.

After coarse positioning, the machine will move to encoder zero rapidly, executing slow positioning several times.

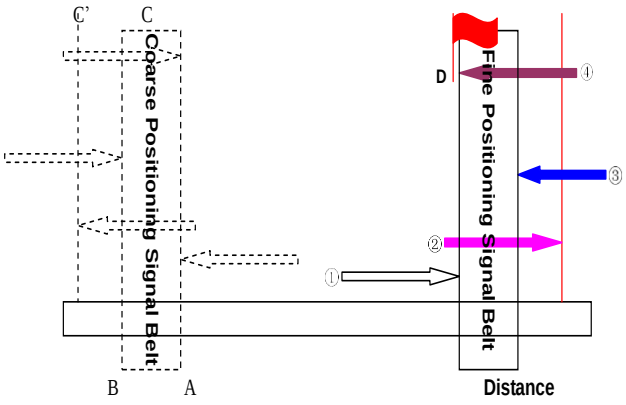


Fig. 3-14 The Process of Fine Positioning

- **Retracting Stage**

After finishing the fine positioning stage, the system will execute retracting motion once with recommended retract distance as half of the screw pitch. The sketch map is shown in Fig. 3-15.

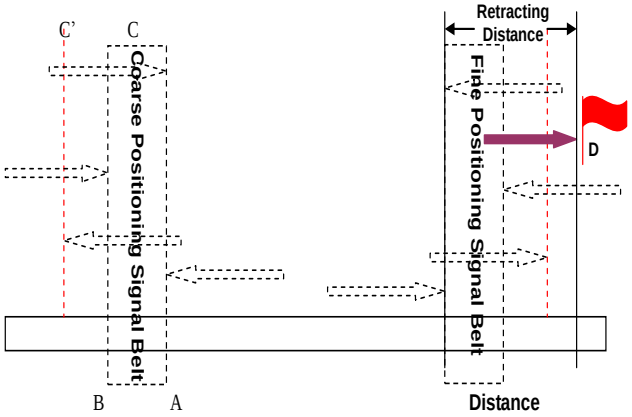


Fig. 3-15 Retracting Stage

3.5.3 Principle of Returning to Machine Origin (with Encoder Feedback)

With encoder feedback function, the system will execute coarse positioning and fine positioning only once in returning to machine origin. The retracting distance after fine positioning is the actual retracting distance adjusted in terms of actual situation. And the concrete process is as Fig. 3-16:

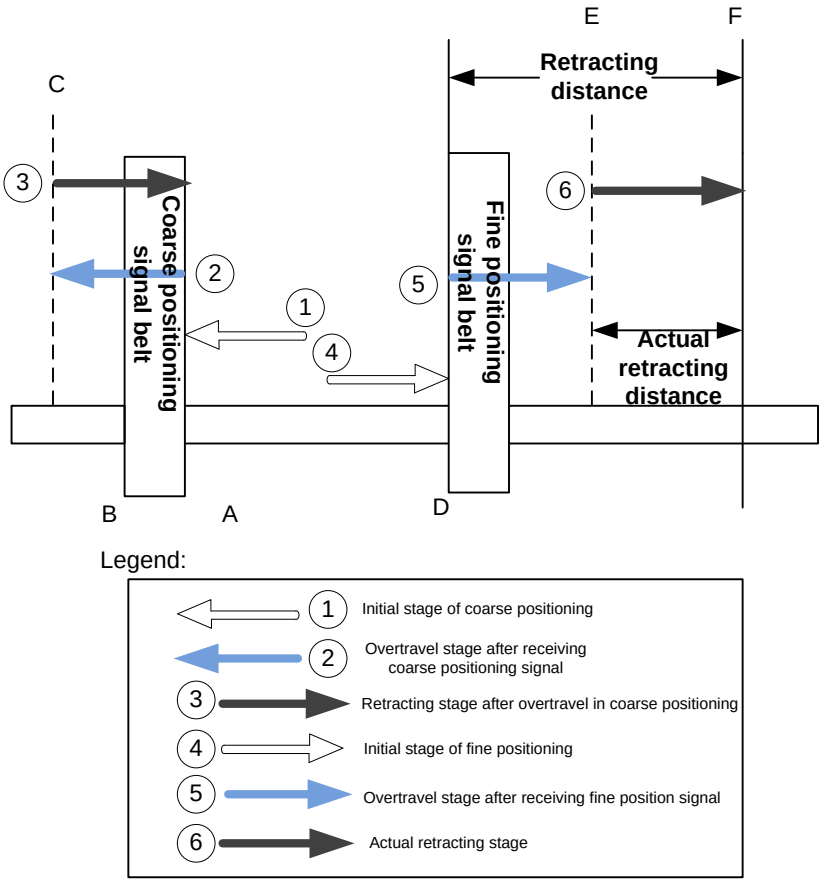


Fig. 3-16 The Process of Returning to Machine Origin

1) In coarse positioning stage, the machine tool should stop immediately at place “A” when receiving REF. point signal, but it may stop at place “C” due to over-travel caused by inertia and time-lag.
2) The machine executes retracting stage of coarse positioning.
3) In fine positioning stage, the machine tool moves reversely and should stop at place “D” immediately when receiving encoder zero signal; at the meantime, the feedback data of encoder will be latched, but the machine will generate over-travel and stop at place E due to inertia and time-lag. And the displacement between encoder zero signal and stop position (i.e. DE) is the across distance of signal deceleration.
4) In terms of the retracting distance and across distance calculated, calculate the actual retracting distance, and use this actual distance to make the machine move and stop at place F, keeping it free from the signal source.

3.5.4 Parameter Specifications

- **Related Parameters of Safe Operations:**

Parameter		Details	Setting Range
N74001	Back to REF Required	Whether backing to machine origin before machining is required or not.	YES: Required NO: Not required
Returning to machine origin before machining can avoid machining offset to ensure position precision. When N74001 is set to “YES”, if there is no backing to machine origin mark “◒”or “L” before each axis, the machine is not allowed to move until returning to REF. point is completed. N74001 can be set to “NO” when failure to return to machine origin is caused by home switch fault.			

- **Related Parameters in the Process of Backing to Machine Origin (N74090 under “Operation”, others under “Axis Parameter”)**

Parameter		Details	Setting Range
N74090	Home Latch Count	Times of fine positioning in returning to machine origin, with default setting of “1”	1~100
N74010	Machine Zero Position	Machine coordinate of machine origin, with default setting of “0	0~ Upper limit of workbench stroke (the value of N10030)(mm)
N74020	Coarse Positioning Dir.	The moving direction of machine at any point towards home switch	1: Positive direction -1: Negative direction
N74030	F in Coarse Positioning	Moving speed of machine towards home switch (coarse positioning speed)	0.001~10000
N74040	Coarse Positioning Switch Inport Addr.	The input port PLC address of coarse positioning switch of each axis	-
N74050	Fine Positioning Dir. (X/Y/Z)	The moving direction of machine at any point towards encoder zero	1: Positive direction -1: Negative direction

Parameter		Details	Setting Range
N74060	F in Fine Positioning	Moving speed of machine towards encoder zero (fine positioning speed)	0.001~10000
N74070	Fine Positioning Switch Inport Addr.	The input port PLC address of accurate positioning switch of each axis	-
N74080	Back Off Distance (X/Y/Z)	The additional moving distance after the end of fine positioning in returning to machine origin, i.e. retract distance to move away from signal sensitive zone.	-1000~1000
In order to establish a machine coordinate system (MCS) correctly for machining, at machine start-up, generally returning to reference point will be executed automatically or manually, i.e. the machine tool will return to its measuring beginning (X, Y, Z=0) to establish the machine coordinate system. Machine reference point can be coincident with machine origin (in default system setting), or not, and the distance between reference point and machine origin can be specified by parameter N74010. When home switches work normally, if the spindle moves away from home switch direction in the process of returning to machine origin (homing), the value of N74020 (coarse positioning direction), opposite to that in fine positioning stage, should be modified, please refer to question No. 2 in section 3.5.5 when the moving direction of machine is incorrect during backing to machine origin. If the speed of returning to machine origin is too low, properly adjust the value of N74030 (coarse positioning speed). “Back Off Distance” refers to a certain moving distance away from REF. point to leave the signal sensitive zone of home switches after backing to machine origin completed.			

- **Related Parameters to Detect Distance between Coarse and Fine Positioning Switches (N74120 under “Operation”, others under “Axis Parameter”)**

Parameter		Details	Setting Range
N74100	Leadscrew Pitch	For analysis of switch distance of fine and coarse positioning in backing to machine origin	0~360
N74110	Coarse/ Fine Switches Min Dist	To detect whether the switches of fine/coarse positioning are too close in backing to machine origin	0 ~ One half of screw pitch
N74120	Coarse/Fine Pos Distance Tolerance	The allowable error range by comparison of current result of backing to machine origin with history average record	0~100
Related to the specific machine tool, N74100 should be set after measured in actual operation. Too close distance between home switch and encoder zero switch may lead to deviation of one			

Parameter	Details	Setting Range
screw pitch in REF. point positioning during returning to machine origin, and the system will check if this offset is reasonable or not via parameter N74110 with setting range of "0 ~ one half of screw pitch" (unit: mm). The normal range of switch distance between fine and coarse positioning is (0+ value of N74110, screw pitch- value of N74110), and the switch distance will be automatically measured in returning to origin. An alarm of " The distance of coarse/fine positioning switch for Z axis was too close " will occur if the distance is out of the above range. To remove this error, adjust the home switch position or check if the setting of parameter N74110 is reasonable or not. With comparison between current measured value and history average value, the percentage of "(current measured value – history average value) / history average value" should be within the setting value of N74120; if not, this measured value is invalid. And the system will prompt alarm of " The result error of returning machine home for Z axis was out of range ". Click the shortcut key N " Clear History Records(N) " to clear the measured history record after changing the home switch.		

3.5.5 Troubleshooting

1. REF. point signal cannot be detected in the process of returning to machine origin. It is generally caused by home switch fault. The adjusting & debugging steps are as shown in Fig. 3-17.

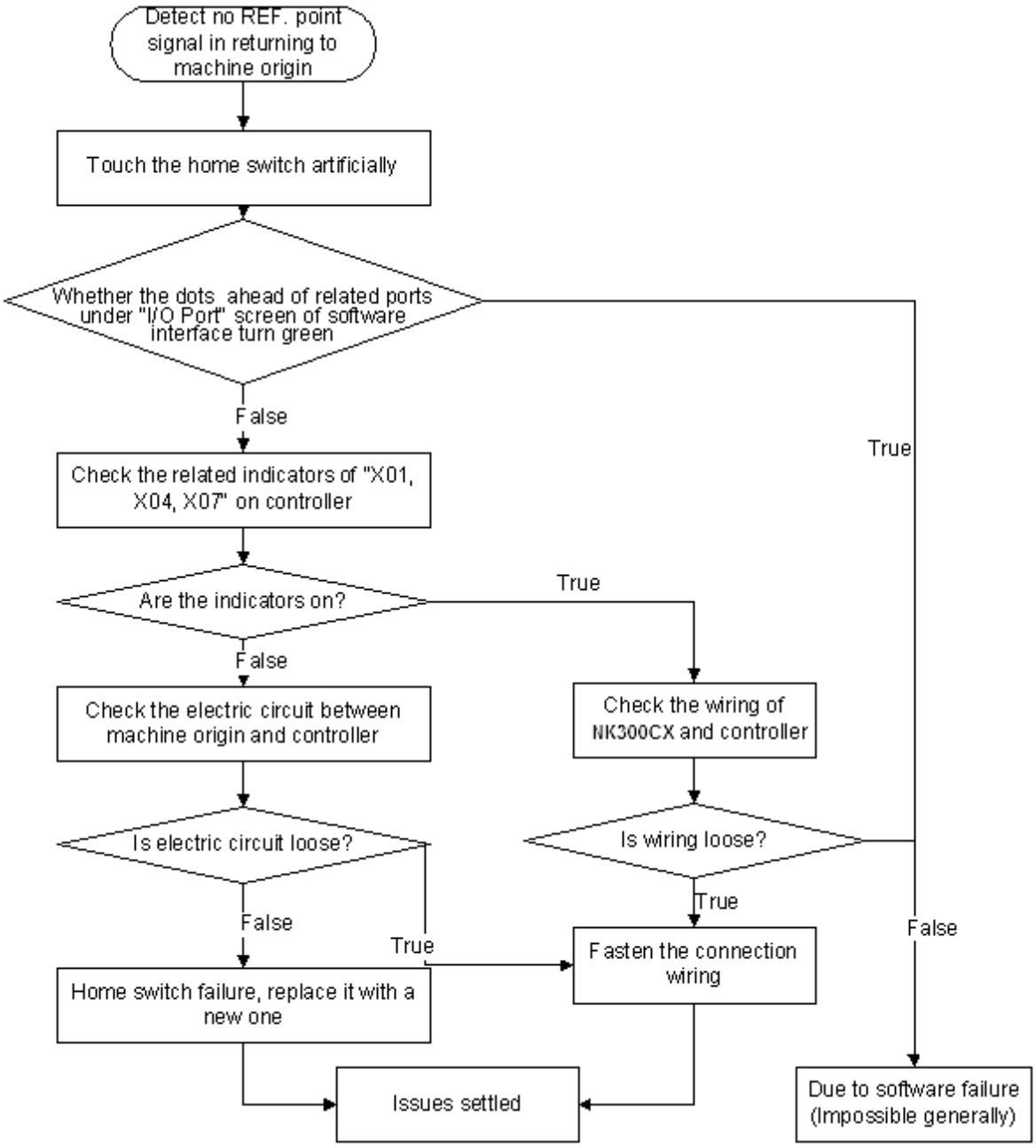


Fig. 3-17 Debugging Steps

2. Incorrect motion direction of machine in returning to machine origin may be caused by the following reasons:
   1) Incorrect polarity of REF. point signal: when the home switch is normally open, the polarity is “NO”; when normally closed, the polarity should be “NC”.
   2) Incorrect parameter setting: check the parameter N74020 “Home Search Dir”, and adjust the related parameters.
3. Too slow coarse positioning speed in returning to machine origin may be caused by the below reasons:
   1) The setting value of N74030 “Home Search Velocity” is too small.
   2) The polarity setting of REF. point signal in the software is mismatching with the home switch type. If a NC-type home switch is adopted and the polarity of REF. point signal is NO, the REF.

point signal is valid at beginning of backing to machine origin, so the machine will slowly move away from machine origin at the speed of fine positioning.

4. The distance between fine and coarse positioning is out of normal range, the system prompting an alarm "The distance of coarse/fine positioning switch for Z axis was tool close", which may be caused by too close switch distance between fine and coarse positioning, so the actual position of home switch and encoder zero should be readjusted to make the distance within the range of (0+ value of N74110, screw pitch- value of N74110).
5. The distance between fine and coarse positioning is out of the allowable error range, the system prompting an alarm "The result error of returning machine home for Z axis was out of range", with possible causes as below:

   1) The accuracy error of home switch: check home switch precision.
   2) The accuracy error of encoder zero: check whether encoder zero signal in the system is correct or not.
   3) After a home switch is reinstalled, the detecting environment changes in returning to machine origin: press the shortcut key N to clear the history record of measurement.

3.6 Returning to Machine Origin (with Absolute Encoder Function)

With absolute encoder function, on the one hand, there is no need to set returning orders of all axes in datum setting process (process of returning to the machine origin); on the other hand, in case of software restart, update, power interruption, or emergency stop and the like, there is no need to return to the machine origin again in order to set datum when saving or exporting datum information, which will simply the process and save preparation time.

3.6.1 Functional Environment

Requirements for the hardware are as follows:

(1) YASKAWA Σ-V Servo Driver

(2) Lambda 5E controller (Note that wiring diagram of Lambda 5E is the same with that of Lambda 5S)

(3) NK300CX integrated system

For servo driver, servo motor with absolute encoder should be used together. You can read nameplate of the motor to make sure absolute encoder function is supported and refer to corresponding manual to get information of encoder type. Here is an example of SGMJV motor.

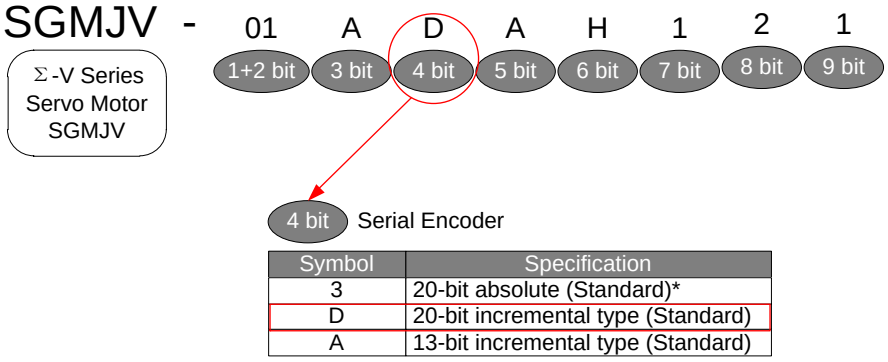


Fig. 3-18 Example of SGMJV motor nameplate

As shown above, the 4th character specifies the specification of serial encoder. You can further get encoder type by referring to the driver manual. For example, if 4th character is "3", the encoder type is absolute type, to put it in other words, absolute function can be supported.

Before enabling the absolute encoder function, you need to do following jobs first:

1) Set relevant parameters in both driver and software;

2) Check whether initialization of the absolute encoder is required.

3.6.2 Related Parameter Setting

3.6.2.1 Enable Absolute Encoder Function

- **Driver Parameter Setting**

Set value of parameter Pn002 to decide the absolute encoder as incremental type encoder or absolute type encoder. Specifically, if absolute encoder function is in need, set the second bit of Pn002 to "0", that is "n.□0□□". Note that parameter modification takes effect after re-power on the driver.

CAUTION

There are altogether 4 bits for the driver parameter, that is, bit 0, bit 1, bit 2 and bit 3 from right to left.

- **Software Parameter Setting**

After locating to the parameter setting interface, press F8 to activate manufacturer's access, and find parameter N11001 "Encoder type", set it to "1", enabling absolute type encoder; next, find parameter N11000 "Enable encoder feedback function" and set it to "Yes", enabling encoder feedback function.

3.6.2.2 Machine Position-related Parameter Setting

When first use of encoder, relationship between absolute encoder and actual machine position should be set, that is, to secure the machine origin. Once set, actual machine position can be directly read after software startup.

Concrete machine position value is decided by setting values of driver parameter Pn000 and software

parameter N11200. See below for details.

- **Settings of Driver Parameter Pn000 and Software Parameter N11200**

Setting values of Pn000 and N11200 should be matched, as follows:

(1) If driver parameter Pn000 is set to "□□□0", the motor CCW rotation is regarded as the positive direction, and software parameter N11200 should be set to "-1" accordingly.

(2) If driver parameter Pn000 is set to "□□□1", the motor CW rotation is regarded as the positive direction, and software parameter N11200 should be set to "1" accordingly.

Note that modification to Pn000 takes effect after re-power on the driver.

CAUTION

1) Viewing from load side of the servo motor, the positive direction in standard setting is counter-clockwise rotation.

2) Positive/negative direction overtravel prohibition fucntion may be changed according to the selection of motor rotational direction.

3.6.3 Initialization of Absolute Type Encoder

3.6.3.1 Situations Where Initialization Is A Must

In following situations, initialization operation to the absolute encoder is required:

(1) First time power on of the driver;

(2) Replacement of battery box for absolute encoder;

(3) Alarm "(A.810) Encoder backup" occurs;

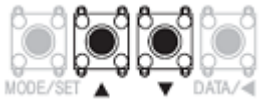
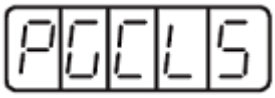
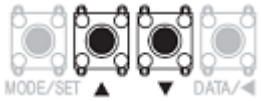
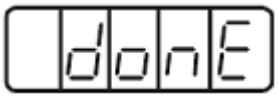
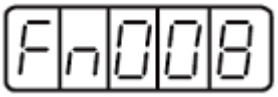
(4) Alarm "(A.820) Encoder and verification" occurs.

3.6.3.2 Steps of Initialization

Note that initialization operation to encoder can ONLY be conducted at Servo-OFF. Detailed steps are as follows:

Table 3-1 Operation steps of absolute encoder initialization

Step	Panel display after operation	Buttons used	Comments
1			Press button MODE/SET to select auxiliary function.

Step	Panel display after operation	Buttons used	Comments
2			Press button UP/DOWN to display “Fn008”.
3			Press button DATA/SHIFT for approx. 1s to display “PGCL1”.
4			Continuously press button UP until “PGCL5” appeared. Note: if there is any wrong button operations, “n0_0P” will appear for approx.1s before returning to AUX. mode. At this time, you need to start from the very beginning.
5			Press button MODE/SET. Start to set (initialize) absolute encoder. When setting (initialization) completes, “donE” will appear in flashing way for about 1s.
6			“PGCL5” will be displayed after “donE” disappeared.
7			Continuously press button DATA/SHIFT for approx. 1s, “Fn008” will be displayed.

CAUTION

After absolute encoder is initialized, datum setting must be executed again.

3.6.4 Software Operation

With REF. point mode activated, press  ==> 1 ==> F7 in turn to access the datum setting interface, as shown below.

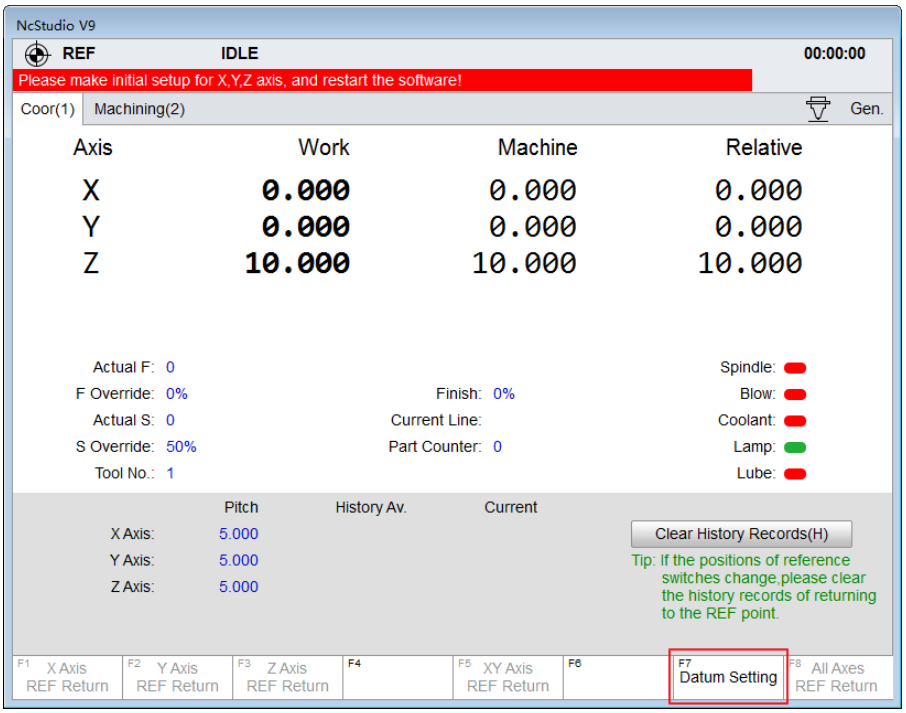


Fig. 3-19 Datum setting interface

- **Software prompt when datum not set**

If it is the first time datum setting operation is executed, prompt information in red “Please initialize X/Y/Z axis first and restart the software” will appear in info bar, as shown in above. You can follows the prompts and make datum settings. After datum is well set, red prompt will disappear, at the same time, homing symbol will appear in front of each axis, which means the system is ready for normal machining task.

- **Datum setting in X/Y/Z axis**

Manually move the axis to the fixed position on the machine, press F1/F2/F3 accordingly to set the datum in X/Y/Z axis. A dialog box prompting “Absolute position setting takes effect after restart, please restart the software!” pops up. Choose “Yes” and restart the software.

- **Export**

Previous datum setting information will lost after softwre upgrading. For this reason, you can choose import/export function to save trouble of repeated setting of machine datum (or the reference point).

After datum has been well set, press F8 to save it to the root directory of U disk.

- **Import**

When a new software has been installed, press F7 to import or load datum information which has been previously saved under root directory of U disk. At this time, a dialog box prompting “Whether to import datum file?” will appear.

If “Yes” is chosen, prompt as “Modification to absolute position initial setting takes effect after restart, please restart the software!” Follow the tips to restart the software and make modification effective.

If “No” is chosen, importation of datum information aborts.

CAUTION

1) Manufacturer's access is needed before datum setting;

2) Because of time sequence, it takes time for the software to read absolute data, possible resulting in interface choking in datum setting.

3.6.5 Parameter Specifications

After locating to parameter management interface, press F8 to activate manufacturer's access. Following parameters should be well set.

<table>
<tr><th colspan="2">Parameter</th><th>Content</th><th>Range</th></tr>
<tr><td>N11000</td><td>Enable encoder feedback function</td><td>Whether to enable encoder feedback function or not.</td><td>Yes: Enable
No: Disable</td></tr>
<tr><td>N11001</td><td>Encoder type</td><td>It specifies encoder type, which can be divided into incremental type and absolute type. LD5E controller is required to work together with absolute type encoder.</td><td>0: Incremental type encoder;
1: Absolute type encoder.</td></tr>
<tr><td rowspan="2">N11110</td><td>Axial encoder direction</td><td>It specifies the encoder direction of each axis.</td><td>1; -1</td></tr>
<tr><td colspan="3">Decide the encoder direction as formual below:
$$\text{Encoder direction} = \text{Axis direction} \times \text{Logic control direction}$$
Positive logic is adopted by default. If negative logic is used, driver parameter Pn200 should be set to 0005.</td></tr>
<tr><td rowspan="2">N11160</td><td>PG division ratio (X4)</td><td>It specifies pluse number fed back from the encoder via servo division per one motor revolution.</td><td>1~999999</td></tr>
<tr><td colspan="3">When setting in the software, you can figure out value of PG division ratio as formula below:
$$\text{PG division ratio} = \text{Pn212} \times 4$$
Setting of Pn212 is shown as table below:
<table>
<tr><td rowspan="3">Pn212</td><td colspan="3">Encoder divider pulse No. Velocity Position Torque</td><td rowspan="2">Type</td></tr>
<tr><td>Range</td><td>Unit</td><td>Default setting</td><td>Effective time</td></tr>
<tr><td>$16 \sim 2^{30}$</td><td>1 P/Rev</td><td>2048</td><td>After Re-power ON</td><td>Setting</td></tr>
</table></td></tr>
</table>

Parameter		Content	Range
N11200	Motor rotate mode	1: CW as the positive direction; -1: CCW as the positive direction. This parameter needs to be set only when absolute type encoder is enabled.	1; -1
	If driver parameter Pn000 is set to □□□0, the motor takes CCW as positive direction, and software parameter N11200 should be set to -1 accordingly; while if driver parameter Pn000 is set to □□□1, the motor takes CW as positive direction, and software parameter N11200 should be set to 1.		
N11303	Wait time at E-stop	It specifies time used to make the machine stop completely when E-stop is cancelled.	0.5~10s
N74100	Screw pitch	It is used to analyse the distance between coarse positioning switch and fine positioning switch during returning to the reference point.	0~360mm
	In case of directly-connected rail, set screw travel to the value of pitch; in case of gear, set screw travel to the result of "$\frac{\text{Mechanical deceleration ratio}}{\text{Screw pitch}}$".		

3.7 Preheat and Wear

After the machine tool is power on, some functions are not stable enough for the tool machine to machine workpiece. To achieve the best machining state of a machine tool, you must preheat and wear the spindle, trial run some properties, such as the spindle speed and the maximal spindle speed, and adjust the relevant parameters before start machining.

After all axes returning to the machine origin, set the value of parameter "N78000 Warm-up and trial-run switch" as "YES", and the system will start preheating and wearing automatically. The flow chart of the proces is as below.

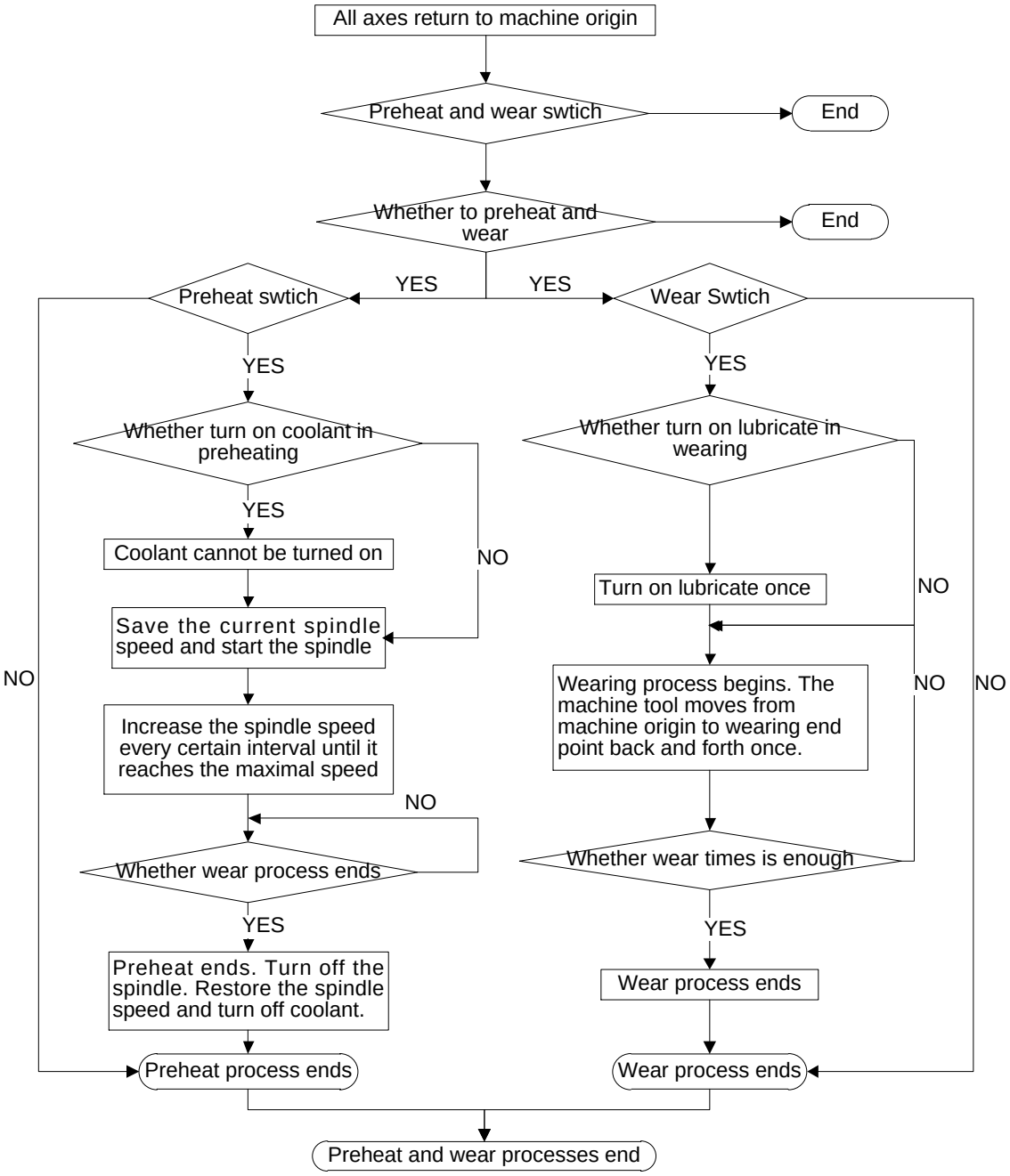


Fig. 3-20 Flow Chart of Preheat and Wear Function

In the whole preheat and wear process, the system is in running state. You can stop the process by pressing key “Stop” or “E-stop” manually. For example, when the spindle is being preheated and the machine tool is running, you can press key “Stop” or “E-stop” to stop the preheating. When preheating process stops, the system will restore the speed of the spindle to that before preheating, close and cool the spindle.

CAUTION

1) If the machine tool stops manually or because of other conditions such as external warning or limits during preheating and wearing, the preheat and weare process will end.
2) If the wear process ends earlier than preheat process does, the preheat process will end although the spindle speed has not reached the maximum.
3) If the preheat process ends earlier than wear process does, the spindle speed will remain the maximum until wear process end.
4) Turn on the output port which controls the switch of lubricate pump. And the lubrication duration is the value of parameter N41002 “Lubricate Duration”.

- **Related Parameters**

Parameter		Details	Setting Range
N78000	Warm-up and trial-run switch	Whether spindle warm-up and crew trial-run operation will be enabled after all axes have returned to the machine origin.	YES: Open NO: Close
N78001	Warm-up switch	The switch to turn on spindle warm-up function.	YES: Open NO: Close
N78002	Trial-run switch	The switch to turn on screw trial-run function.	YES: Open NO: Close
N78100	Coolant On during warming up	During spindle warming up, turn on the coolant.	YES: Open NO: Close
N78110	Warm-up startup speed	The startup spindle speed (RPM) during warming up process.	0~the maximal warm-up startup speed
N78111	Warm-up max. speed	The maximal spindle speed (RPM) during warming up process.	0~24000
N78112	Spindle speed increment	The spindle speed increased every specified time period warming up process.	0~24000
N78113	Spindle speed increase interval.	During warming up, spindle speed will be increased after this interval.	1~60
N78200	Lubrication On during trial run.	Turn on lubrication during screw trial-run process.	YES: Turn on NO: Turn off
N41001	Lubricating Interval	It is the time interval between two start-ups of lubrication pump.	1~1000000
N41002	Lubricating Duration	It is the filling time of lubrication pump each time.	1~100
N78210	Trial-run end	Machine coordinates of end position during screw trial-run process.	-99999~100000

Parameter		Details	Setting Range
N78220	Trial-run times	Repeated times from the machine origin to end position.	1~1000
N78221	Trial-run speed	Moving speed of axis during screw trial-run process.	0~100000

3.8 Spindle Parameter Adjustment

In auto mode, press  to enter the interface as shown in Fig. 3-21, in which spindle speed can be set directly.

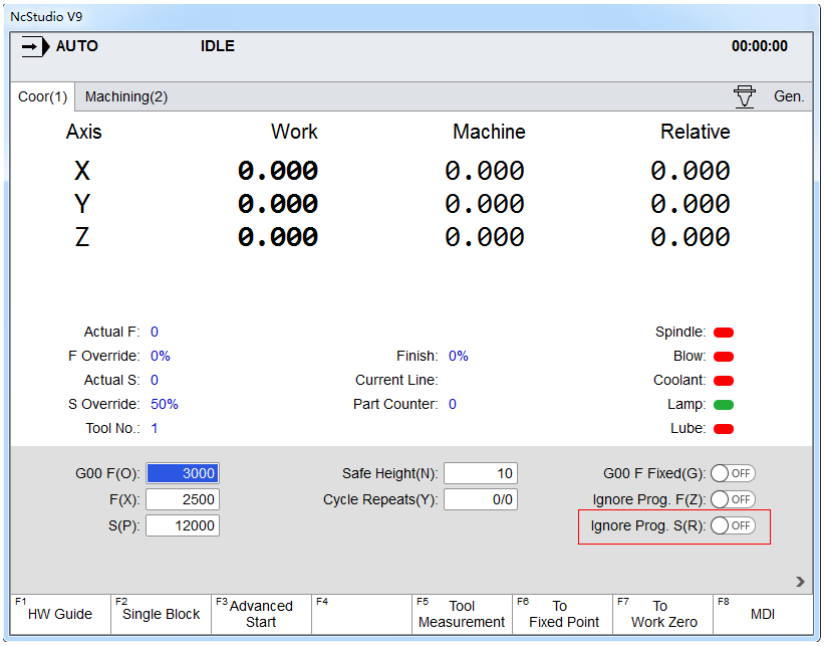


Fig. 3-21 Spindle Speed Setting

Spindle speed can be directly set in the parameter setting area above the manipulation button bar. When parameter N72002 “Ignore Programmed Spindlerate” is set to “YES”, spindle speed in auto machining will adopt the system setting value, i.e. the value of “Prog.S”; when set to “NO”, spindle speed in auto machining will adopt the specified spindle speed in the machining file.

There are two ways for changing parameters under [Coordinate-auto screen]:

1) Press “↑”, “↓”, “→” or “←” to move the cursor onto the desired parameter, and then press “Enter” to eject an input box.

2) Press the corresponding shortcut key to eject an input box. For instance, for “Prog.S(S)”, pressing the letter key “S” on the operation panel will eject an input box for entering a value.

Spindle speed can be controlled by adjusting spindle override. Their relationship is as following:

$$\text{Current spindle speed} = \text{Spindle speed} \times \text{Current spindle override}$$

Spindle override knob is on the operation panel, as shown in Fig. 3-22.

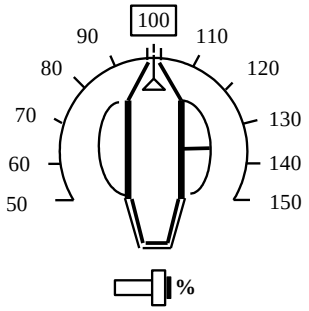


Fig. 3-22 Spindle Override Knob

The least unit of ruler of spindle override is 10% (10% for each scale), with setting range of spindle override "50% ~ 150%".

- **Related Parameters**

Parameter		Details	Setting Range
N20001	Max Spindle Speed	The max. allowable rotation speed of spindle (matched with the inverter setting)	0~999999
N20002	Spindle Speed	The speed of spindle, which should be consistent with the setting of inverter.	0~Max Spindle Speed
N20005	SpindleCool Off Delay Time	Delay time of closing spindle cooling pump after spindle stop	0~600
N20010	Spindle On Delay Time	The delayed time before turning on the spindle so that it can accelerate to the setting speed.	0~60
N20011	Spindle Off Delay time	The delayed time before turning off the spindle so that it can stop completely.	0~60

The value of "Prog.S" under [Coordinate-auto] must be less than that of N20001; the max. setting value of N20001 is corresponding to analog SVC 10V; when the inverter reaches the max. voltage 10V, the corresponding rotary speed of inverter is the max. spindle speed, i.e. the value of N20001.

$$\text{Real-time voltage of analog SVC} = \frac{\text{Current spindle speed}}{\text{N20001}} \times 10\text{V} \times \text{Spindle override}$$

Parameter N20010 and N20011 set the delay time of spindle on/ off, due to a certain time is needed before spindle reaches rated rotary speed since start-up or stops until reaching zero speed; if machining begins before the machine reaching rated rotary speed or other operation is executed before spindle stops completely, it's possible to damage the tool or produce a scrap.

- **Related Parameters**

Parameter		Details	Setting Range
N72004	Spindle Off when	Whether spindle will automatically stop	YES: Stop;

Parameter		Details	Setting Range
	Cycle Stop	when machining stops regularly	NO: Not stop
N72008	Spindle On when Cycle Start	Whether spindle will automatically rotate when machining begins	YES: Stop; NO: Not stop
N73005	Stop Spindle on Pause	Whether spindle will automatically stop when machining pauses	YES: Stop; NO: Not stop
This group of parameters sets the spindle action when commands of machining stop/ start/ pause are executed.			

3.9 Tool Measurement

The process of tool measurement refers to the process of establishing the concrete position of workpiece coordinate system (WCS) in the machine coordinate system (MCS).

When the parameter N11304 “Encoder Feedback” is set to “Yes”, tool measurement with encoder feedback function will be used; while it is set to “No”, tool measurement without encoder feedback (or the traditional one) will be used.

With the help of a tool presetter, tool measurement is realized. As shown in Fig. 3-23, there are ports on the controller corresponding to CUT and COM on the tool presetter. If necessary, such port as “Over-travel Protection” can be added on the controller according to customers’ needs. According to the different installation positions of a tool presetter, tool measurement is divided into mobile calibration and fixed calibration, first calibration and calibration after tool change.

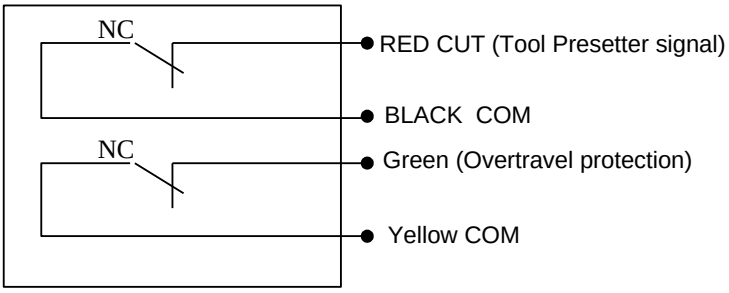


Fig. 3-23 Electrical Wiring Diagram of A WEIHONG Tool Presetter

Fig. 3-24 is the sketch map of tool calibration using of a tool presetter.

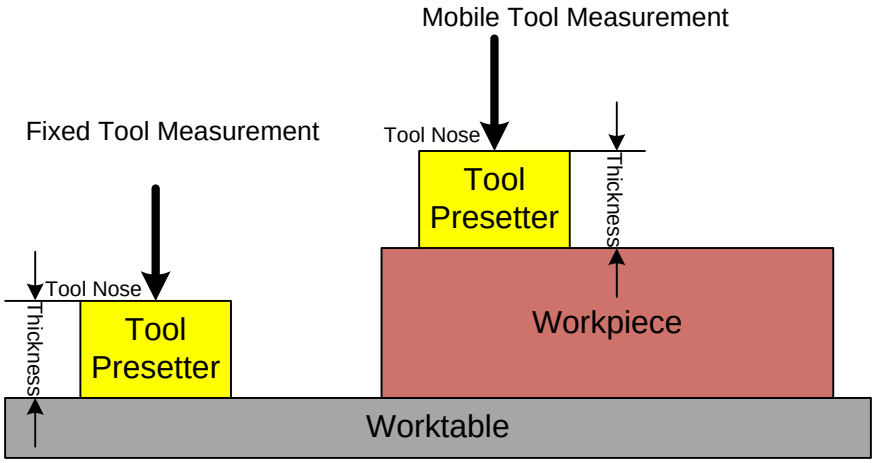


Fig. 3-24 Sketch Map of Using A Tool Presetter

3.9.1 Software Interface

Press key  to access functional area [Machining], and then press F5 to open dialog box “Fixed Calibration (Measure Tool Length)”, as shown in Fig. 3-25. Pressing a shortcut key will select the corresponding measurement type under this interface.

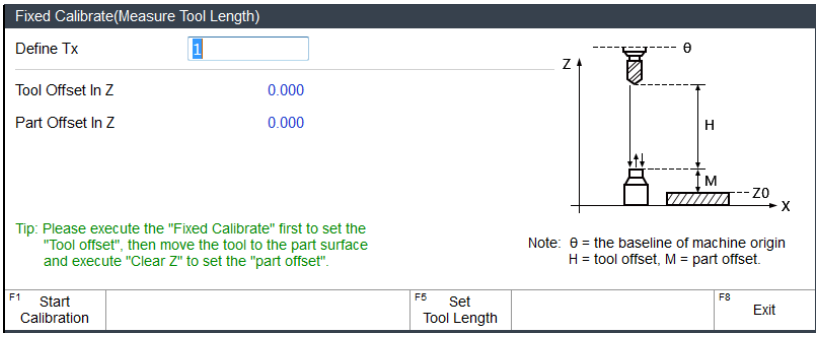


Fig. 3-25 Dialog Box “Fixed Calibration(Measure Tool Length)”

The default for calibration type is fixed calibration. You can modify parameter N80005 to set the calibration type according to your needs.

- **Related Parameters**

Parameter		Details	Setting Range
N80005	Calibration Type	Selection of calibration type.	0; 1; 2
0: Mobile calibration; 1: Fixed calibration; 2: First/Exchanged calibration The default is 1.			

3.9.2 Fixed Calibration

Fixed calibration refers to the measurement operation on a certain fixed position of a machine tool. You can set the fixed presetter position by modifying the value of parameter “N75210”.

Due to tool damage or other causes, the length of a tool and the clamping position may vary during calibration. In this case, you can reconfirm tool offset by fixed calibration. The calibration type is used for multi-tool mode and mainly used in tool machines with tool magazine. See Fig. 3-26 and Fig. 3-27 for the sketch map of fixed calibration.

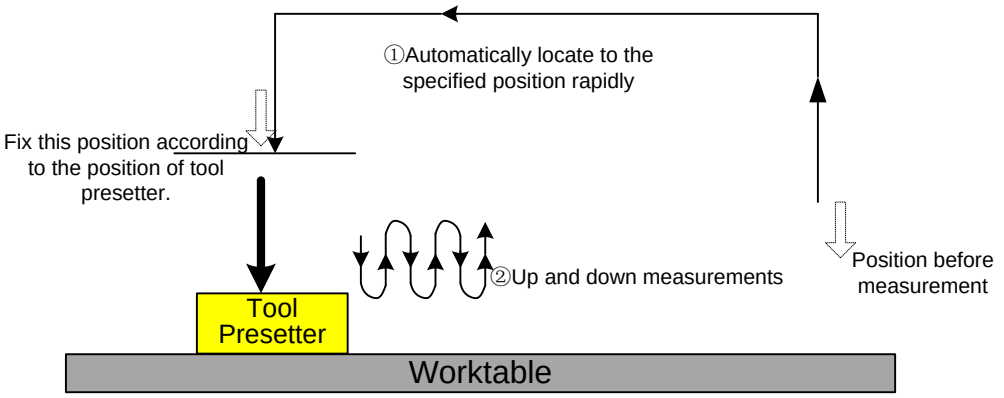


Fig. 3-26 The Process of Fixed Calibration without Encoder Feedback Function

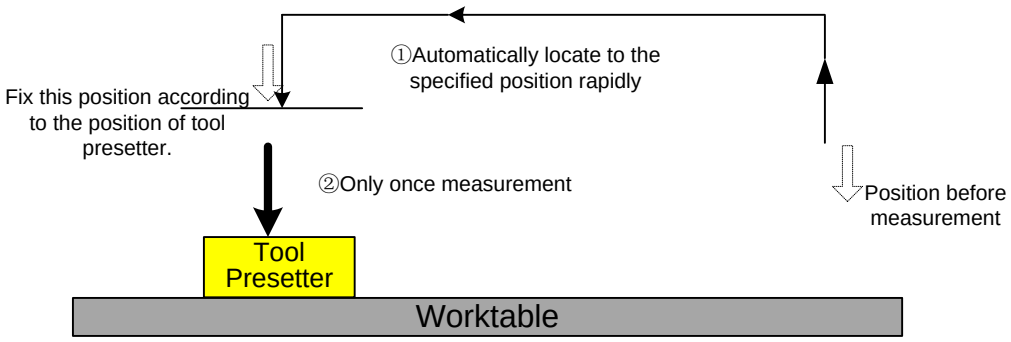


Fig. 3-27 The Process of Fixed Measurement with Encoder Feedback Function

The process of fixed calibration records the machine coordinate when the tool nose touches the surface of the tool presetter. Tool offset is set as the recorded machine coordinate.

Tool offset= Machine coordinate

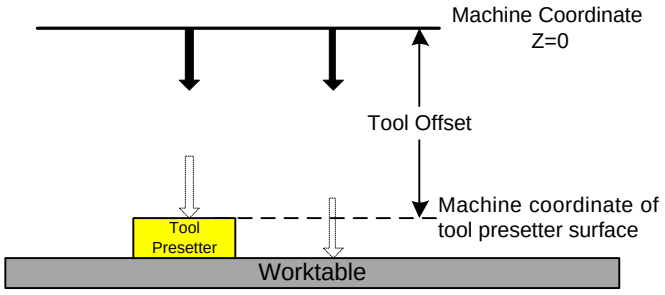


Fig. 3-28 The Sketch Map of Tool Offset

Press key  and F5 to enter dialog box “Fixed Calibration”, as shown in Fig. 3-25.

The steps of fixed calibration are as below:

1) Select a tool according to tool No.;
2) Execute fixed calibration to the selected tool and record the tool offset. When tool presetter is available, press F1 “Start Calibration” to enable auto calibration; while tool presetter is absent in this step, you can press F5 “Set Tool Length” to manually set the tool offset in Z-axis.
3) Record tool offset values.
4) Execute step 1 and 2 to each tool;
5) Select any tool to move to workpiece surface for clearing.

CAUTION

In fixed calibration, tool offset must be set before moving any tool to workpiece surface for clearing.

- **Related Parameters**

Parameter		Details	Setting Range
N75203	F in Fixed Calibration	The speed that the tool moves from the highest point to the calibration-start point in fixed calibration.	-
N75210	Fixed Presetter Position	The machine coordinates of the fixed tool presetter	-99999~99999

For other related parameters about fixed calibration, such as N75001, N75002, N75020, N10050 and N10060, refer to section 3.9.3.

3.9.3 Mobile Calibration

Set the value of parameter N80005 as “0”, and press key  and F5 to enter dialog box “Mobile Calibration”, as shown below.

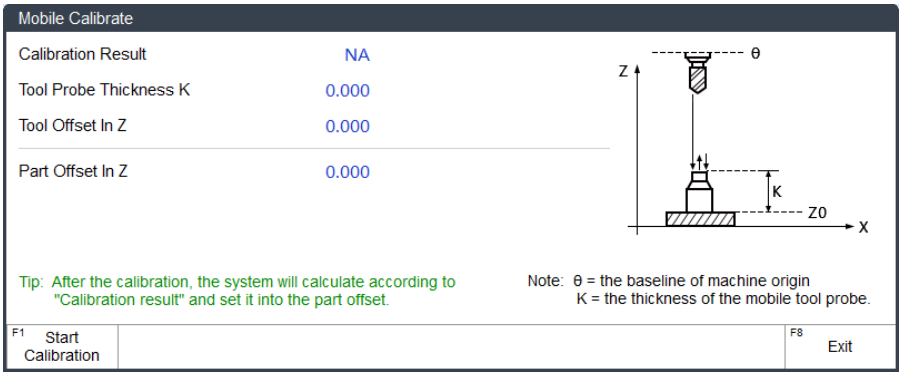


Fig. 3-29 Dialog Box “Mobile Calibration”

Mobile calibration can be used to set workpiece origin of Z-axis by executing calibration at the current

position. After mobile calibration, the system will calculate according to “Calibration result” and set it into the part offset. The thickness of the mobile presetter is decided by parameter N75100.

Workpiece offset = Machine coordinate -Thickness of tool presetter - Public offset - Tool offset

Generally, the default setting values of public offset and tool offset are both “0”.

See Fig. 3-30 and Fig. 3-31 for the sketch map of the process of mobile tool calibration.

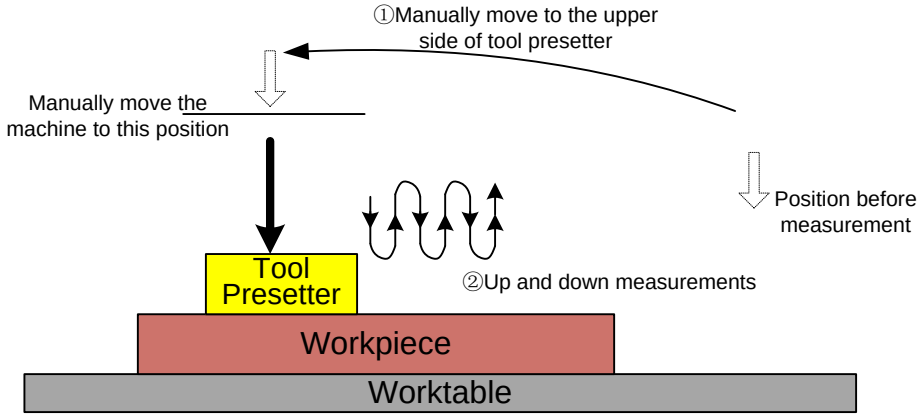


Fig. 3-30 The Process of Mobile Calibration without Encoder Feedback Function

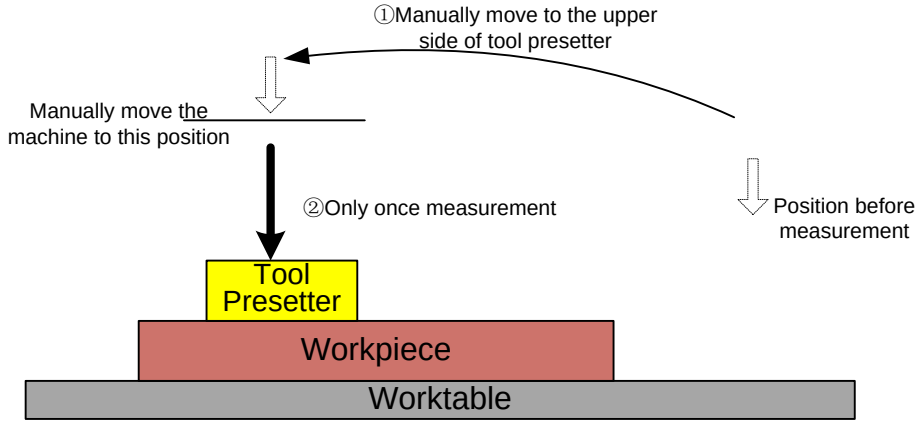


Fig. 3-31 The Process of Mobile Calibration with Encoder Feedback Function

- **Related Parameters**

Parameter		Details	Setting Range
N75100	Mobile Presetter Thickness	The distance from the surface of mobile presetter to WCS Z0.	0~1000

The measurement method of this parameter is as follows:

1) Manually move Z-axis to a certain point over workpiece surface→ shift down the tool nose until reaching workpiece surface→ record the current coordinate of Z-axis (Z1).
2) Uplift Z-axis→ put a tool presetter on workpiece surface→ shift down Z-axis slowly until reaching the presetter and getting the tool presetter signal→ record the current coordinate of Z-axis (Z2).
3) Z2- Z1, and its result equals to the thickness of the tool presetter. Manually enter this result into parameter N75100.

- **Related Parameters**

Parameter		Details	Setting Range
N75001	F in Precise Probing	Tool speed when approaching the presetter surface in tool measurement	-
N75002	Precise Probing Duration	The times of repeated up & down measurements after receiving tool presetter signal when the tool approaches the presetter surface in tool measurement	-
N75020	ToolMea Result Tolerance	The max. allowable error value of tool measurement in multiple tool measurements	0~10
N75024	ToolMea Overtravel Port Addr	The PLC address of input on I/O board, which system gets overtravel signal from the presetter.	-
N75025	ToolMea Overtravel Alarm	Alarm will occur when overtravel in tool calibration.	YES: Enable; NO: Disable
N10050	Positive ToolMeas. Travel Limits	Machine coordinate of upper limit of worktable range in tool measurement	-99999~99999
N10060	Negative ToolMeas. Travel Limits	Machine coordinate of lower limit of worktable range in tool measurement	-99999~99999
Parameter N75020 refers to the max. allowable error of tool measurement set in the system, relative to the average error value of repeated tool measurements in the process of measurement; if average error value is less than N75020, tool measurement succeeds, or measurement fails.			

3.9.4 First-time Calibration/Calibration after Tool Changed

Set the value of parameter N80005 as “2”, and press key  and then F5 to enter dialog box “First Calibration/Calibration after Tool Changed”. The calibration will generate a new part offset value according to the former one and the “Deviation value”.

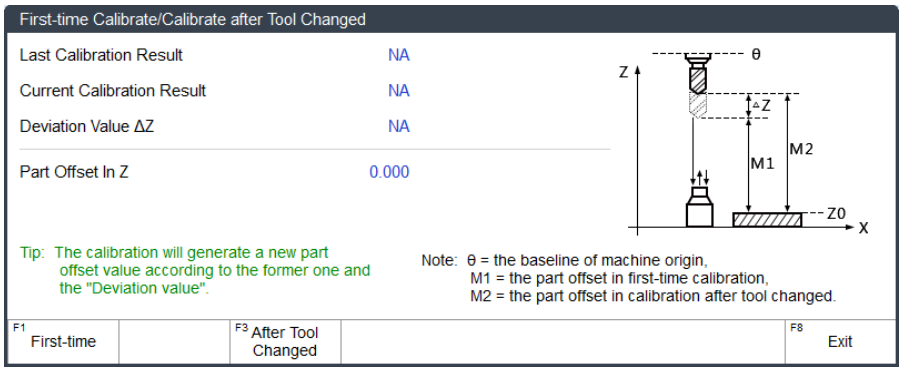


Fig. 3-32 Dialog Box "First Calibration/Calibration after Tool Changed"

The operation steps are as below:

1) Firstly, manually move Z-axis to workpiece surface, and then confirm workpiece origin by mobile calibration or manual clearing. To clear manually, press key 1 to enter functional area [Advance], and press F4 "Clear Z" in interface 「Coor Manager(1)」.

2) Secondly, in interface 「Coor(1)」 under functional area [Machining], press F5 to open dialog box "First Calibration/Calibration after Tool Changed", and then press F1 to start the calibration. The system will automatically record the current machine coordinate of Z-axis. The process is automatically completed by the system, as shown in Fig. 3-33.

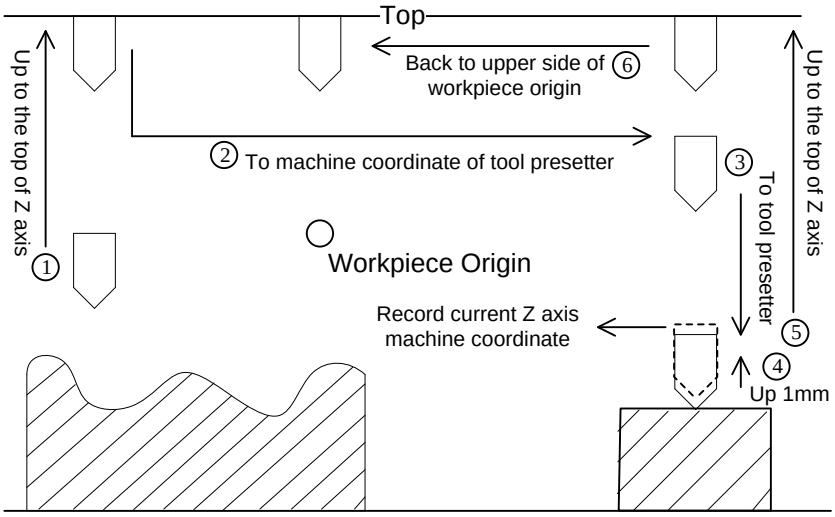


Fig. 3-33 First-time Calibration

3) Start machining after first-time calibration completed.

4) After tool change or tool break, press "F3" to execute calibration after tool changed. The process is automatically completed by the system, shown as Fig. 3-34.

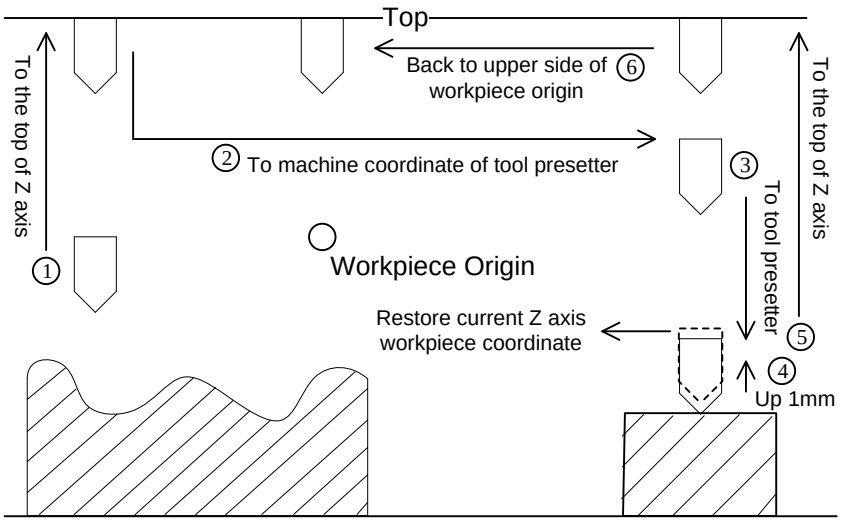


Fig. 3-34 Calibration after Tool Changed

5) Start machining after process “Calibration after Tool changed” is completed.

CAUTION

“Calibration after Tool Changed” must be executed after completion of “First-time Calibration”.

3.10 Offset Setting of Workpiece Coordinate System

3.10.1 Workpiece Coordinate System

In programming, programmers select a certain given point on workpiece as origin (also called program origin) to establish a new coordinate system (i.e. workpiece coordinate system), also a set of right-hand coordinate system. The origin of WCS, i.e. workpiece origin, is fixed relative to a certain point on workpiece and mobile relative to machine origin. The selection of origin of WCS should meet the conditions of simple programming, simple dimensional conversion, and small caused machining error, etc.

The corresponding coordinate systems of work offset are G55, G56, G57, G58, G59 and G54 (the default coordination system). And the relationship of work offset and machine coordinate system is as shown in Fig. 3-35.

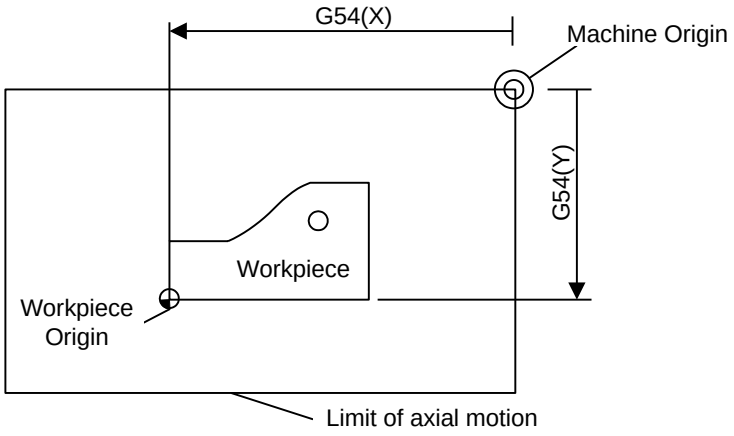


Fig. 3-35 The Relationship of Work Offset and Machine Coordinate System

One, two or multi-work offset can be used in machining program. As shown in Fig. 3-36, if three workpieces are installed on the worktable, then each workpiece holds a workpiece origin relative to G code of WCS. The programming example is as follows: drill one hole on each of the three workpieces, with calculation height Z-0.14.

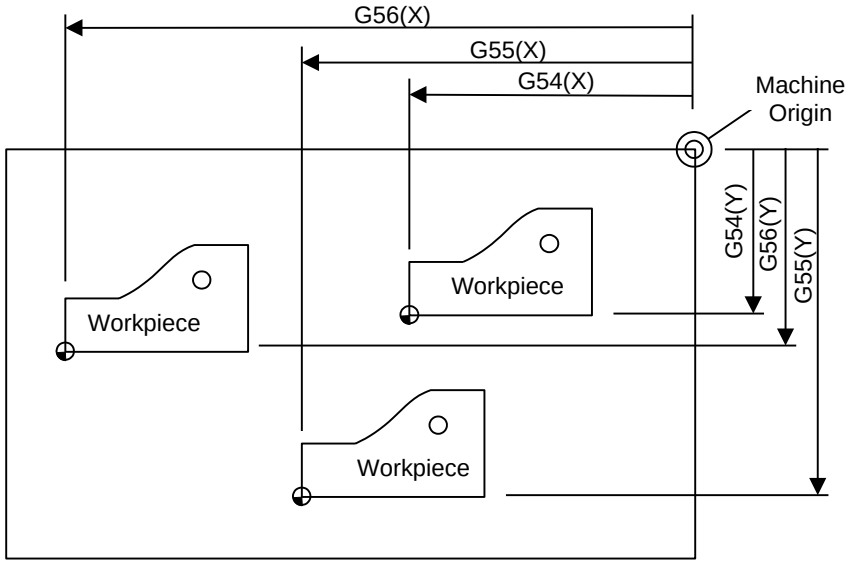


Fig. 3-36 Sketch Map

```
O1801
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X5.5 Y3.1 S1000 M03          （Select G54）
N4 G43 Z0.1 H01 M08
N5 G99 G82 R0.1 Z-0.14 P100 F8.0
N6 G55 X5.5 Y3.1                            (Switch to G55)
N7 G56 X5.5 Y3.1                            (Switch to G56)
N8 G80 Z1.0 M09
N9 G91 G54 G28 Z0 M05                       (Switch to G54)
```

N10 M01

…

Program segments N3 ~ N5, within WCS of G54, are related to the first workpiece; Segment N6 will drill the hole on the second workpiece of the same batch in WCS of G55, while segment N7 will drill the hole on the third workpiece of the same batch in WCS of G56.

Aiming at all WCSs, public offset is used for adjusting workpiece origin of X-, Y-, and Z-axis, but will not change the offset value of “G54 ~G59”.

The related formula of work offset, tool offset and public offset is as below:

Workpiece coordinate= Machine coordinate – Work offset - Tool offset – Public offset

3.10.2 Extended Coordinate System

With up to 120 extended coordinate systems (also known as additional coordinate systems) provided, the total number of WCS is 126 (6+120) in NK300CX system. With the value of parameter N80010 set as “YES’, 126 work offsets can be programmed. The extended coordinate systems are the extension for G54, from G54P0 to G54P119. To view or change the setting of these systems, “PgUp” and “PgDn” are used for page turning while “Home” and “End” for page heading and page footing.

Command G54 Px: Select an extended coordinate system, and “x” here refers to a number within [0, 119].

Example:

G54 P0	Select extended coordinate system 1
G54 P1	Select extended coordinate system 2
G54 P2	Select extended coordinate system 3
G54 Px	Select extended coordinate system (x+1)
G54 P119	Select extended coordinate system 120

- **Related Parameters**

Parameter		Details	Setting Range
N80010	Support Extension Part Offsets	Show coordinate system page which contain 120 groups extra-Part offsets under the advanced function.	YES: Support; NO: Not support
The default parameter setting is “NO”. When above 6 parts are clamped on a worktable, set this parameter to “YES” to support the extended coordinate systems of work offset, so as to save multiple groups of part offsets, which is user-friendly.			

3.10.3 Software Interface

Press key  and then key “1” to access coordinate system management interface, as shown in

Fig. 3-37. This interface displays currently being edited WCS and its corresponding work offset and public offset.

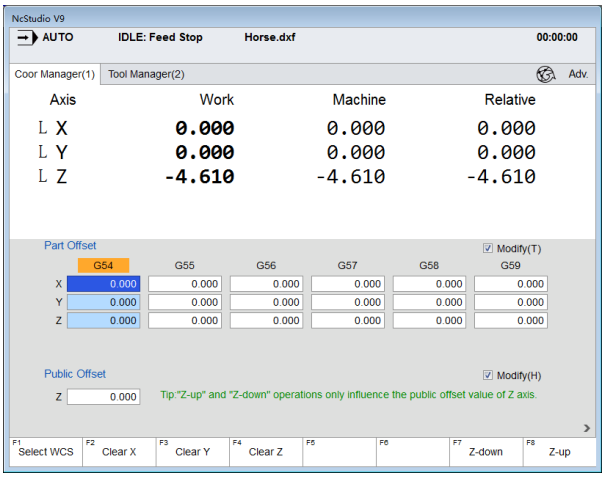


Fig. 3-37 Coordinate System Management Interface

Press the arrow keys to move cursor onto “Part offset”, or to “Public offset”, and then press key “Enter” to modify the part offset or public offset. Or you can press F7 “Down” or F8 “Up” to modify the public offset of Z-axis. “Part Offset” and “Public Offset” can only be modified when the boxes before “Modify (T)” and “Modify (H)” are selected.

CAUTION

1) Switchover Coordinate Systems. In interface 「Coor Manager(1)」, press key “←” or “→” to move the curser to the a coordinate column, whose color will change into light blue. Then press F1 to select the coodinate system, the name of the system such as “G54” will be highlighted. Swtichover is completed.

2) When G54 is set as the current coordinate system, “G54” will not show on the interface 「Coor(1)」 under functional area [Machining] and 「Coor Manager(1)」 under functional area [Advance].

3) When any other coordinate system except G54 is set as the current coordinate system, the name of the system will show on the interface 「Coor(1)」 under functional area [Machining] and 「Coor Manager(1)」 under functional area[Advance]. For example, if G55 is set as the current coordinate system, “G55”will show on the interfaces.

4) When the value of parameter N80010 “Support Extended Part Offsets” is set as “YES”, a tip “’Z-up’ and ‘Z-down’ operations only influence the public offset value of Z-axis” will show beside the public offset in interface 「Coor Manager(1)」.

- **Clear X, Clear Y, Clear Z**

Press F2/F3/F4 to set the value of current machine coordinate of X/Y/Z as the part offset, while the corresponding machine coordinate will not change.

- **Down**

Press F7 and an input box will pop up. You can input the adjustment of Z-axis in the box and press key “Enter”. The workpiece origin on Z-axis will move down a specified distance.

- **Up**

Press F8 and an input box will pop up. You can input the t the adjustment of Z-axis in the box and press key “Enter”. The workpiece origin on Z-axis will move up specified distance.

Both “Up” and “Down” only modify public offset of Z-axis.

- **Coordinate Backup**

Press key  to turn to the next buttons bar. Then press F1 to open coordinate backup sub-screen, as shown in Fig. 3-38. You can save the current part offsets into the system by pressing key “↑” and “↓” to select the directory you will save the part offset and F1 to confirm the saving.

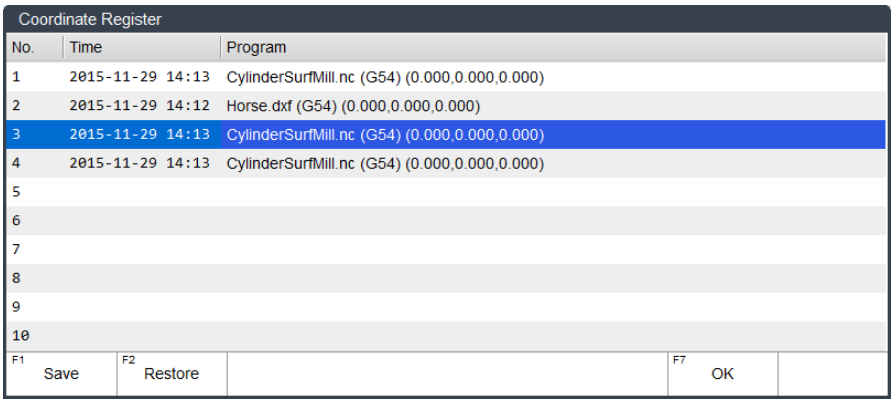


Fig. 3-38 Coordinate Backup

With part offsets saved before, anytime a new machining program file has been loaded, you can press arrow keys “↑” “↓” to select the desired offset and press F2 to restore the offsets into the current WCS, as shown in Fig. 3-39. Press F7 to confirm the loading and F8 to cancel the loading.

Furthermore, after you choose to restore and load the offsets, a new dialog box will show up, asking whether to change Z offset too, as shown in Fig. 3-40. If you select “Yes”, Z-axis offset will be changed. If you select “No”, offsets of axes except Z-axis will restore.

After you select “Yes” of “No”, another dialog box will pop up, asking whether to change public offset, as shown in Fig. 3-41.

Fig. 3-39 Work Offset Restore Prompt

Fig. 3-40 Prompt to Change Z Offset or not

Fig. 3-41 Prompt to Change Public Offset or not

Press F7 to confirm and back to the upper level interface.

CAUTION

Ten sets of part offsets can be backuped in the system. You can save the current part offsets into the system by pressing key " ↑ " and " ↓ " to select the directory you will save the current part offset. The default place is the first line. If there is already a part offset, the newly saved part offset will replace the original one.

3.11 Centering

Centering must be proceeded in handwheel mode.

The system supports manual centering. Manual centering is divided into "Manual Center (two-point centering) and "Circle Center". An edge finder can be used for accurate centering.

When press "F6 ENBL Edge Finder" to make it turn to blue and start spindle, spindle speed decided by the parameter "N20006 Spindle Speed when Centering", whose value is 500 by default and should not be set too large.

When "F7 ENBL Edge Finder" is not available in manual centering, turn on spindle, press "Spindle CW" or "Spindle CCW" at spindle speed set in the software.

- **Related Parameters**

Parameter		Details	Setting Range
N20006	S in Centering	Spindle speed when "Centering", which must be consistent with the setting of spindle transducer.	0~The maximal spindle speed

3.11.1 Line Centering

Line centering refers to the process of locating the midpoint of a line connected by two points, mainly used for locating the center of a blank and set it as the workpiece origin.

In manual mode, press key  to and key "1" to enter interface 「Coor(1)」. Then press F2 to open dialog box "Line Centering", as shown in Fig. 3-42.

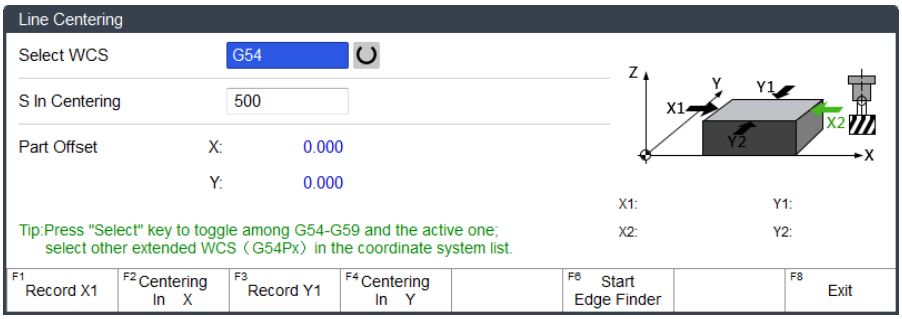


Fig. 3-42 Dialog Box “Line Centering”

In this interface, you can press key “Select” to choose a coordinate system and set the spindle speed in centering.

The operation steps of line centering are as below (An example of X-axis):

1) In HW mode, manually move the cutter to one side of workpiece, and then press F1 [Record X1] to record the machine coordinate of current point.

2) Move the cutter to the other side of workpiece, and then press F2 [Centering in X] to calculate the midpoint coordinate based on the coordinate of current position and last recorded value and set it as workpiece origin.

3.11.2 Circle Centering

Circle centering, means automatic calculation of center point coordinates (generally set as workpiece origin) of a circular blank in terms of the three recorded circle coordinates.

In manual mode, press key  , “1” and then F3 to open the Dialog Box “Circle Centering”, as shown in Fig. 3-43.

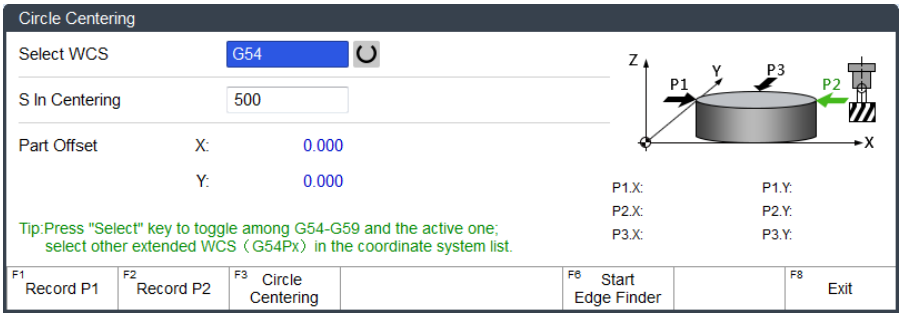


Fig. 3-43 Dialog Box “Circle Centering”

The steps of circle centering are as below:

1) Manually move the cutter to one point on the circumference of a circular blank, and then press F1 [Record P1] to record the machine coordinates of current point as the first group of coordinate;

2) Move the cutter to another point on the circumference, and then press F2 [Record P2] to record the machine coordinates of current point as the second group of coordinate;

3) Move the cutter to the third point on the circumference, and then press F3 [Circle Centering] to calculate the circle center coordinates and set it as workpiece origin based on the current machine

coordinates and the two groups of coordinate recorded previously.

CAUTION

1) When using extended coordinate system, by pressing key “Select”, you can only select coordinate system G54~G50 and the current extended coordinate system. If you need to use other coordinate systems, you can set it in interface 「Coor Manager(1)」.
2) When setting “S in Centering” in a centering dialog box, the modified value will be effective the moment the curser leaves the input box.
3) When executing centering on an axis, the other axes should remain static. For example, the position of Y-axis should remain unchanged when centering is executed on X-axis.

3.12 Adjustment of Velocity & Acceleration

3.12.1 Feedrate Setting

Feedrate (feed speed) can be set directly in the system interface, as shown in Fig. 3-44.

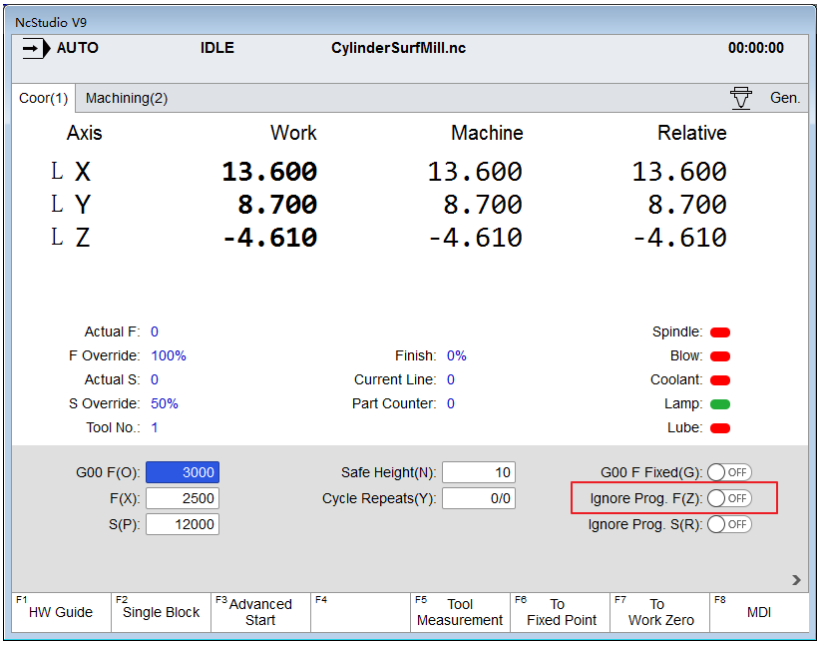


Fig. 3-44 Parameters Setting Zone-feedrate Setting

In auto mode, press key  to access interface 「Coor(1)」. Feedrate can be directly set in the parameter setting zone above the manipulation button bar, as shown in Fig. 3-44. When the parameter N72001 “Ignore Programmed Feedrate” is set to “YES”, the system will adopt feedrate set in the system, i.e. the value of “Prog.F”. When set to “NO”, the system will adopt the feedrate specified in the machining file.

There are two methods to select and set a parameter:

1) Press " ↑ " or " ↓ " to move cursor to the corresponding parameter, and then press Enter to eject an input box.
2) Press the corresponding shortcut key behind the desired parameter to eject an input box. Take "Prog.F(F)" as an example, pressing "F" will eject an input box for entering the desired value.

Feedrate is also related with current feedrate override, so it can be controlled by adjusting the current feedrate override, and the formula is as below:

$$\text{Current Feedrate} = \text{Setting Feedrate} \times \text{Current Feedrate Override}$$

Feedrate override knob is on the operation panel, as shown in Fig. 3-45.

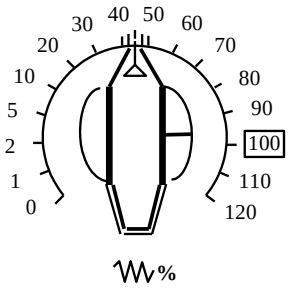


Fig. 3-45 Feedrate Override Knob

The adjusting range of feedrate override is "0% ~ 120%".

3.12.2 G00 Speed Setting

G00 speed refers to the running speed of a machine tool under G00 command.

When the parameter N72003 "Fix Traverse Rate Override" is set to "YES", the running speed of a machine tool under G00 command is fixed, i.e. the value of "G00 F";

When set to "NO", the running speed of a machine tool under G00 command varies with the setting of feedrate override knob.

Similar to feedrate, G00 speed can also be set directly in the system interface.

The concrete setting method is the same as that of feedrate.

3.12.3 Jog Speed/ Rapid Jog Speed

In manual-jog mode, press key  to access the interface 「Coor(1)」. "Manual Low Speed" (jog speed) and "Manual High Speed" (rapid jog speed) can be set directly in the parameter setting zone above the manipulation button bar. See Fig. 3-46.

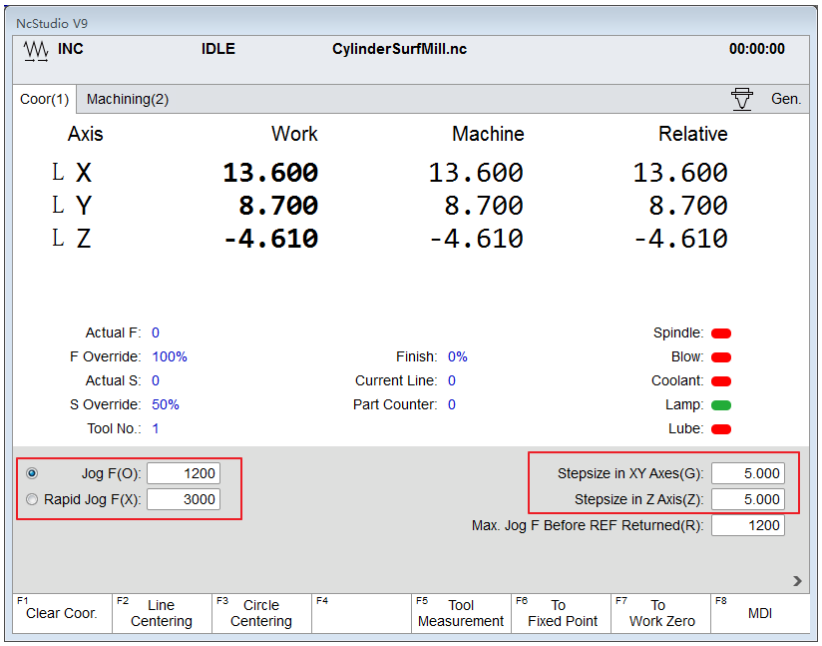


Fig. 3-46 Setting of Jog Speed and Rapid Jog Speed

The concrete setting method is the same as that of feedrate.

3.12.4 Parameter Specification

Except for feedrate and G00 speed, the other involved parameters can be divided into following 5 types: velocity, acceleration, reference circle & circular speed limit, interpolation algorithm, and smooth setting.

1) Speed

- **Related Parameters**

Parameter		Details	Setting Range
N64000	Startup Speed	The max. achievable speed of a stepping motor in startup without acceleration	0~600
N64060	Max Feedrate	The max. speed of a machine tool in machining	0~100000
N71000	Slow Jog F	There are two kinds of speed for option under manual mode: jog speed (Slow Jog Speed) and rapid jog speed, which can be switched by pressing the acceleration key on the operation panel. The system default running speed mode is jog speed.	0 ~ Rapid Jog Speed
N71001	Rapid Jog F		Slow Jog Speed~N13000 Max Feedrate of each axis (Note: The maximal federate supported by hardware is 60000mm/min.)

Parameter		Details	Setting Range
N71002	Jog Max. F before Returning to REF point.	The maximum federate before returning to the REF point in jog mode.	0~3000mm/min
Parameter N64000 “startup speed” applies to the startup frequency of a stepping & a servo driver, zero in default setting of driver. The startup frequency refers to the highest frequency of direct working startup without acceleration of motor. Reasonable setting of this parameter will improve machining efficiency, and avoid low speed segment with bad motion feature of motor. “Startup frequency” is generally included in the ex-factory parameters, but after installation, it will vary, especially in loading motion, thus, it should be set based on the actual measurement of motor power and inertia of a machine tool. Parameter confirmation method: set a lower value at first, and repeatedly make the machine execute typical motion & multi-axis synchronization motion, and then gradually increase this value until fixing the max. startup speed. The actual setting value of this parameter is half of the max. startup speed, with general setting range “300 ~ 400”.			

2) Acceleration

- **Related Parameters**

Parameter		Details	Setting Range
N64101	Rapid Motion Axial Acceleration	The max. acceleration of each feed axis in machine positioning	0.001~100000
N64102	Z-axis Acceleration	The max. acceleration of Z-axis	0.001~100000
N64103	Speed Up Acceleration	Acceleration during speed up	0.001~100000
N64104	Speed Down Deceleration	Deceleration during speed down	0.001~100000
N64120	Acceleration for Corners	The max. acceleration of feed motion on adjacent axes	0.001~100000
N64150	Axial Jerk	The change rate of acceleration of a single axis (acceleration’s acceleration)	0.001~1e+011
N64204	Acc or Dec Time after Interpolation	The larger the value is, the smoother the speed will be. This parameter has no effect on track precision.	0~99999
“Acceleration for Corners” refers to the max. Acceleration of feed motion on adjacent axes, and “1 ~2” times of “Axis Acceleration” is recommended, generally within “1200 ~ 5000”. For higher speed requirement, “2 ~ 4” times of “Axis Acceleration” is recommended. “Axial Jerk” refers to growth rate of acceleration, i.e. the increment of acceleration in unit time, with unit “mm/s3”. It is available for S_type and LEP_type acceleration & deceleration, used to mitigate			

Parameter	Details	Setting Range
the bad effect caused by abrupt acceleration & deceleration of a machine.		

3) Reference Circle and Circular Speed Limit

- **Related Parameters**

Parameter		Details	Setting Range
N64207	Arc Velocity Limit	Only when this parameter is set to "YES" do N64208 and N64209 work.	YES: Valid NO: Invalid
N64208	MAX Velocity of REF Circle	Reference circle is the reference of a machine in processing a circular workpiece. The max. speed of reference circle refers to the max. allowable speed of a machine in processing this circle without strong vibration.	0.001~100000
N64209	MIN velocity of REF Circle	Limit circular motion speed to avoid too low speed	0.001~100000
After installation of a machine completed, you can make the machine process a circle, in which vibration will occur due to centrifugal force. The higher the speed is, the stronger the vibration will be. Gradually increase the feed speed to see the state of vibration of the machine tool until the max. circular speed is achieved, i.e. the max. allowable speed of the machine tool without strong vibration. This circle is regarded as the reference circle, and its max. allowable speed is the max. speed of reference circle. Encountering other circles in machining, the system will calculate their max. centripetal acceleration in terms of the reference circle and its max speed to ensure the centrifugal force is within the debugging value, i.e. the vibration will not be stronger than that during ex-factory debugging. In processing a circle with small radius, even quite low feed speed of the circle will generate very high centripetal acceleration, thus the machining speed will be quite low caused by circle speed limit to limit the centripetal acceleration. To ensure machining efficiency, when the speed calculated by the system is lower than the setting value of N64209, the setting value of N64209 will be adopted in machining.			

4) Interpolation Algorithm

- **Related Parameters**

Parameter		Details	Setting Range
N64203	Path Interpolation Algorithm	Select the most suitable interpolation algorithm to reduce error after debugging.	0: Trapezoid algorithm; 1: S_type algorithm; 2: LEP algorithm;

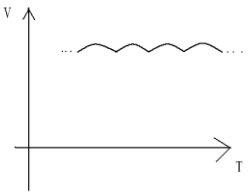
Parameter		Details	Setting Range
			3: Acceleration trapezoid algorithm
N64205	MIN Velocity in LEP	The min. velocity in LEP algorithm interpolation	0~100000
N64203 is used for algorithm selection. The system currently supports trapezoid, S_type, LEP, acceleration trapezoid algorithms. Among them, trapezoid algorithm & S_type algorithm hold the highest efficiency, while LEP algorithm holds the highest machining quality in three-dimensional machining. "Acceleration Trapezoid Algorithm" means acceleration curve is a trapezoid. The relationship of acceleration and time: accelerate to the max. acceleration at "axial jerk", then keep this acceleration constant, and then decelerate to "0" at "axial jerk". Generally, if this algorithm is used, N64150 "axial jerk" can be set within "100000 ~ 200000" (mm/s3). The flexibility of acceleration and deceleration in this algorithm is better.			

5) Smooth Setting

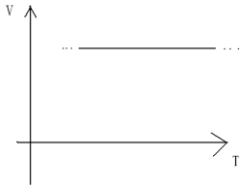
- **Related Parameters**

Parameter		Details	Setting Range
N63002	Delay for Exact Stop	During machining, because of different inertia of each axis, the servo system may meet lag phenomenon at sharp turning corners. This parameter is used to overcome the lag phenomenon produced by the servo system by setting an extra stop time.	0.0~999
N63006	Path Smoothing Time	The larger the value of the parameter is, the smoother the workpiece surface will be. But if the value of the parameter is too large, it will affect the dimension of the workpiece. For a mold machine, it generally should be within 0.01, for a woodworking machine, within 0.03.	0.0~0.064
N64200	Smoothing the Path Velocity	If set to "NO", each motion instruction starts and ends at zero speed. If set to "YES", the system will set a proper start speed and end speed for each motion instruction according to the specific tool path to ensure smoothness of high speed	YES: Valid; NO: Invalid

<table>
<tr><th colspan="2">Parameter</th><th>Details</th><th>Setting Range</th></tr>
<tr><td></td><td></td><td>machining.</td><td></td></tr>
<tr><td>N64201</td><td>MAX Angle Smooth Velocity</td><td>The machine tool will move at the startup speed with the speed instead of “Smoothing the path velocity”, if path corner angle is greater than MAX angle. The system will do “Smoothing the path velocity” if path corner angle is less than MAX angle.</td><td>0~180</td></tr>
<tr><td>N64241</td><td>Decelerate at Max Connect Angle</td><td>Whether to decelerate when the connection angle is approaching its max. value</td><td>YES: Valid;
NO: Invalid</td></tr>
<tr><td>N64245</td><td>Prepared number of path for optimizing performance</td><td>Segments for performance optimization, having no effect on the result of velocity planning.</td><td>1~2000</td></tr>
<tr><td>N64246</td><td>Slide speed for small lines</td><td>Eliminating velocity fluctuation when machining short segments.</td><td>YES: Valid
NO: Invalid</td></tr>
<tr><td rowspan="2">N64247</td><td>Reference length of slide speed for small lines</td><td>Segments shorter than the value of this parameter will be executed speed smoothing.</td><td>0.001~10</td></tr>
<tr><td colspan="3">In machining an arc (or other curves) composed of short segments, velocity fluctuation, like frequent acceleration and deceleration obvious in our S-type algorithm, will occur at places where curvature is relatively large, as follows:
 

Assume that each segments is very short, and the curvature from A to B is large, the actual velocity planning will probably be as follows:
 

The above velocity curve (acceleration→ deceleration→ acceleration…) will lead to oscillation of a machine tool. At this time, the parameter “Slide speed for small lines” should be set to “YES”, and the value of the parameter “Reference length of slide speed for small lines” should be set larger than the length of short segments in the tool path. When the short segments in the tool path are shorter than the reference length, the velocity will be executed smooth treatment. Otherwise, there is no treatment. The velocity after treatment is as follows:</td></tr>
</table>

Parameter		Details	Setting Range
	 In this way, frequent acceleration and deceleration is avoided, oscillation of a machine tool is eliminated, and machining quality is improved.		
N64249	Velocity Smooth for Single Axis	With the function, moving speed of single axis will be restricted in order to get more smooth speed.	YES: Enable; NO: Disable

3.13 Simulation & Track

3.13.1 Simulation

The function of simulation provides a fast but lifelike simulation machining environment.

Running under the simulation mode, the system will not drive a machine tool to do the relative actions but only show the moving track of the cutter at high speed in the track window. By simulation, you can see the moving form of the machine tool in advance, avoiding machine tool damage due to programming mistakes in the machining file. And you can also learn other additional information.

Press key  to access functional area [Program], and load a machining file into the system. Press key  , “2”, and then F1 to start simulation. Operation bar “Simulate” will appear, as shown in Fig. 3-47. You can see the whole machining track in the track window. During normal machining, you can read the completion information of machining file and program information on the right of the track window. Operation bar “Simulate” can help you control the simulation process and simulation track freely.

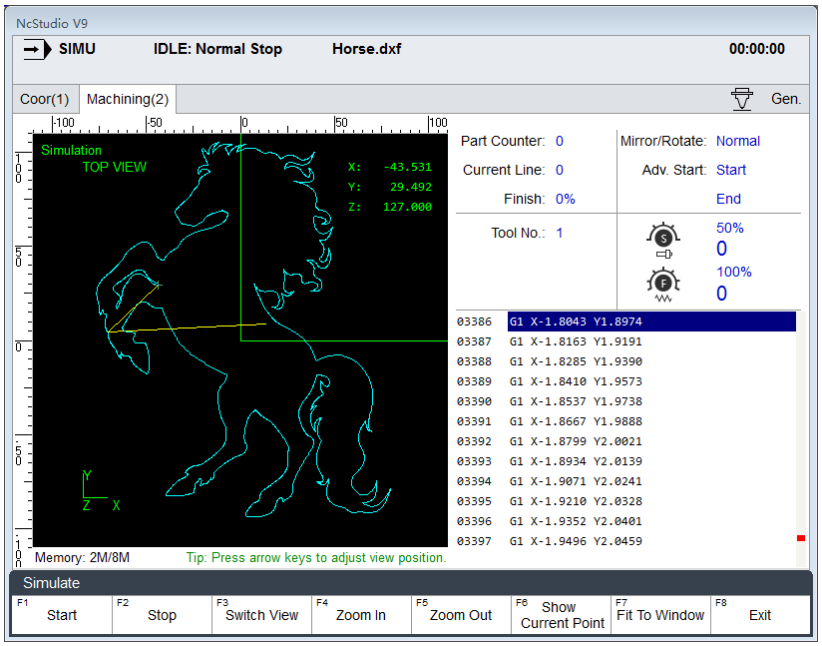


Fig. 3-47 Simulation and Track Window

3.13.2 Motion Track

The track window displays the moving track of the cutter in real time. 3D display enables that you can view the tool path more intuitionally so as to ensure the accuracy of the loaded machining file. In the 3D tracking mode, abundant operation methods are offered by the system for the convenience of viewing the motion track from different perspectives and in an appropriate scaling.

By pressing F2 in the interface 「Machining(2)」, you can see the pop-up “Adjust Graphic” window, as shown in Fig. 3-48. Travel limits of the workbench will show on the window as system default. You can press the displayed shortcut keys to switch view, zoom in and out machining track, and show current point, fit to window, load and clear tracks. You can freely view the machining track and machining state.

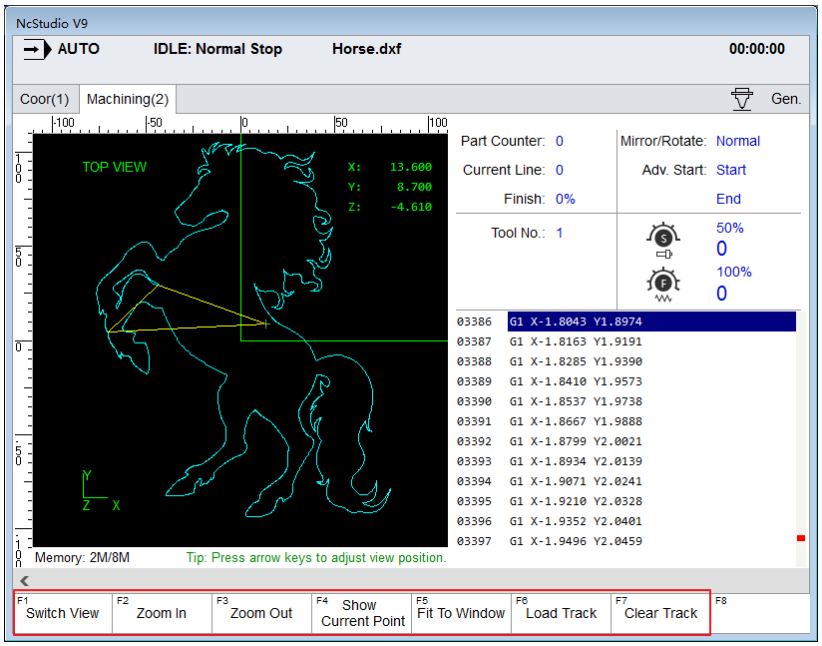


Fig. 3-48 Adjust Graphic

3.13.3 Machining Statistics

As shown in Fig. 3-49, press F3 “Statistics” to open the dialog box. This dialog box mainly displays statistics info of all the current processing and previously processed machining files. See Fig. 3-49. The statistics info shown in the dialog box includes such as number, program name, initial time, run time, total length and part counter. A maximal of 21 machining files can be saved in this dialog box. And the continuous machining records of a same file only record one line of file information.

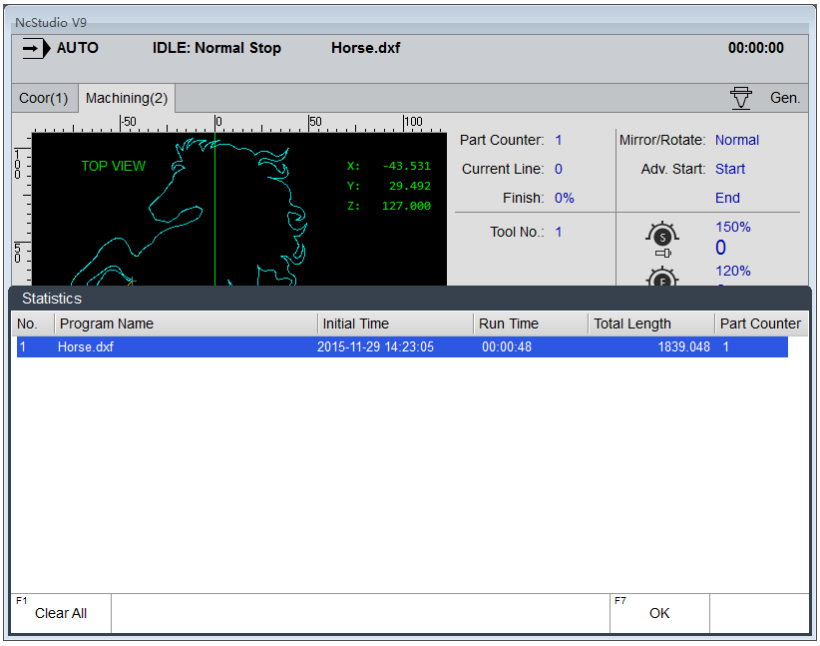


Fig. 3-49 Dialog Box “Statistics”

Press F1 to clear all the history statistics in the list.

Press F7 to exit the dialog box “Statistics”.

3.13.4 Parameter Specification

- **Related Parameters**

Parameter		Details	Setting Range
N81000	Auto Load Graph	It sets whether the system will analyze the machining track automatically after a machining file is loaded.	NO: Not analyze YES: Analyze
N81001	Max File Size	It sets the file size limit in auto track loading. Only when the file size is smaller than or equal to this value can its track be loaded automatically, i.e. “Auto Load Graph” works.	0~100000
N81010	Gradient Fill	Setting whether to use gradient color fill in the track window	NO: Not use YES: Use
N81011	Draw Workbench	Setting whether to draw the boarder of the worktable in the track window	NO: Not draw YES: Draw
N81012	2D Mode	Setting whether to use 2D mode to view the track in the track window	NO: Not use YES: Use
N81015	Clear on Loading	Setting whether to clear the contents of the current view when a new file is loaded	NO: Not clear YES: Clear
N81016	Draw WC Origin	Setting whether to display workpiece origin in the track window	NO: Not display YES: Display
N81017	Draw MC Origin	Setting whether to display machine origin in the track window	NO: Not display YES: Display
N81018	Bkground Color 1	Setting the background color for the track window	Select a color
N81019	Bkground Color 2	Setting the background color for the track window	Select a color
N81020 ~ N81023	G00/G01/G02/G03 Color (running)	Setting the color for motion track commanded by G00/G01/G02/G03 when running	Select a color
N81032 ~ N81035	G00/G01/G02/G03 Color (loading)	Setting the color for motion track commanded by G00/G01/G02/G03 when loading	Select a color
N81045	Grid Color	Setting grid color in the track window	Select a color
N81046	Coordinate Color	Setting coordinate color in the track window	Select a color
N81049	WC Origin Color	Setting a color for workpiece origin in the track window	Select a color
N81050	MC Origin Color	Setting a color for machine origin in the track window	Select a color

3.14 Compensation

There are four types of compensation in the system, i.e. tool compensation, part compensation, screw error compensation and across quadrant compensation. The principles and operation interface of the different types of compensation are introduced in the following content.

3.14.1 Tool Compensation

In CNC machining, the CNC system actually controls the tool center or the related point of the tool rest whose motion track is controlled directly to realize profile processing for the actual parts.

The cutting part actually used is the tool nose or the cutting edge which has dimensional variation with the tool center or the related point of the tool rest, so the CNC system has to compute the corresponding coordinates of the tool center or the related point of the tool rest according to the actual coordinate position of the tool rest or the cutting edge (namely the actual coordinate position of the part profile), which is called tool compensation.

Input the new tool parameter values in the tool compensation interface if the tool nose radius is altered due to tool wear, tool sharpening or tool change, avoiding the trouble to modify the programmed machining file.

Press the advanced function selection key  and key “2” to access interface 「Tool Manager(2)」, as shown in Fig. 3-50. Detailed tool information is shown on the interface, including No., name, diameter, dia. wear, length, len wear. wear, and offsets in different axes. Press F1 to set the tool capacity, i.e. the maximal number of tools. Currently, the system supports a maximum of 255 tools.

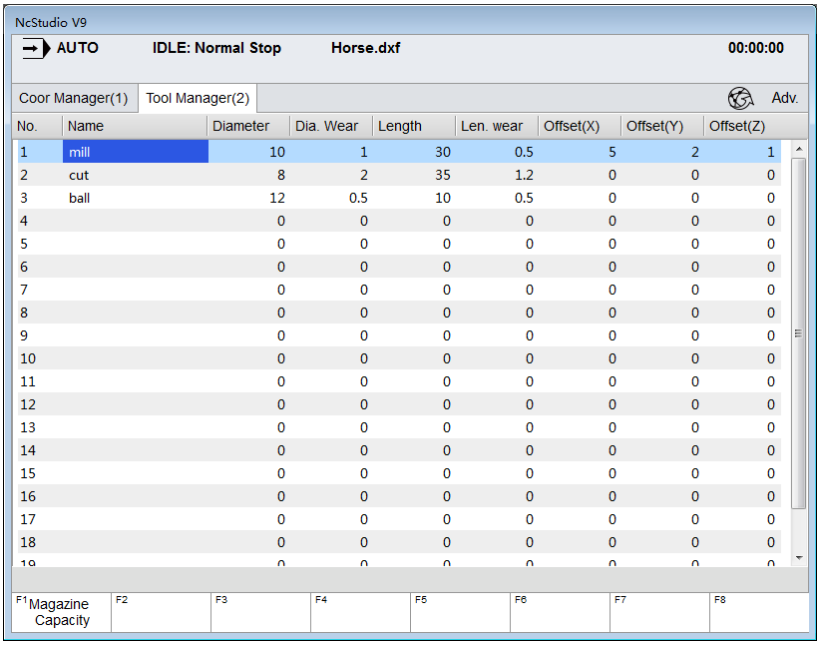


Fig. 3-50 The Interface of Tool Manager

To make tool compensation (including tool radius compensation and tool length compensation) effective, parameter “N62410 Enable Cutter Compensation” should be set to “YES”. G43 (positive offset) and G44

(negative offset) are used for tool diameter compensation while G41 and G42 for tool radius compensation. And G40(cancel tool radius compensation) and G49(cancel tool diameter compensation).

The commands above must be used together with G00/G0a to make tool compensation.

- **Related Parameters**

Parameter		Details	Setting Range
N62410	Enable Cutter Compensation	Setting whether to perform tool compensation	YES: Valid NO: Invalid
N62411	Cutter Compensation Type	The type to establish and cancel cutter compensation	1: Normal type 2: Intersect type 3: Insert type
N62412	Cutter Compensation Direction	Specifying the direction of tool compensation	0: No tool compensation 1: Left compensation 2: Right compensation
N62413	Num of Intervene Detected Graphics	See below for explanation.	1~5
	Interference here refers to over-cut caused by too large tool radius. Parameter N62413 decides interference detection among how many adjacent shapes. When interference phenomena detected, an interference alarm will be given. Generally, setting a smaller tool diameter will relieve the alarm. Note the default value of this parameter is 3. When set to 1, there is no interference detection and alarm.		
N65206	Force to Use Tool Compensation	If it is set to “YES”, when parameter “Enable Cutter Compensation” is set to “YES”, translation of an ENG file calls codes about tool length compensation or tool radius compensation; if it is set to “NO”, even though parameter “Enable Cutter Compensation” is set to “YES”, translation of an ENG file does not call codes about tool length compensation or tool radius compensation.	YES: Force to use NO: Not force to use

3.14.1.1 Tool Radius Compensation (G40~G42)

Tool radius compensation code, namely from G40 to G42, can make the tool moved by the offset value, see in Fig. 3-51.

To make the offset value is the same with the tool radius value, the system will firstly create an offset vector (known as “Starting”), whose length equals to radius of the tool.

Direction of the offset vector is perpendicular to the forward direction of the tool, looking into the tool center from the workpiece. If linear interpolation or circular interpolation is called after “Starting”, the system will contour with the tool moved by the offset, namely, with the tool compensated by radius value.

To end the compensation and make the tool return to the starting point, tool radius compensation code will be canceled and disabled.

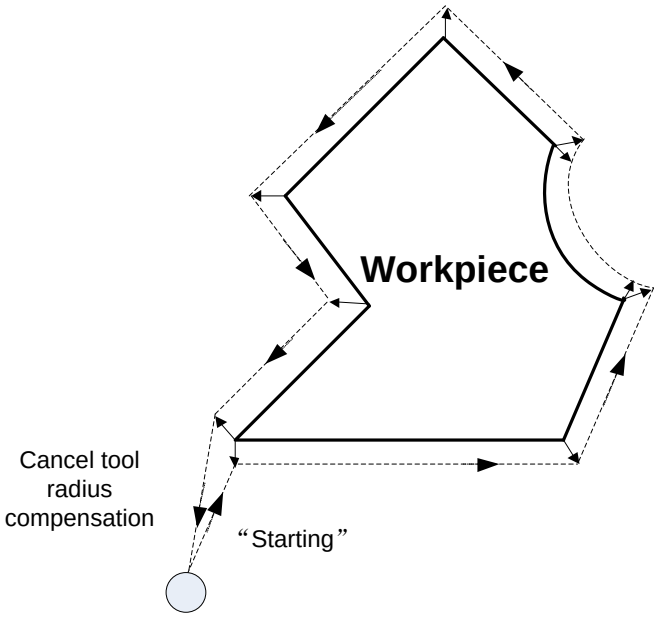


Fig. 3-51 Schematic Diagram for Tool Radius Compensation

3.14.1.2 Tool Compensation Type

Tool compensation should be established before executed, and cancelled after workpiece machining completed. To establish tool compensation is moving the tool to the edge of workpiece in a reasonable way, while to cancel tool compensation is moving the tool to the specified point from the edge of workpiece.

Generally speaking, tool compensation establishment consists of two segments, see segment 1 and segment 2 in Fig. 3-52. The software offers 3 ways to establish and cancel the tool compensation:

1) Normal type: the programming path is translated by 90 degrees to get the segment 2 for establishment, next, make the starting point of segment 2 the end point of the segment 1. Segment 1 and 2 constitute the tool nose path with tool radius compensated. Please note that this type is not available to arc command.
2) Intersect type: the programming path is translated in parallel to get the segment 2 for establishment, next, make the starting point of segment 2 the end point of the segment 1. Segment 1 and 2 constitute the tool nose path with tool radius compensated. Please note that this type is not available to arc command.
3) Insert type: after the programming path is translated, figure out the intersection point of segment 1 and 2. Insert a line from the starting point of segment 1 before translation and the starting point of segment 1 after translation, to get the tool nose path. It is available to arc command as well, but machining efficiency will be affected since an extra segment needs to be completed.

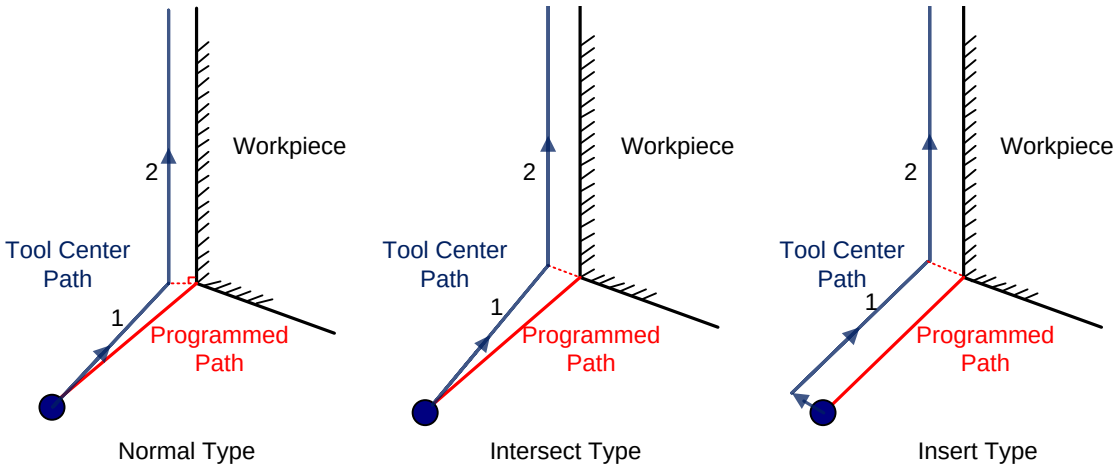


Fig. 3-52 Type to Establish Tool Compensation

3.14.1.3 Tool Compensation Direction

The schematic diagram of tool compensation direction is as shown in Fig. 3-53.

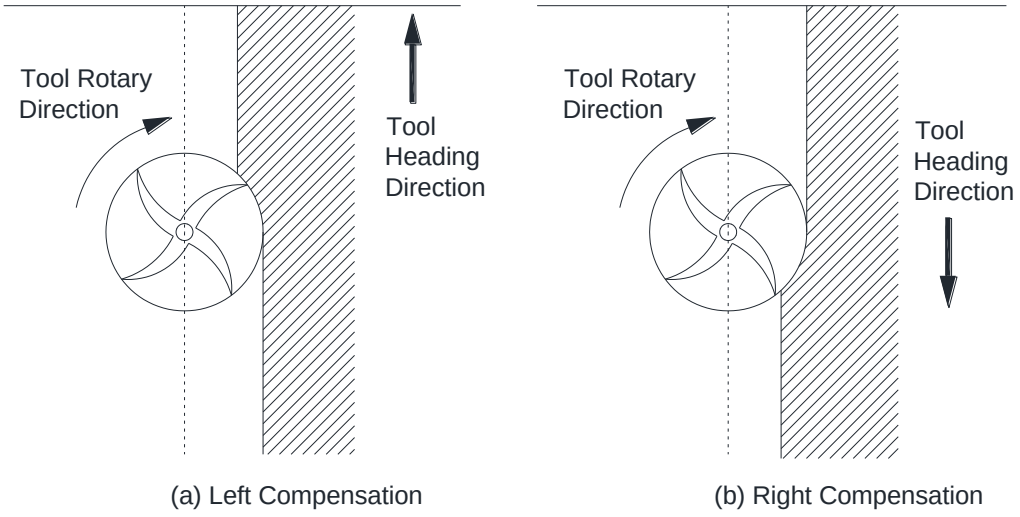


Fig. 3-53 Direction of Tool Compensation (A: Left Compensation; B: Right Compensation)

Programming for tool radius compensation is as shown in Fig. 3-54:

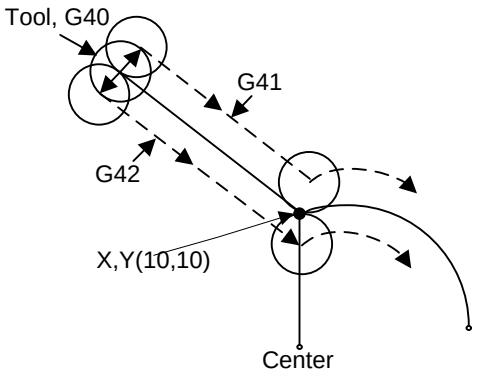


Fig. 3-54 Schematic Diagram of Tool Compensation Machining

```
G17 G01 G41(G42) X10 Y10 F1000 D01    'linear interpolation and tool radius compensation
G02 X_ Y_ I_ J_      'circular interpolation
```

Among the above programming, G41 means left compensation, namely the tool will deviate a distance towards the left side of tool heading direction and this distance is tool radius; G42 means right compensation, namely the tool will deviate a distance towards the right side of tool heading direction and this distance is tool radius. X10Y10 is the endpoint coordinates of linear motion. F1000 represents the tool moves at the speed of 1000. D01 is the parameter of G41/G42, namely the tool compensation number. From D00 to D07, they have their own corresponding radius compensation value in the tool compensation table.

For the details of programming of tool compensation instruction, see *Programming Manual*.

3.14.2 Part Compensation

Press key  to enter functional area [Parameter], and then press key “3” to open interface 「Personalized(3)」. Set the value of parameter N80002 as “YES”. Restart the software, and press key  to enter functional area [Advance] and then press key “3” to open the interface 「Part Comp(3)」.

The system includes single compensation and array compensation, as shown in Fig. 3-55. In single compensation, each workpiece is compensated separately, i.e. the compensation offset of each machining file can be different. In array compensation, the same rows or columns are compensated the same offset. Taking X01Y01 as an example, it compensates the first rows and columns.

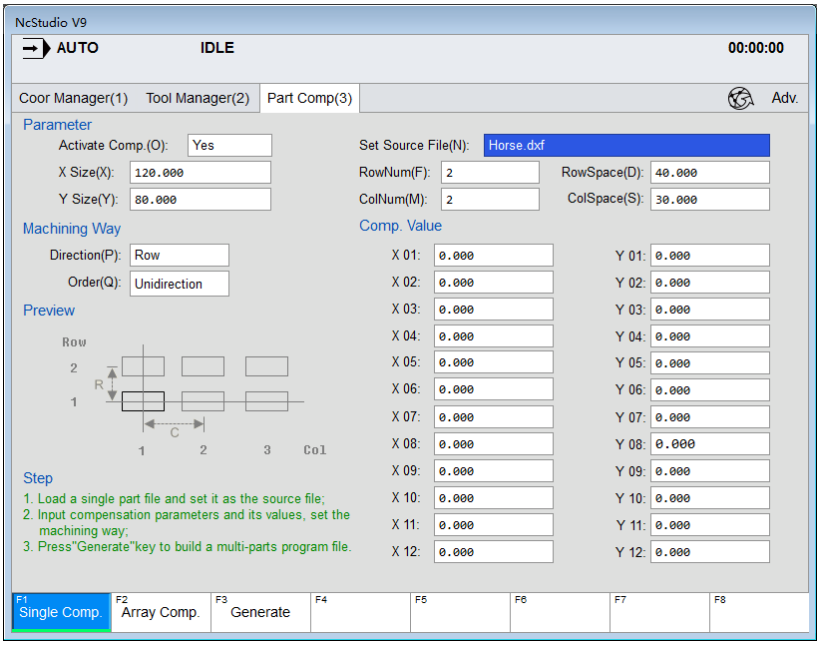


Fig. 3-55 Interface 「Part Comp(3)」

To enable part compensation, set the value of parameter N80002 as “YES”. Otherwise, interface 「Part Comp」 will not show on the functional area [Advance].

Set Source File (N): load the desired single workpiece file into the system firstly, and then turn to this interface and press “Set Source File (N)” to load the file for work compensation. Otherwise, a prompt “No File!” will pop up, as shown in Fig. 3-56.

Fig. 3-56 Source File Error Prompt

After manually entering workpiece size (X size and Y size), rows (CountRow), columns (CountCol), row space (SpaceRow) and column space (SpaceCol), set machining direction and order to determine machining sequence of each workpiece, then enter the compensation offset for each workpiece, and then press F3 “Generate” to generate a multi-workpiece file. Enter the file name, and then press Enter to load the file into the system automatically. The new file is saved to “D:\NcFiles”.

CAUTION

1) After the file is loaded for work compensation, the source file in the system will be deleted automatically, since the final machining file loaded into the system will be the compensation file newly generated.
2) G28, G29, G65, G92, M30 and M2, etc. are not supported in scale and array functions, neither are subprograms in the tool path. If there are codes mentioned before, the system will prompt manual or automatic deletion.

3.14.3 Screw Error Compensation

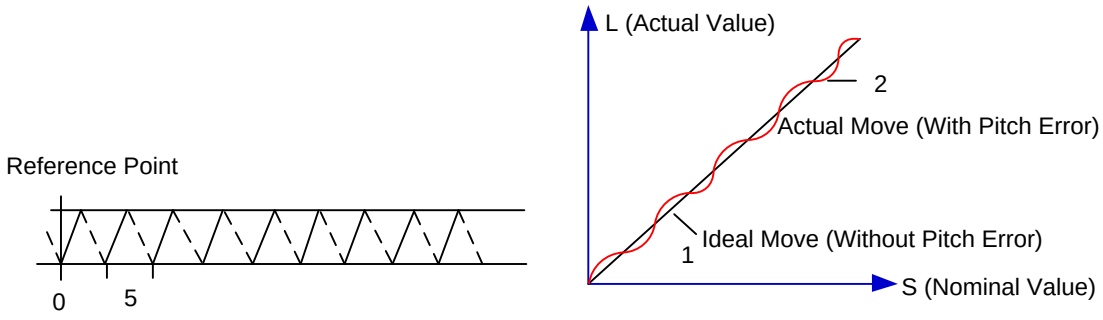
3.14.3.1 Causes of Screw Error and Compensation Method

Screw error consists of screw pitch error and errors caused by backlash. Generally, these two errors don’t need compensation, but backlash compensation is needed in high precision required situation, if higher precision is required, both the two compensations are needed.

- **Pitch Compensation**

Pitch error is caused by screw defect and long-term wear, etc. In order to improve precision, pitch compensation is needed to meet the requirement. The sketch of a screw is shown in Fig. 3-57(A). A coordinate system is established, based on “0” point on the screw as the reference point, nominal value as X-coordinate, and actual value as Y-coordinate. Then the ideal moving curve is as curve “1” in Fig. 3-57(B), however, the actual curve will be curve “2” due to pitch error. That is to say, the Actual value is not the same as its corresponding Nominal value, the actual moving curve deviating from the ideal one, and their difference is called error, i.e.:

Error = Nominal machine coordinate – Actual machine coordinate



(A) Sketch of Screw (B) Picture of Moving Curve

Fig. 3-57 Analysis of Pitch Error

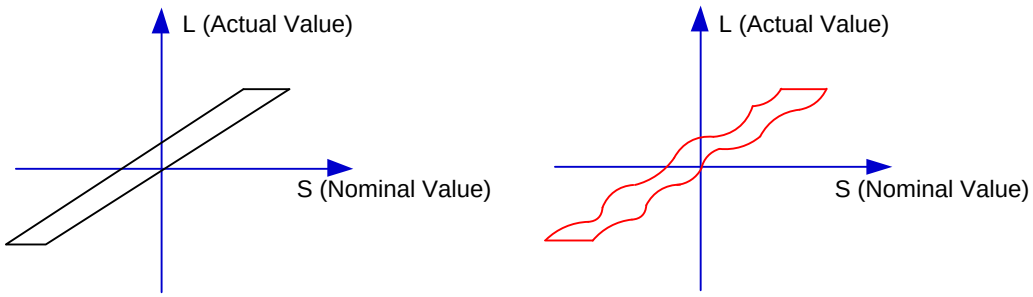
- **Pitch Error Compensation Method**

In pitch compensation, generally pitch error value isn't related to feed direction. That is, when the pitch is too small in positive feed, additional pulse is needed, and thus, when negative feed passes the same position, the same amount of feed pulse should be added. But if the pitch is large, deduction of pulse is needed, and neither is the reducing amount related to feed direction. In software compensation, correction of each point on the error curve should be tabulated and saved to the system memory. Then auto compensation for coordinates of each point is available in running, so as to improve machine precision.

Please see pitch compensation wizard part in section 3.16.1 for detailed compensation method.

- **Backlash Compensation**

Hysteresis feature is caused by forward and reverse clearance. Assume that driving shaft drives driven shaft in negative (CW) rotation, servo motor will be idling without moving worktable because of mechanical driving chain backlash, when the driving shaft suddenly begins CCW rotation (positive motion). After staying at a certain position for some time, the worktable will move backward with the driving shaft; when the direction of the driving shaft changes again, the situation is the same, which is called Hysteresis. If pitch error doesn't exist, under ideal condition, the moving curve of worktable is shown in Fig. 3-58 (A), in which the curve of horizontal section is during the idling of servo motor without worktable movement. The actual moving curve of worktable is shown in Fig. 3-58(B).



(A) Hysteresis Feature (B) Actual Moving Curve

Fig. 3-58 Analysis of Backlash

The popular explanation is: because spindle is generally fixed on the screw whose outer wire and the inner wire on the outer wire cannot be completely matched, backlash compensation compensates the

clearance between the screws of last direction that the spindle needs to finish after reversing its moving direction.

- **Measuring Method and Compensation Method**

Backlash can be measured by a specialized gauge. Firstly, fix the instrument nearby the spindle. Secondly, make the watch hand at the zero point position (machine origin). Thirdly, manually move “a” millimeter, then move back “a” millimeter, and then see the actual moving distance of watch hand “b” millimeter. Therefore, the backlash is measured, namely (a-b) millimeter.

If one axis moves from positive to negative, “+Q” pulse will be output before reversal; conversely, from negative to positive, “-Q” pulse will be output before reversal (Q is backlash, preset by the program).

3.14.3.2 Screw Error Compensation Operation

Actually the system has already combined the above two errors (screw pitch error and backlash) to deal with and will execute error compensation automatically based on the error data in the file after the backward error and forward error of the corresponding nominal coordinate of each coordinate axis are listed into the screw error compensation file.

The detailed operation is: save the measured value of compensation in file “axeserr.dat”, found under the installation directory, i.e. under D:\Naiky\NK-300A\Config\std (varies with system configurations). The system will execute compensation as the file described immediately.

- **Screw Error Compensation File “axeserr.dat”**

The name of the screw error compensation file is “axeserr.dat”, found under the installation directory, i.e. under D:\Naiky\NK-300A\Config\std. Modification to the data in the screw error compensation file will become valid after the software is restarted.

The file format is:

1) Firstly specify length unit, currently the supported length unit is mm and the style of writing is: unit = mm
2) Then specify error sequence of each axis. To work properly, the contents in this sequence must be in the ascending order of nominal machine coordinate value. Refer to Table 3-2 for details.
3) Annotation: it must be in a separate line and started with a semicolon. Its syntax is:

;<Annotation contents>

Note that a semicolon must be the first character of the separate line, that is, no other character should be in front of the semicolon, even blank space.

Table 3-2 Explanation about Axis Error Sequence

Item	Specification
Axis Name	X, Y, Z, (Case-insensitive)
Nominal Machine Coordinate	It is the machine coordinate with a sign with respect to reference point, which is calculated by the given pitch and pulse equivalent (i.e. the length calculated based on the nominal pitch, not on the actual physical one), arranged in ascending order. Nominal machine coordinate must be within the stroke range,

Item	Specification
	or the compensation is invalid.
Backward Error	The error generated by the motion towards decreasing direction of coordinate
Forward Error	The error generated by the motion towards growing direction of coordinate value.
The style of writing of each axis error sequence: [Axis Name] <Nominal Machine Coordinate>, < Forward Error>, < Backward Error> <Nominal Machine Coordinate>, < Forward Error >, < Backward Error > <Nominal Machine Coordinate>, < Forward Error >, < Backward Error > The sign of nominal machine coordinate and actual machine coordinate Pay special attention to the sign of nominal machine coordinate and actual machine coordinate, especially when equipment like laser interferometer is used to measure the length. Calculate after the measured length is converted to the corresponding machine coordinates, or a wrong result may occur.	

Table 3-3 Example of Screw Error Compensation File Format

Condition	Example	Remark
Common cases	;unit=mm 【X】 -570.025, 0.027, 0.083 -450.020, 0.025, 0.077 -330.015, 0.015, 0.068 -210.010, 0.000, 0.057	
A certain axis only needs backlash compensation	;unit=mm 【Y】 0.000, 0.000, 0.030 1000.00, 0.000, 0.030	Only the data of start point and end point of this axis needs writing down. If the backlash compensation on Y-axis is 0.03mm, the setting range is 0 → 1000.

- **Related Parameters**

Parameter		Details	Setting Range
N12000	Screw Error Comp	It sets whether to enable screw error compensation and decides compensation type.	0; 1; 2
N12001	Backlash Compensation Only	It sets whether to enable backlash compensation.	YES: Valid NO: Invalid

1. There are three options for parameter N12000, which are 0 (no compensation), 1 (unidirectional compensation) and 2 (bidirectional compensation).

   Unidirectional compensation

   a) To compensate by reading “Err Pos.” data (unidirectional error data) and backlash value in the screw compensation interface, set N12000 to “1” and N12001 to “YES”.

Parameter	Details	Setting Range

b) To compensate by only reading “Err Pos.” data (unidirectional error data) in the screw compensation interface, set N12000 to “1” and N12001 to “NO”.

Bidirectional compensation

To enable bidirectional compensation, i.e. to compensate by reading “Err Pos.” (forward error) and “Err Neg..” (backward error) data in the screw compensation interface, set N12000 to “2”.

No compensation

To disable compensation, set N12000 to “0”, and N12001 to “NO”.

2. When parameter N12001 is set to “YES”, it means to enable the backlash compensation; when it is set to “NO”, it means that backlash compensation will not be enabled and comprehensive compensation will be made by reading backlash value and pitch error data from the error file.

Enable Backlash Compensation:

To only enable backlash compensation, set N12000 to “0” and N12001 to “Yes”.

3.14.3.3 Software Interface and Operation

Press the advanced function selection key  to access interface 「Parameter」, open manufacturer access, and you can find parameter N12000. Set the value of the parameter as you need no compensation, single compensation or double compensation. And then press key “4” to access interface 「Screw Error Comp (4)」. Unidirectional and bidirectional compensation interfaces are shown in Fig. 3-59 and Fig. 3-60.

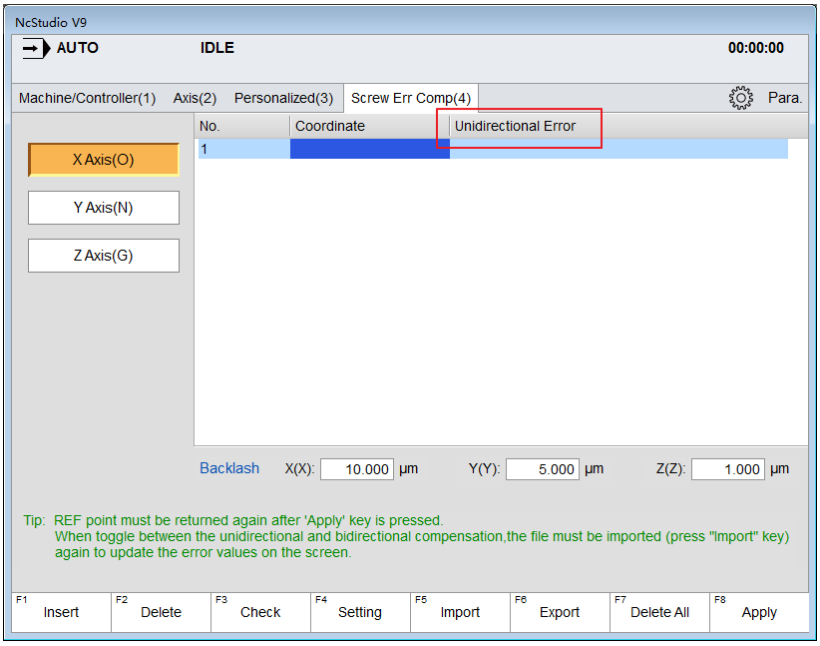


Fig. 3-59 Interface 「Unidirectional Compensation」

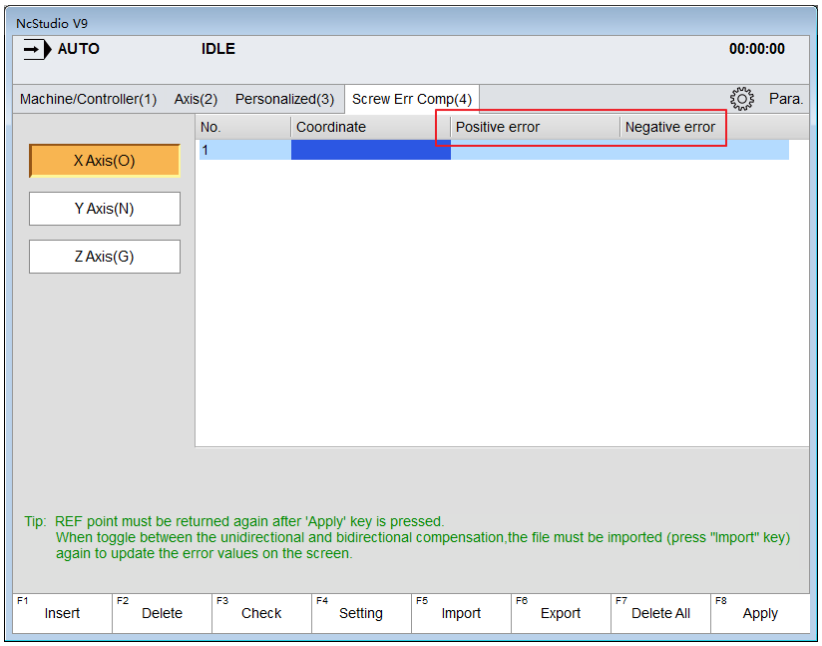


Fig. 3-60 Interface 「Bidirectional Compensation」

Unidirectional compensation is fit for the situation that the forward error and backward error are relatively stable. If forward direction is the first in measure, enter the forward error (default) into “Err Pos.”; if backward direction the first in measure, enter the backward error into “Err Pos.” Backlash of each axis can be used together.

Bidirectional compensation reads forward error and backward error to execute comprehensive compensation, fit for the situation that forward error and backward error are not stable. “Backlash” is not shown on the double compensation interface.

- **Unit**

Coord. (Position coordinate): mm

Err Pos. (unidirectional error), Err Pos. (forward error), Err Neg. (backward error), backlash: μm

- **Insert and Delete**

Press F1 to insert an input line, enter the coordinate and error value; press F2 to delete the compensation data in the line where the cursor stays.

- **Check**

Check function is used to check whether the compensation inputted is valid. If the data is valid, the system will prompt “The data is legal and can be applied or exported”; if the data is invalid, the system will show the reason why it is invalid; if there is no compensation data and you press F3, the system will prompt “Valid data is not detected”.

You can enter and check the compensation data line by line, and you can also check the validity of all data you have entered in all lines.

- **Setting**

Press F4 to access the dialog box “Setting”. In this dialog box, you can generate a set of compensation position in the compensation list by entering input box “Starting Position”, “Pointing Spacing” and “Measuring Points”. You can enter the compensation data line by line. See Fig. 3-61.

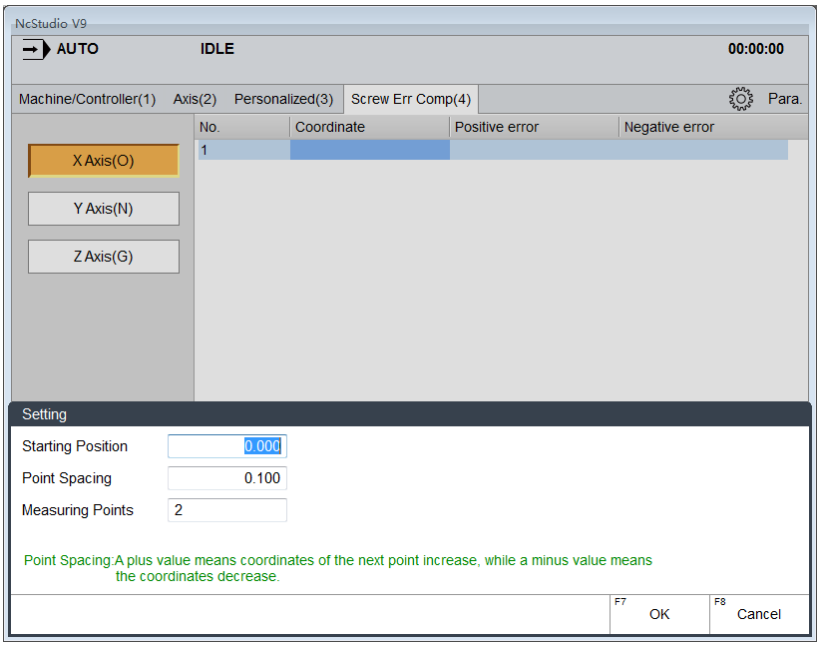


Fig. 3-61 Setting-[Screw Err Comp(4)]

- **Import and Export**

By pressing F5, you can import three types of files, ".lin", ".rtl" and "axeserr.dat". The system supports importing files from an USB disk.

Press F6, and the system will prompt "<export> will cover original file, are you sure?"; press "Yes", and the system will export the above compensation data into file "axeserr.dat". And the system will automatically detect whether any flash disk exist. If flash disks are detected, the data will be exported to a flash disk; if flash disks are not detected, the data will be exported to directory "D:\Naiky\NK300A\Config".

CAUTION

1) Check whether the .lin or .rtl file to be imported is generated correctly.
2) After manually modifying the axeserr.dat file, check whether the data of each axis is arranged in ascending order or descending order, and whether the data, including nominal coordinate, forward error, and backward error, are correct compared to the backlash of each axis.
3) After switching from unidirectional compensation to bidirectional compensation and restarting the software, load the file again to refresh the forward error and backward error of bidirectional compensation.
4) To improve the precision of the imported file, the data should be the average of multi-measurement (at least two times) on the same coordinate position.

   Compensation error data= Measured error data- Error data of machine origin

- **Delete All**

This key is used for deleting all data.

- **Apply**

After this key is pressed, the compensation data will be written into the drive, and the axeserr.dat file will

be saved to the D disk.

CAUTION

1) After modification to Coord. (position coordinate), Err Pos. (unidirectional error), Err Pos. (forward error), Err nEG. (backward error), backlash, parameters N12000 and N12001 and execution of "Apply", there is a must to return to machine origin first to ensure compensation accuracy.
2) Error value= Actual machine coordinate- Nominal machine coordinate
3) Ascending sequence and descending sequence can be set.
4) Check whether there is any invalid data in the axeserr.dat file after opening the software and importing the file.

3.14.4 Across Quadrant Error Compensation

Across quadrant error compensation, also called friction compensation, refers to the distortion, the most commonly seen is a spike, at the conversion part of two adjacent quadrants in circle machining of a machine tool. To eliminate this kind of distortion, error compensation is necessary.

Across quadrant compensation parameters are used for spike compensation when machining arc passes across quadrants. The setting method along positive and negative directions of X/Y/Z is similar.

- **Related Parameters**

For there are 12 groups of parameters "time", "distance", "delay" and "intensity", only one of them is listed in the following table.

Parameter		Details	Setting Range
N12020	Turn On AQE Compensation	Turn On AQE Compensation	Setting whether to enable across quadrant compensation
N12100	Time (Group 0)	The bigger the value is, the larger the area will be influenced by the compensation. The recommended value is about 0.02 s.	0~10
N12101	Distance (Group 0)		0~10
N12102	Delay (Group 0)		0~10
N12103	Intensity (Group 0)		0~1
To make across quadrant compensation effective, parameter N12020 should be set to "YES". The larger the value of "Distance" is, the more obvious the compensation result will be. But note that too large value will make the arc concave, and too small value cannot decrease the arc height effectively. The recommended setting value is 0.3~3 times of the actual height of the spike measured by a measuring device like a laser interferometer (compensation result and compensation time are related to compensation intensity).			

Parameter	Details	Setting Range
Delay: the spike may not exactly appear at the conversion part on some machine tools due to the difference of mechanical property of each machine tool, but a distance away from the quadrant point. Estimate the time for finishing this distance and set it as the value of the “Delay”. Intensity has an influence on the compensation result: the bigger the value is, the more obvious the result will be.		

3.15 Log and Diagnosis

3.15.1 Alarm

Press  to access functional area [Diagnosis] and open interface 「Alarm (1)」, as shown in Fig. 3-62. What shown in the interface is alarm information, including “alarm description”, “alarm appeared” and “alarm disappeared”. You can press F1and F2 in the operational button bar to choose to show the existing alarms or history of alarms.

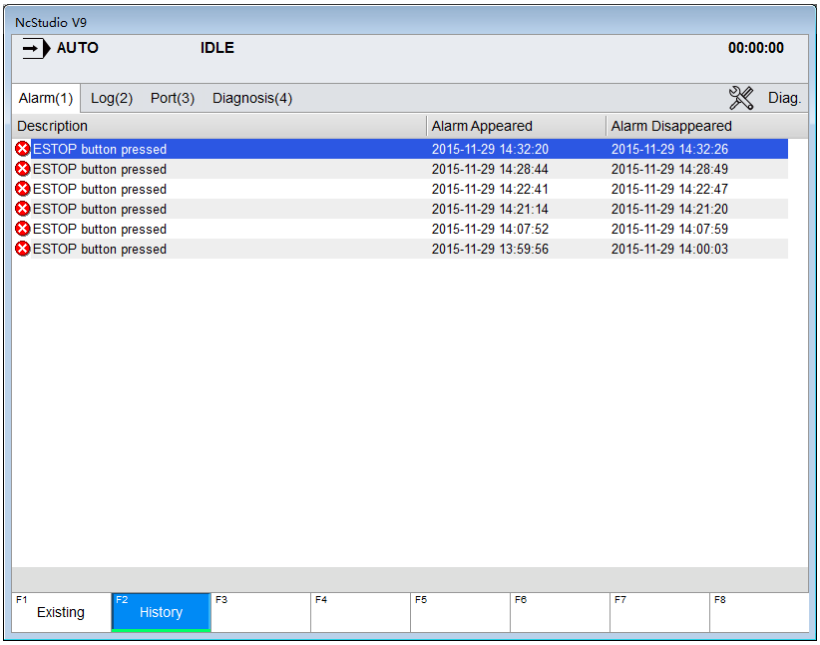


Fig. 3-62 Interface 「Alarm (1)」

3.15.2 Log

Press key  to access functional area [Diagnosis], and then press key “2 “ to open interface 「Log(2)」.

The interface 「Log(2)」 shows important operations and system events. Not only can the log info since

this time start-up be browsed, but also history records can be viewed. See Fig. 3-63.

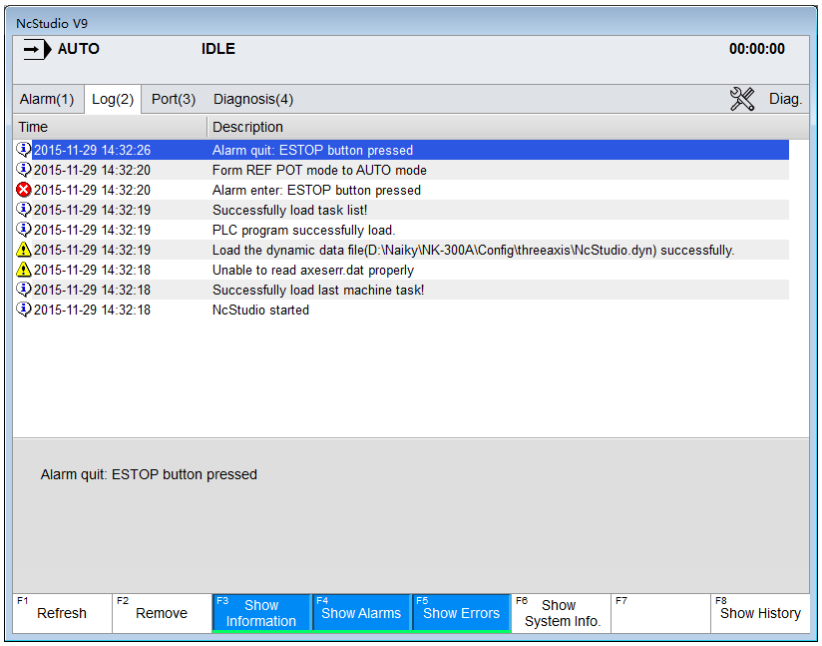


Fig. 3-63 Interface 「Log(2)」

- **Refresh**

Press F1 to refresh the log list in order to make it synchronize with the system.

- **Remove**

Press F2 to remove all the current log information.

- **Show Information, Show Alarms and Show Errors**

The shortcut keys are “F3”, “F4” and “F5” respectively for “show information”, “show alarms” and “show errors”.

The default state is checked and highlighted in blue, namely the system displays normal information, alarms and error info by default. If you don’t need certain info displayed, you can press the corresponding shortcut key to eliminate the blue highlight. For example, you can press F5 (shortcut key of [Show Errors]) to make the button bounced and the system will hide the error info.

- **System**

Press F6 to view the system info.

- **History**

Press F8 to display all the logs since recording.

3.15.3 Port

Press  to access functional area [Diagnosis], and then press key “3 “ to open interface 「Port(3)」.

Please see section 3.2 for operation about interface 「Port (3)」.

3.15.4 Diagnosis

Press  to access functional area [Diagnosis], and then press key “4“ to open interface 「Diagnosis(4)」.

The interface 「Diagnosis(4)」 displays current feedback machine coordinates of each axis. After inputting a valid sampling port into the channel and setting sampling interval, press F1 to diagnose the corresponding port. See Fig. 3-64.

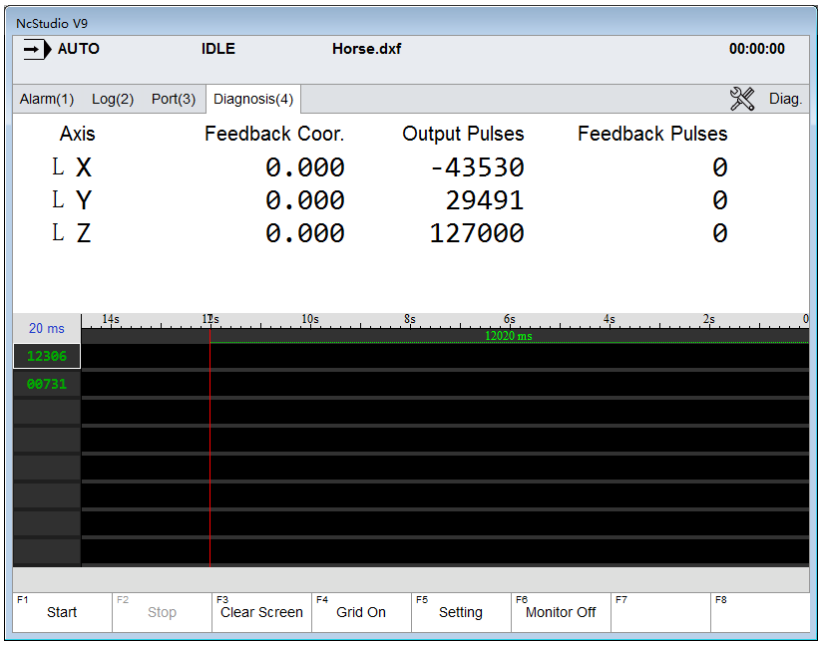


Fig. 3-64 Interface 「Diagnosis(4)」

- **Start**

Press F1 to start diagnosing the corresponding port.

- **Stop**

Press F2 to stop diagnosing the corresponding port.

- **Clear Screen**

Press F3 to clear the diagnosis result of the corresponding port.

- **Grid On**

Press F4 to bring grid lines into the sampling window.

- **Setting**

Press F5 to set the sampling interval, as shown in Fig. 3-65.

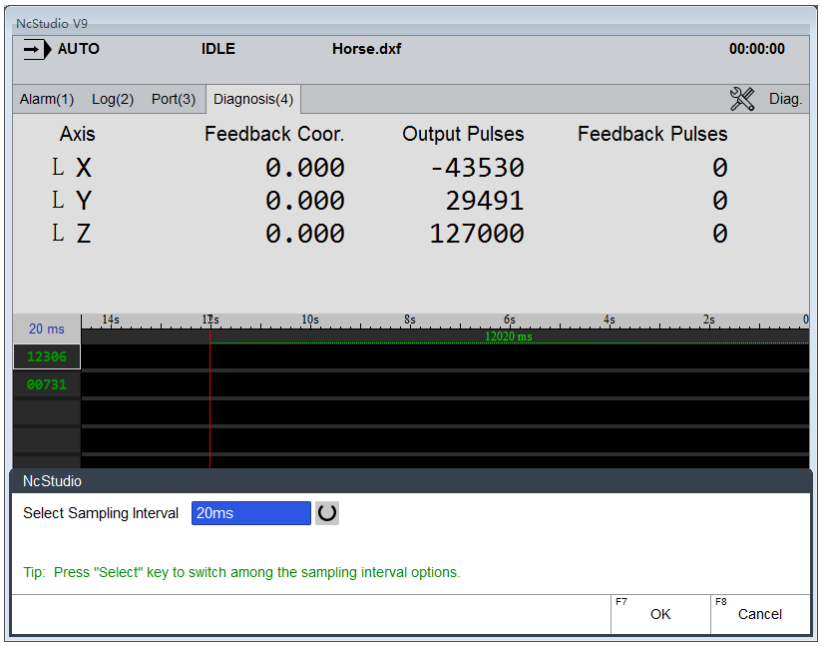


Fig. 3-65 Dialog Box of Setting Sampling Interval

Select a sampling interval in range (20ms, 100ms, 1s and 20s), and press F7 to confirm. And then back to interface 「Diagnosis(4)」, and press F1 to start sampling periodically the corresponding port or PLC address, realizing tracking detection of the port.

- **Monitor Off**

Press F6 to cancel the monitoring of the corresponding port.

3.16 Program File Management

Program file management manages the machining files in the system, related to operation of machining program.

3.16.1 Machining Wizard

NK300CX offers 5 basic machining program wizards: circular contour, circular pocket, rectangular contour, rectangular pocket and screw measure. You just need to input some simple parameters to complete the operation of circular contour and rectangular contour, etc. Take circular contour milling as an example in the following:

Press key  to access functional area [Program], and press key “4” to open interface 「Wizard(4)」. Then press key “O” to enter the circular contour wizard screen, as shown in Fig. 3-66. To achieve the desired results, you can set parameters for the selected machining shape, such as milling inner contour or outer contour (milling inner contour mills the region inside, and milling outer contour mills along the contour), part diameter, start point X/Y, single infeed, total depth (of several accumulated cutting) and tool diameter. After parameters are set, it is suggested to save them before loading the wizard into the system.

Besides, export of the settings is supported in the system. You can export the complete program file of the current parameter setting to the local directory. Press F8, and you can view the exported file in interface 「Local (1)」 under the same functional area. In interface 「Local (1)」, you can execute operations including “Load”, “Edit”, “Delete”, “Array”, “Unload”, “New”, “Rename” and “Copy to USB” to the selected file. The default file names of the exported file in the five basic wizards are “CirContour.nc”, “CirPocket.nc”, “RectContour.nc”, “RectPocket.nc”, and “ScrewErr Measure_X.nc”. Please note that only one program file of a wizard exported can be copied to local directory. In other words, for the same wizard, the latest program file exported to local directory will cover the file exported last time.

The operation method and parameter setting principle of circular pocket, rectangular contour and rectangular pocket are the same as those of circular contour, except the setting of some parameters.

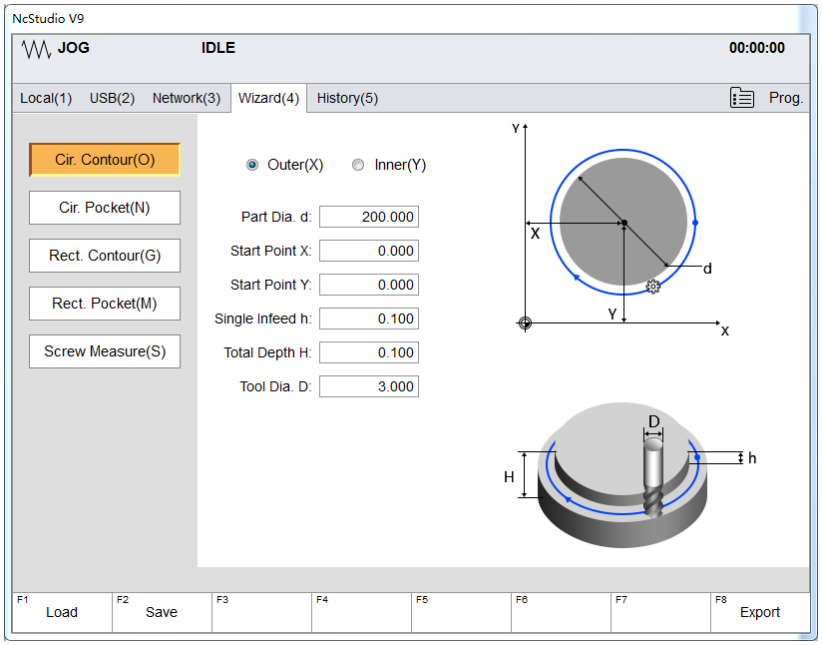


Fig. 3-66 Interface 「Circular Contour(O)」

In interface「Wizard(4)」, press key “S” to access interface「Screw Measure(S)」, as shown in Fig. 3-67. You can set relevant parameters for desired effect of machining drawing.

Wizard “Screw Measure” is used to measure screw error via laser interferometer.

Enter the values for start and end points, measuring points, repeats and dwell time, and then press F2 to save the setting. The system will generate a program file automatically to the directory D:\NcFiles\Wizards. Press F1 to load the file into the system.

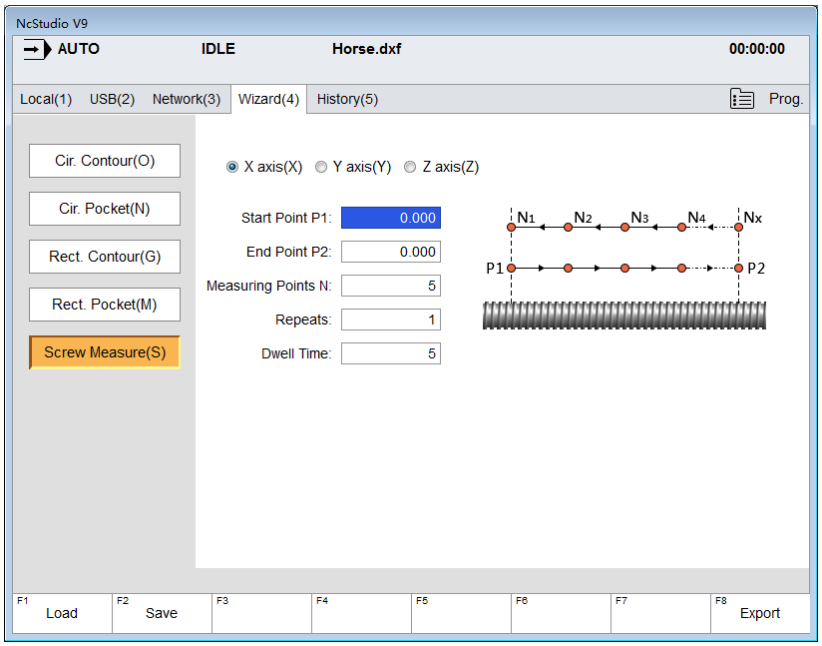


Fig. 3-67 Interface 「Screw Measure(S)」

Or you can directly press F1 after setting parameters to save and load files. After the first time setting, if you modify the parameter values and press F1, a prompt will pop up, as shown in Fig. 3-68. Select “Yes” to save and load the newly generated file.

Fig. 3-68 Prompt for New Parameter File

CAUTION

1) To begin with the operation, an axis should be selected, besides, X/Y/Z-axis can only be selected alone at one time.
2) The startpoint and end point should be located within the travel range and the latter must be larger than the former in absolute value.
3) One cycle refers to the process from the starting position to end position, during which, interferometer will record a group of data. However, a mean value will be used when written into the screw error file.
4) Measuring interval = (End position-Start position) / (Num of measuring points-1). To get an accurate measuring result, the starting position and end position should be calculated precisely and the number of measuring points should be an integer.

3.16.2 Program File

Press the program function selection key  to enter the machining file screen, and then press key “1”, “2”,”3” to switch between interface 「Local(1)」, 「USB(2)」 and 「Network(3)」, as shown in Fig. 3-69.

- **Local**

A list of local program files under the root directory D:\NcFiles are displayed in interface 「Local(1)」. The upper part of this interface is a file list box, while the lower part prompts the path of the currently selected file and available space of the driver. Press “↑” or “↓” to move the cursor onto a program file, and then press key "F1~F8" to execute corresponding operations to the file. Press F1 to load a file, and loading progress displayed on the information bar. At the same time, the system will automatically check the file being loaded. If an error is found in the file, a specific prompt about the error will be displayed on the information bar. After successful loading, other operations can be executed.

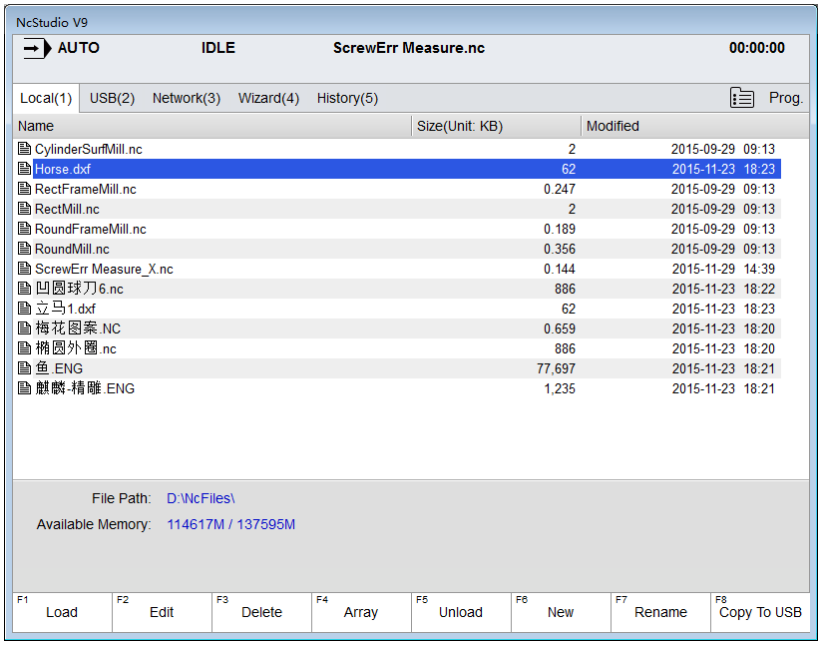


Fig. 3-69 Interface 「Local(1)」

You can find the machining files under the default path of the hard disk (D:\NcFiles) and execute such operations as load, edit, delete and rename, etc. on them. In addition, you can create a new machining file under the default path and edit it.

- **USB**

Press key “2” to access interface 「USB(2)」 where the program files under the root directory and subdirectory folders of USB disks, as shown in Fig. 3-70. The operation in the interface is similar to that in interface 「Local(1)」.

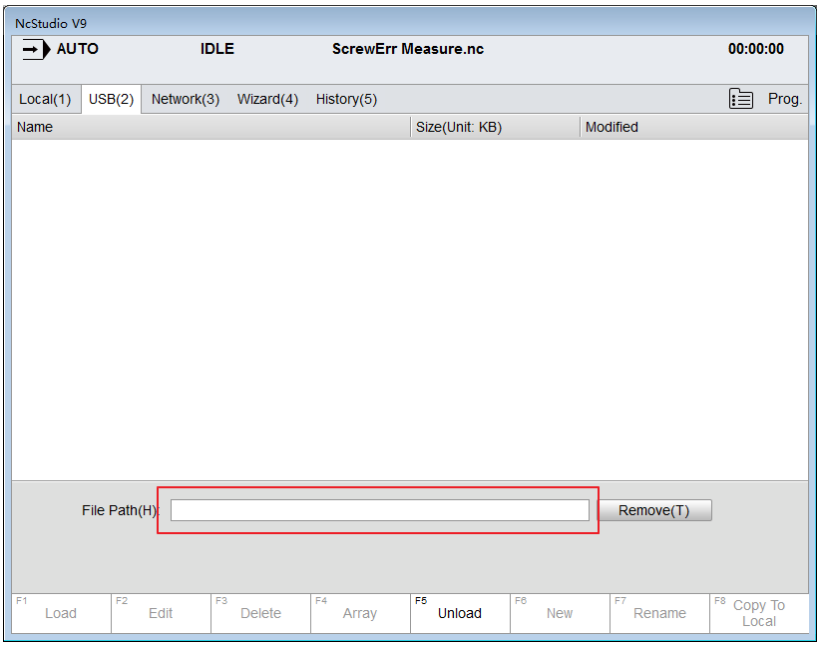


Fig. 3-70 Interface 「USB(2)」

- **Network**

Press key “3” to access interface 「Network(3)」. In this interface, visiting among several machines sharing the same LAN are supported. Tool path files on the local directory or shared within the LAN are displayed in this interface, click the blank box behind “Specify File Path(H)” to select files on the local directory or shared within the LAN, as shown in Fig. 3-71. The operation in the interface is similar to that in interface 「Local(1)」.

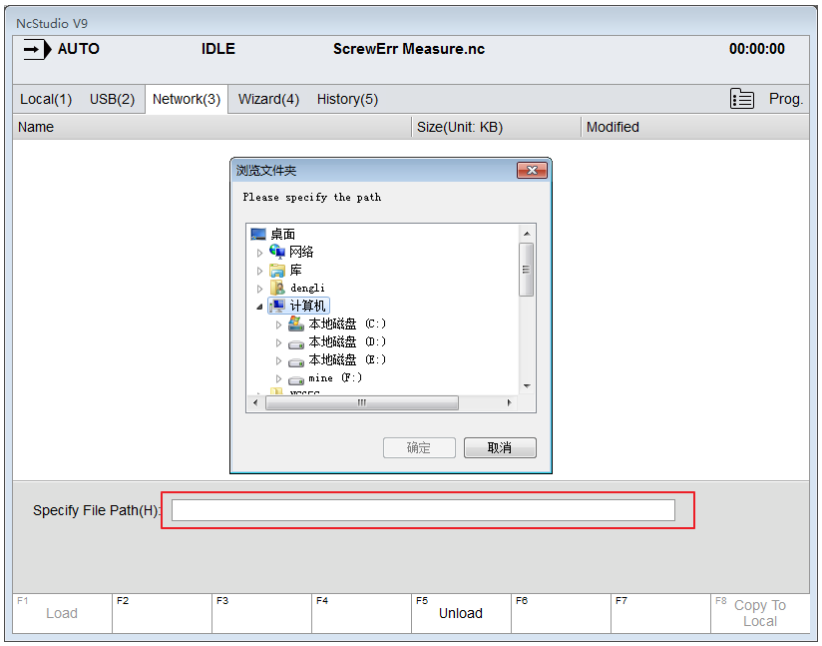


Fig. 3-71 Interface 「Network(3)」

CAUTION

1) Folder NcFiles is the default folder for sharing files or open files, for convenience of easy access to several machines.

2) To avoid mal-operations, delete function is beyond available for network programs. If you need to make any modification to files on the network, you can copy it to the local first before any further operations.

3) When the network is disconnected, program files loaded from the LAN network to the local (not copy) will be un-readable after power off or restart of the system or the software.

- **Edit**

After a machining file is selected, press F2 to make the system eject its embedded program editor automatically, in which you can do the following operations to the file, like “Insert line”, “Delete line”, “Copy line”, “Goto line”, “Find”, “Replace” and “Save”.

After selecting a file, press F3, a prompt box asking whether to delete the file will pop up.

CAUTION

1) Currently loaded file cannot be edited. Unload it before editing if necessary.

2) If the selected file is under the state of being loaded, edited or processed, deleting it is prohibited.

- **Array**

This function executes array machining for a machining file. Press F4 to access the sub interface, as shown in Fig. 3-72.

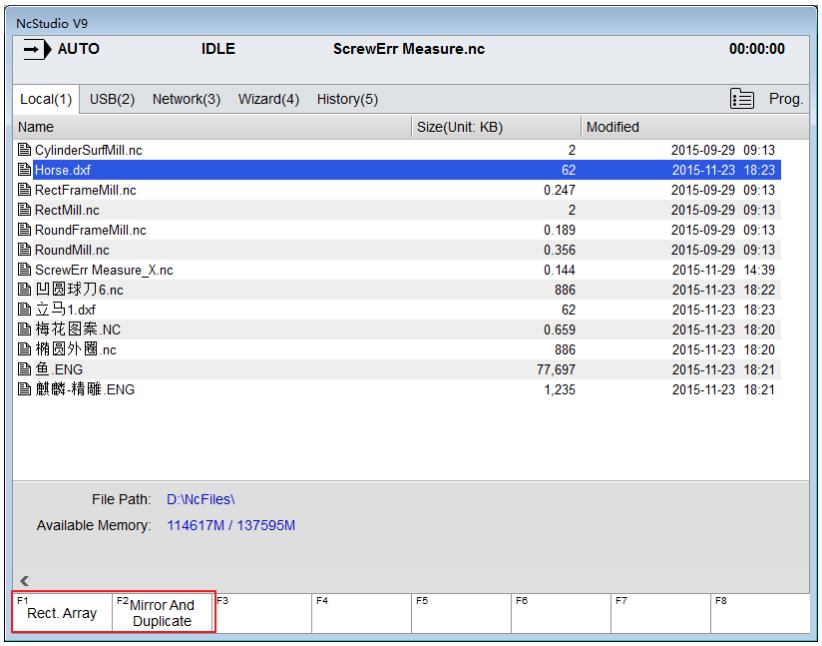


Fig. 3-72 Array Machining

Press F1 “Rect. Array” to open a lower hanging dialog box, where you can set the row number, column

number, row spacing and column spacing, etc. After setting the items, press F7 to generate a file, whose name can be user defined. After confirmation, the new generated file will appear in the program list.

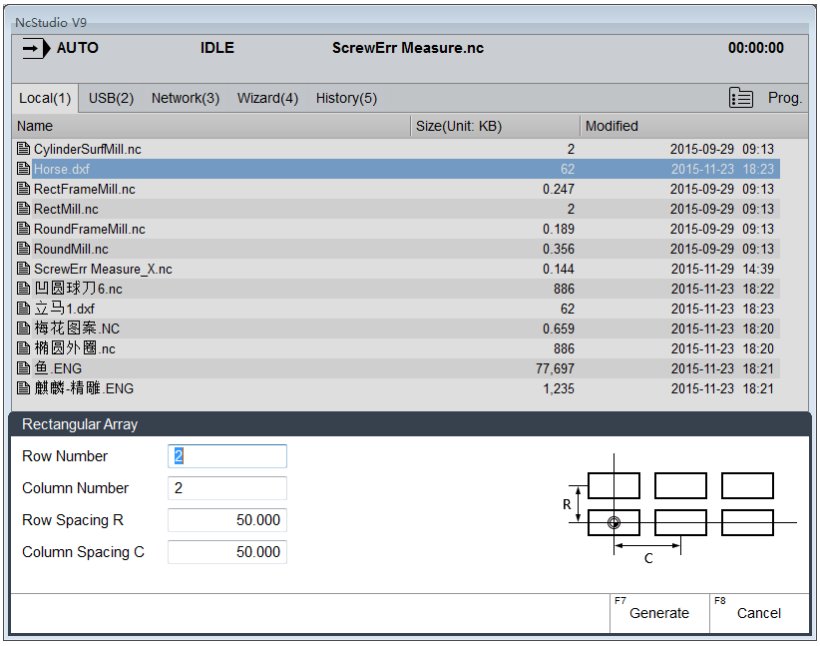


Fig. 3-73 Dialog Box "Rectangular Array"

Press F2 to enable image mirror function, as shown in Fig. 3-74. Then press key "↑" or "↓" to switch between "Mirroring Direction" and "Position Offset". When "Mirroring Direction" is selected, press "Select" key to select "X-axis" mirroring or "Y-axis" mirroring. When "Mirroring Direction" is selected, enter a value in input box "Positioning Offset". After settings, press "F7" to generate a file, whose name can be user defined. After confirmation, the new generated file will appear in the program list.

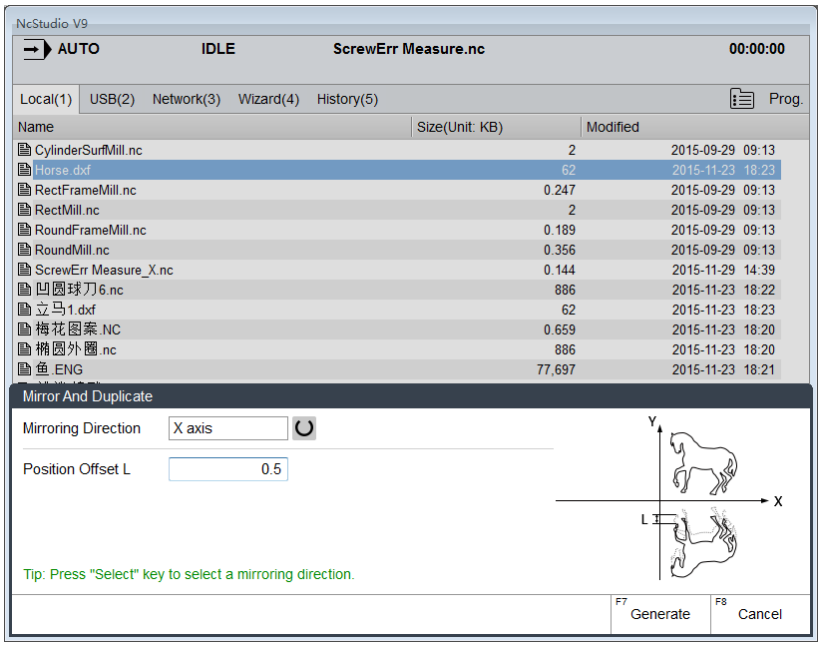


Fig. 3-74 Dialog Box "Mirror and duplication"

CAUTION

The fucntion “array” only can be applied to NC files less than 5M, and subprograms cannot exist in the file centent.

The fucntion “array” cannot be applied to commands such as “G28”, “G29”, “G65”, “M30”, “M2”, and tool path files which contains subprograms. If any of the above commands appears in a tool path file, the system will remind you to delete the command automatically of manually.

- **Unload and Copy to Removable Disk**

Press F5 to unload the currently loaded machining file, opposite to the operation of “Load”.

Press F8 (the premise is that a removable disk has already been inserted) to copy the selected file to a removable disk.

Apart from NC files, the system also supports PLT files, DXF files and ENG files.

- **New**

Press F6, and the system will create a .nc file under the path D:\NCFILES with the default file name “Untitle1.nc”. The system will then automatically enter the program editor for your programming.

- **Rename**

After selecting a machining file, press F7. A file name input box will pop up. After entering the new name, press “OK” to complete the operation.

3.16.3 History

Press key  to access functional area [Program] and press key “5” to open interface 「History (5)」 as shown in Fig. 3-75. A maximal of 22 records of successfully loaded program files are displayed on the interface. The most recently loaded file is shown at the first place. When the record number exceeds 22, the software will automatically clear the most remote records.

Load: you can load program files into history record by pressing key “↑” and “↓” to select a record and then press F1 to load a program file as the current program file.

Unload: pressing key “↑” and “↓” to select a record and press F5 to unload the program file in this record.

Clear All: clear all the records in interface 「History (5)」.

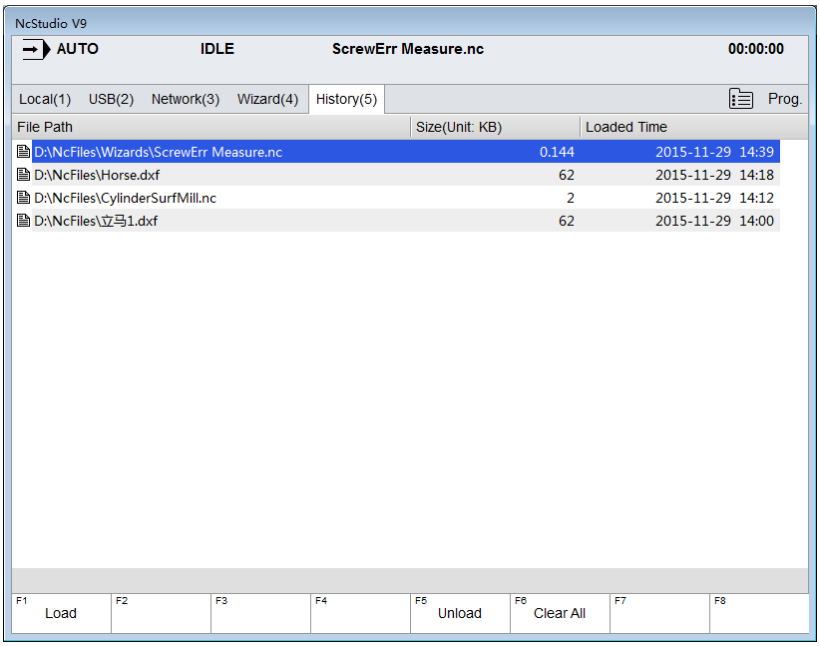


Fig. 3-75 Interface 「History (5)」

CAUTION

History records will remain after the software restarts or the system restarts caused by poweroff.

3.16.4 Parameter Specification

- **Parameters related with PLT File Translation**

Parameter		Details	Setting Range
N65000	Retract	It sets the tool lifting height during rapid traverse.	0~99999
N65001	PLT Units	Normally, 1plt=40.195mm, which can be magnified or reduced by setting this parameter.	0.001~99999
N65002	Tool Offset	To process the workpiece adequately, tool spacing set needs to make the parts between the adjacent tool paths overlapped based on the tool diameter. Tool offset here refers to the tool spacing in PLT file machining.	0.0001~99999
N65003	Cutting Depth	It specifies the machining depth for 2D files.	-99999~0
PLT file translation parameters are applied to translation of PLT files. PLT is a format of 2D machining files defined by an American company Hewlett Packard (HP), usually used in			

Parameter	Details	Setting Range
embossment and advertising carving, including such parameters as “retract”, “PLT units”, “tool offset” and “cutting depth”. At the same time, PLT is a kind of unit. Normally, 1plt=40.195mm, which can be magnified or reduced by setting the parameter N65001.		

- **Parameters related with DXF File Translation**

Parameter		Details	Setting Range
N65100	Retract	It sets the tool lifting height during rapid traverse.	0~99999
N65101	Cutting Depth	It specifies the machining depth for 2D files.	-99999~0
N65102	Layer Depth	It decides the cutting depth each time in 2D machining.	-99999~0
N65103	First Point as Origin	It sets whether to set the firstly met coordinate point as zero point when a DXF file is processed.	YES: Use the first point as zero point NO: Not use the first point as zero point
N65104	By Contour	Every time a shape is being machined, next shape will be processed only after previous one is completed.	YES: Valid NO: Invalid
N65105	Enable Bottom Cutting	Valve operation is enabled only when [3D cutting] is on the workpiece surface.	YES: Valid NO: Invalid
N65106	Use Metric	It forcibly sets a DXF file in metric size.	YES: Forcibly set in metric size NO: Not forcibly set in metric size
DXF file translation parameters are applied to translation of DXF files, including “retract”, “cutting depth”, “layer depth”, “first point as origin” and “by contour”, etc. When processing a Dxf file, the system treats the action of tool lifting as the separate mark for the adjacent shapes. If there is no tool lifting, the system will consider only one shape is being processed. If tool lifting occurs, it indicates the processing of a complete shape is finished. For example, process several circles adjacent to each other, but not overlapped. The depth of each circle is 10mm, and each feed depth of Z-axis is 2mm. If parameter N65104 is set to YES, the machine tool will process the current circle 5 times, lift its tool, and then go to process the next circle. If it is set to NO, the machine tool will process the current circle once, lift its tool, and then go to process the next circle. After all the circles are processed once, this process will be re-executed 4 times to finish processing all the shapes.			

- **Parameters related with ENG File Translation**

Parameter		Details	Setting Range
N65200	Retract	It sets the tool lifting height of Z-axis when a machine tool processes an ENG file in rapid traverse.	0~99999
N65201	Prompt for Tool Change	If it is set to YES, when tool change command is encountered, the machine tool will suspend machining and uplift its Z-axis, and the prompt bar in the system will prompt tool change. At this time, you can perform the operation of tool change. If it is set to NO, when tool change command is encountered, the machine tool will not suspend machining, but the prompt bar in the system will still prompt tool change.	YES: Valid; NO: Invalid
N65203	Cutting by Tool Number	If this parameter is set to YES, opening an Eng file will eject a dialog box asking to select a tool (the tool specified in the Eng file instead of the system default tool) for machining based on the machining program.	YES: Use; NO: Not use
N65204	Deep Hole Cutting Type	It sets the manner for processing deep holes.	0: Reciprocating chip removal 1: High-speed reciprocating chip removal 2: Up to safe height
N65205	Lifting Distance	It indicates the retract value after feed each time in the manner of high-speed reciprocating chip removal for deep hole drilling.	0~99999
These two parameters are related to processing of deep holes.			
N65206	Force To Use Tool Compensation	Yes: Use the length or diameter compensation of tools to translate ENG when selected [Enable Cutter Compensation]; No: the length or diameter compensation commands are inactive in ENG translation.	YES: Forcibly use; NO: Not forcibly use

Parameter		Details	Setting Range
N65207	Modify Tool No. in ENG File	With the function, the tool No. can be modified in machining.	YES: Enabled NO: Disabled
N65208	Z-axis Plunge Type	Starting position od Z-axis plunging.	0: From safe height; 1: From the highest point (N10030 Table Travel Upper Limit -1)
N65209	Lift when Change Tool	Lift the spindle when changing tool.	YES: Keep fixed NO: Lift to safe height
N65210	Ignore Coordination System Instruction	Ignore Coordinate system instruction in the ENG file.	YES: Enabled NO: Disabled
N65211	Z Lift Type after Drilling	Lifting types of Z-axis after drilling.	0: To R Plane 1: To specified position, exclusively of ENG file.
N65212	Z Position after Drilling	Lift Z-axis to this position when “Z Lift Type after Drilling” is set as “1”.	-1000~1000
N65213	Z Plunge Feedrate	The plunging federate of Z-axis.	0: Feedrate in machining 1: Feedrate in rapid traversing

3.17 Handwheel Operation

3.17.1 Handwheel Mode

The system supports three operation modes—auto mode, manual mode and reference point mode. Manual mode is subdivided into jog mode, stepping mode and handwheel mode. You can turn the mode selection knob on the operation panel to “handwheel”, as shown in Fig. 3-76.

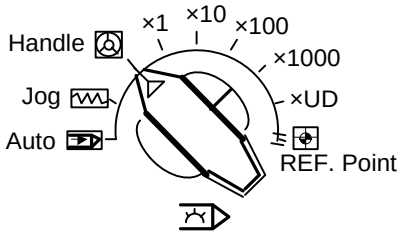


Fig. 3-76 Mode Selection Knob

In handwheel mode, you can configure a handwheel to control the machine tool. As shown in Fig. 3-77, select a motion axis by rotating “Axis Selection Button”, select handwheel override gear by rotating “Gear Selection Button”, and control the selected axis running at the selected handwheel override gear by rotating “Handwheel Control Rotation Disk”. Handwheel override gear regulates the displacement

(linear displacement or rotation angle) of moving parts of a machine tool with each case turning of a handwheel. The displacement is set by parameters N52003, N52004 and N52005.

Fig. 3-77 A Handwheel

- **Related Parameters (under [Machine/Controller])**

Parameter		Details	Setting Range
N52001	Precise Pulse Counting	When the parameter is set as valid,the moving distance of the machine tool corresponds to handwheel counts strictly.	YES: The machine moves the exact pulses generated by handwheel; NO: The machine stops when handwheel stops turning
If the parameter is set as valid, the machine tool will remain moving for a long time after the handwheel stops which rotates too fast before stopping. The driver will receive all pulse signals sent by the handwheel. However, if the parameter is set as invalid, the react time for the rotation of the handwheel is shorter, while the distance the machine tool moves will not correspond to the handwheel indicated when the handwheel rotates too fast.			
N52002	Handwheel Direction	Positive/negative motion direction of a machine tool when turning a handwheel	1: Maintain the original machine motion direction in handwheel turning -1: Reverse the original machine motion direction in handwheel turning
N52003	Multiple at X1	The system will interpret 1 pulse is received when a handwheel sends 1 pulse.	0.001~10
N52004	Multiple at X10	The system will interpret 10 pulses are received when a handwheel sends 1 pulse.	0.001~10

Parameter		Details	Setting Range
N52005	Multiple at X100	The system will interpret 100 pulses are received when a handwheel sends 1 pulse.	0.001~10
N52010	Handwheel Acceleration	It sets the acceleration during handwheel mode (the smaller the value is, the more stable the motion will be).	1~1000
N52012	Deceleration when Switching Axis	If set to “YES”, oscillation of a machine tool may be reduced, but over-travel may occur. Otherwise, oscillation of the machine tool may occur.	YES: Decelerate; NO: Not decelerate
N52030	HW Connection Code	Control the HW Connection Code through setting parameters	0: To terminal Board 1: To operation Panel

3.17.2 Handwheel Guide

NK300CX system supports handwheel guide function.

Handwheel guide refers to a way of operation that the automatic execution speed of a machining program is manually controlled during auto processing so as to guard against dangers caused by a wrongly loaded program or an inappropriate tool path.

In Auto mode, press the handwheel guide key 

on the operation panel. If the top-left indicator on, it means the function is activated. After machining starts, the system will execute the machining file with clockwise turning of the handwheel and stop machining with the stop of the handwheel. Machining speed varies with the handwheel turning speed.

NK300CX also holds the function of handwheel reverse guide. Turn the handwheel anticlockwise when an error is found in machining to make the machine tool reverse along the previous machining track. If you do not need the function, you can change the value of parameter N52013 “Forbid HW Reverse Guide” into “YES”, i.e., to forbid handwheel reverse guide function. When you turn the handwheel anticlockwise, the machine tool cannot reverse along the previous machining track and will not move.

- **Related Parameters (under [Machine/Controller])**

Parameter		Details	Setting Range
N52006	HW Lead Gear (Numerator)	This ratio is used to control the feed speed of a machine tool in handwheel mode.	1~1000
N52007	HW Lead Gear (Denominator)		

Parameter		Details	Setting Range
N52013	Forbid HW Reverse Guide	YES: Axis stops when HW is turning reversely in HW Guide; NO: Axis moves normally when HW is turning reversely	YES: Forbid NO: Do not Forbid

3.18 System Management

To access the system info functional area, press key  . In interface 「System (1)」 of functional area [System], you can view system info, proceed with system maintenance and switch configurations.

3.18.1 System Info

In functions area 「System」, the default interface displays system info, including CNC software info and hardware info, as shown in Fig. 3-78.

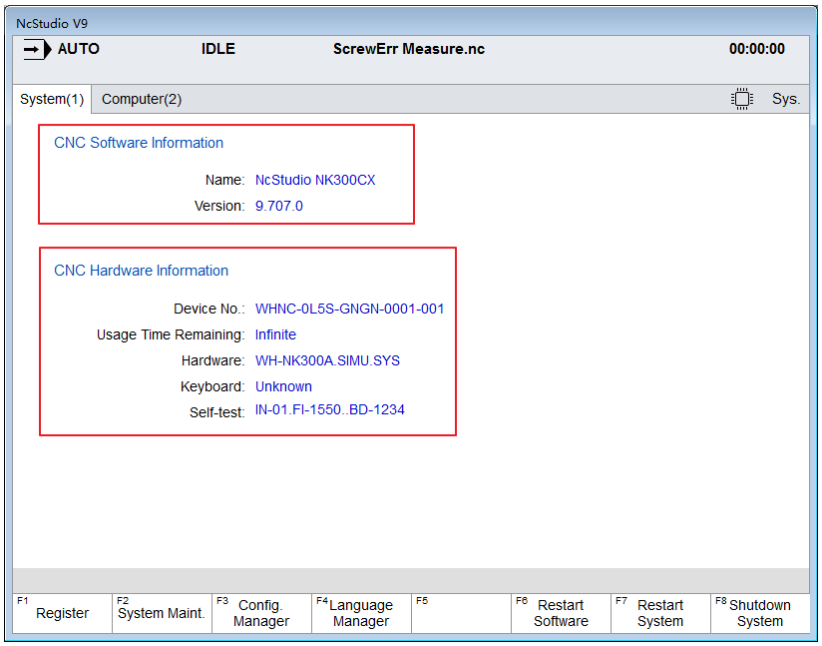


Fig. 3-78 Interface 「System (1)」

Press F1 to register the time you use the software. For more details about registration, refer to section 3.18.4.

3.18.2 Configuration and Language Setting

1) Configuration Setting

NK300CX currently supports three-axis configuration, four-axis configuration and five-axis configuration. In addition, you can add other configurations according to your needs. Press F3 in interface 「System

(1)」, the prompt box popping up is about manufacturer access detection. You can verify your access by entering your manufacturer password. Once your access is verified, dialog box [Configuration Manager] will pop up. You can select the configuration in the box according to your needs. The current configuration is shown in blue at the bottom of the box. See Fig. 3-79.

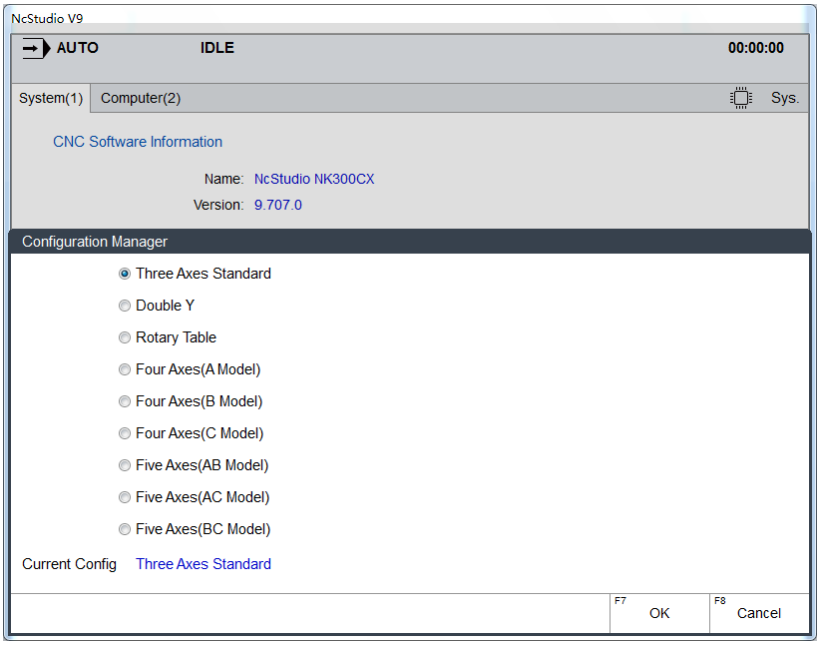


Fig. 3-79 Dialog Box 「Configuration Manager」

To switch to other configurations, press " ↑ " or " ↓ " to move the cursor to the desired one, and then press F7 to confirm. A dialog box asking "Set successfully!" will pop put. It takes effect after restarting the system. Would you like to restart the system immediately?" Press F6 and the system will be restarted to enable configuration switchover. After system restarted, you need to set relevant parameters again.

- **Three Axes Configuration**

Three Axes Standard: the machining axes are X-axis, Y-axis and Z-axis.

Rotatory Table: the machining axes are X-axis, Y-axis and Z-axis, and Y-axis is rotatory table.

Double Y: the machining axes are X-axis, Y1 axis, Y2 axis and Z-axis

- **Four Axes Configuration**

Four Axes (A Model): the machining axes are X-axis, Y-axis, Z-axis, and A-axis.

Four Axes (B Model): the machining axes are X-axis, Y-axis, Z-axis, and B-axis.

Four Axes(C Model): the machining axes are X-axis, Y-axis, Z-axis, and C-axis.

- **Five Axes Configuration**

Five Axes (AB Model): the machining axes are X-axis, Y-axis, Z-axis, A-axis and B-axis.

Five Axes (AC Model): the machining axes are X-axis, Y-axis, Z-axis, A-axis and C-axis.

Five Axes (BC Model): the machining axes are X-axis, Y-axis, Z-axis, B-axis and C-axis.

2) Language Setting

Press F4 in functional area [System] to access a dialog box "Language Manager". At present, there are two languages for option—"Chinese" and "English". You can run the system in a familiar language by

pressing “ ↑ ” or “ ↓ ” to move the cursor onto the desired language and then pressing F7 to confirm. The system will then give a prompt “Succeeded! Restart the software to take effect.” Press F7 to restart the software.

3.18.3 IP Setting

Press key  to enter functional area [System], and then press key “2” to view computer information and set network.

NK300CX supports network connection. You can obtain IP address automatically or set it manually.

- **Auto Obtain IP Address**

DHCP function is enabled to achieve auto obtain. In sub-interface 「Computer (2)」, press F1 to open dialog box “Network Setting”. Press key “Select” to make the software obtain IP address automatically. See Fig. 3-80.

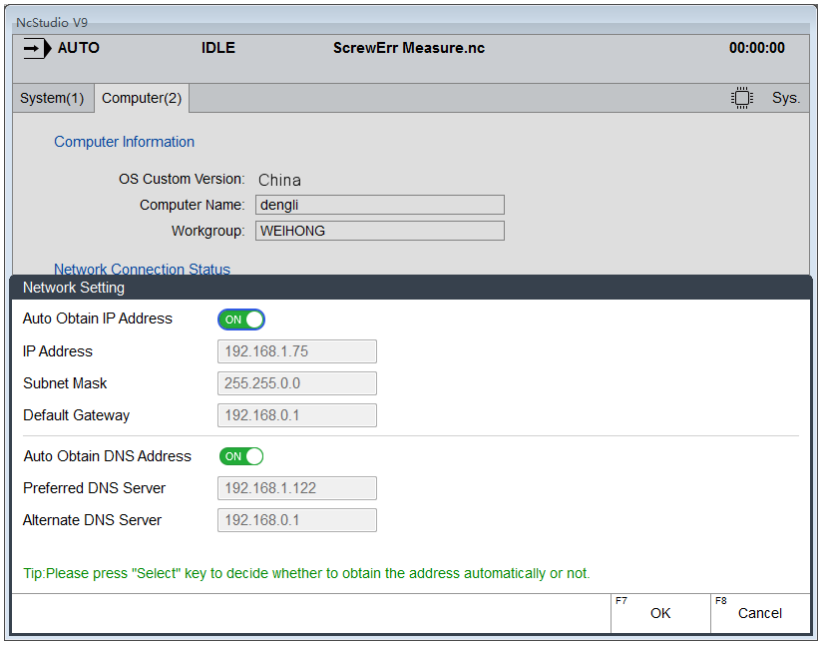


Fig. 3-80 Auto Obtain IP Address

The system will obtain an IP address automatically when “Auto Obtain IP Address” is set as “ON”.

- **Manually Set IP Address**

In Fig. 3-80 , press letter key N to manually set the IP address.

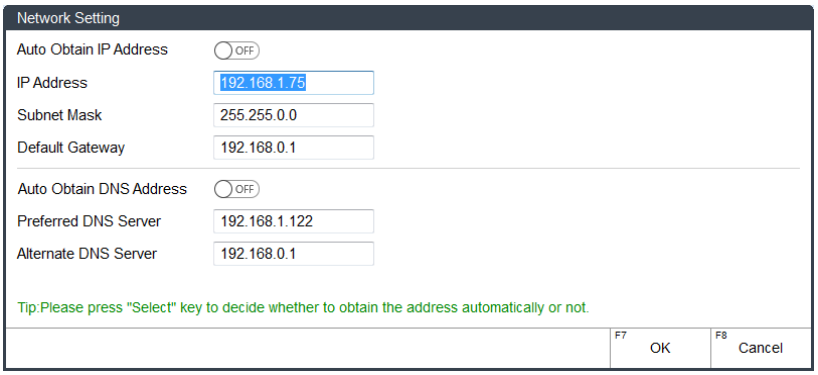


Fig. 3-81 Manually Set IP Address

When “Auto Obtain IP Address” is set as “OFF”, you have to set IP Address manually. Press key “↑” and “↓” to select input boxes and enter the IP address.

- IP address: 192.168.1.188 (within the same range of that of the computer)
- Subnet Mask: 255.255.255.0 (same as that of the computer)
- Default Gateway: 192.168.1.1 (same as that of the computer)

After setting, press F7 for confirmation. And you can turn to the interface 「Computer (2)」 to view the setting.

CAUTION

Please note that manual setting of NK300CX IP will reset the IP of the computer.

3.18.4 Registration

“Register” in interface 「System(1)」 in functional area [System] is used to decide system service time with the help of a registration code generated in APP “NcStudio Generator” launched by Weihong Electronic Technology Co. Ltd.. The registration steps are as follows.

3.18.4.1 “NcStudio Generator”

You can search Apple APP Store for “Weihong”, find and install APP “NcStudio Generator”. Make sure you have already filled and stamped the fax file “APP Registration Info Confirmation Letter” sent rom Weihong and have returned it to Weihong before using the APP. Weihong will record the information in the confirmation letter you have returned.

- **Bind your mobile phone number to the APP**

You must bind your mobile phone number to the APP before using it. A dialog box shown as Fig. 3-82 will pop up when you launch the APP for the first time. Click on “OK” to close the box. Then click on “Bind” in the lower left of the interface. Fill in the information to bind your number to the APP in the interface as shown in Fig. 3-83.

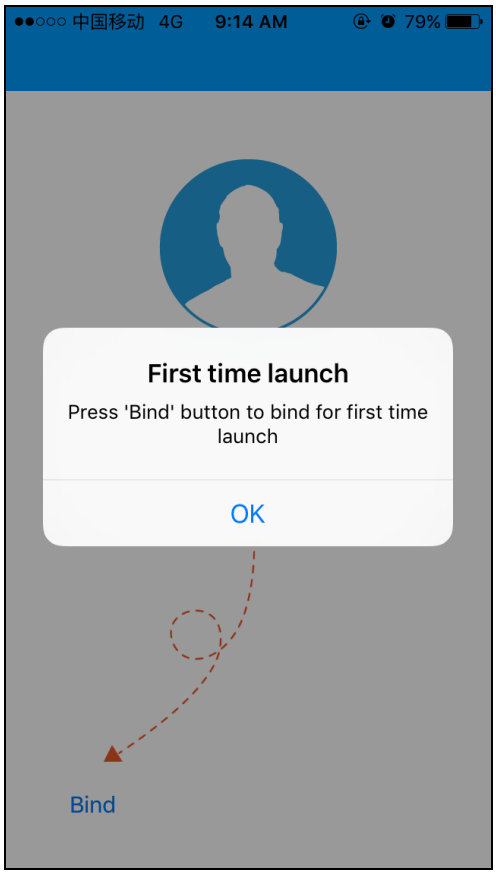


Fig. 3-82 Bind(1)

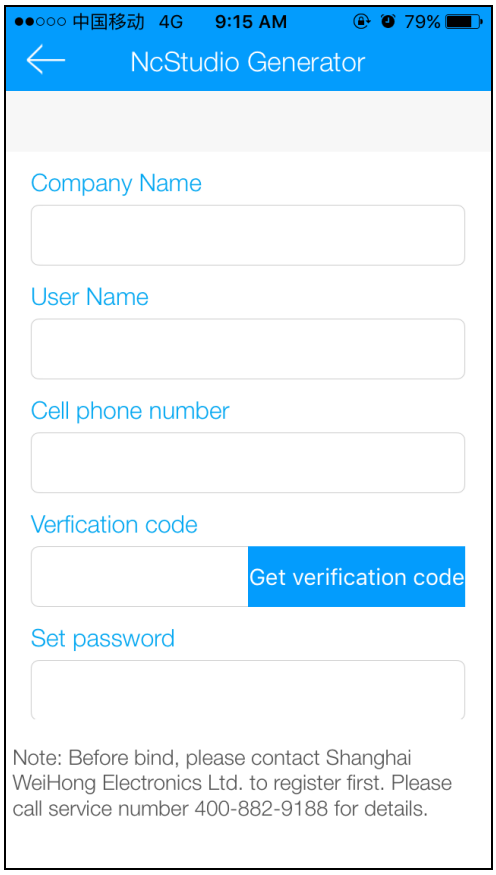


Fig. 3-83 Bind (2)

CAUTION

1) Your company name, user name, cellphone number must be the same as recorded by Weihong. Otherwise, you can't register successfully.

2) The "set password" must be 6 digits or characters. Repeat the password in the next line, otherwise the password cannot be set successfully.

- **Login**

You can login with the phone number you have bound to the APP. Click "Verification code" in the interface as shown in Fig. 3-84. The APP will send a short message containing a verification code to you. And a countdown from 59 to 0 is shown in the interface. The prompt info "Please input verification code" will turn into "Time out, please request code again". After receiving the code, input the code and then the password in the interface, as shown in Fig. 3-85, to login.

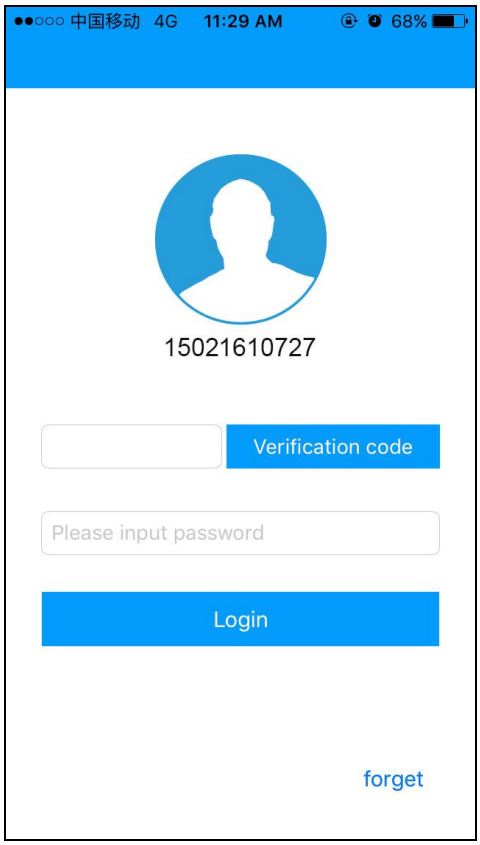


Fig. 3-84 Login-1

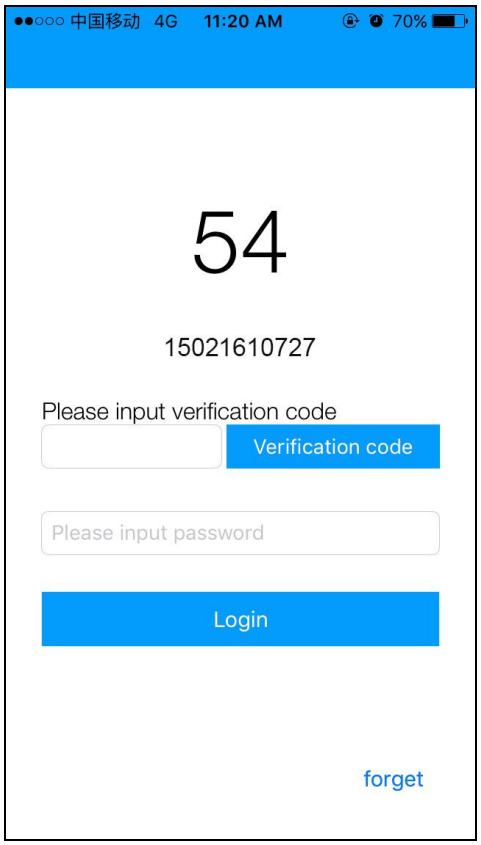


Fig. 3-85 Login -2

If you forget your password, please contact with Weihong Electronic Technology Co. Ltd. Click on “forget”, and a prompt message containing the official service number of Weihong will appear on the screen, as shown in

Fig. 3-86.

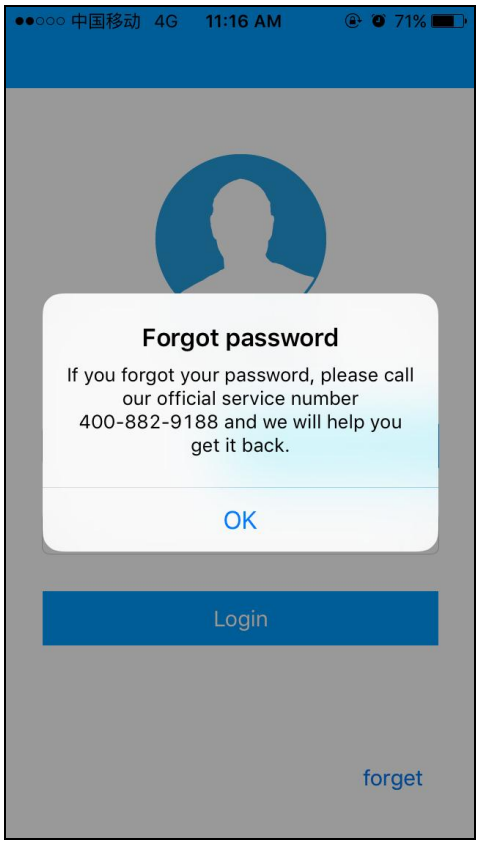


Fig. 3-86 Forget Password

- **Function Page**

After login, you will enter the function page. Your company name will be shown under the navigation bar automatically.

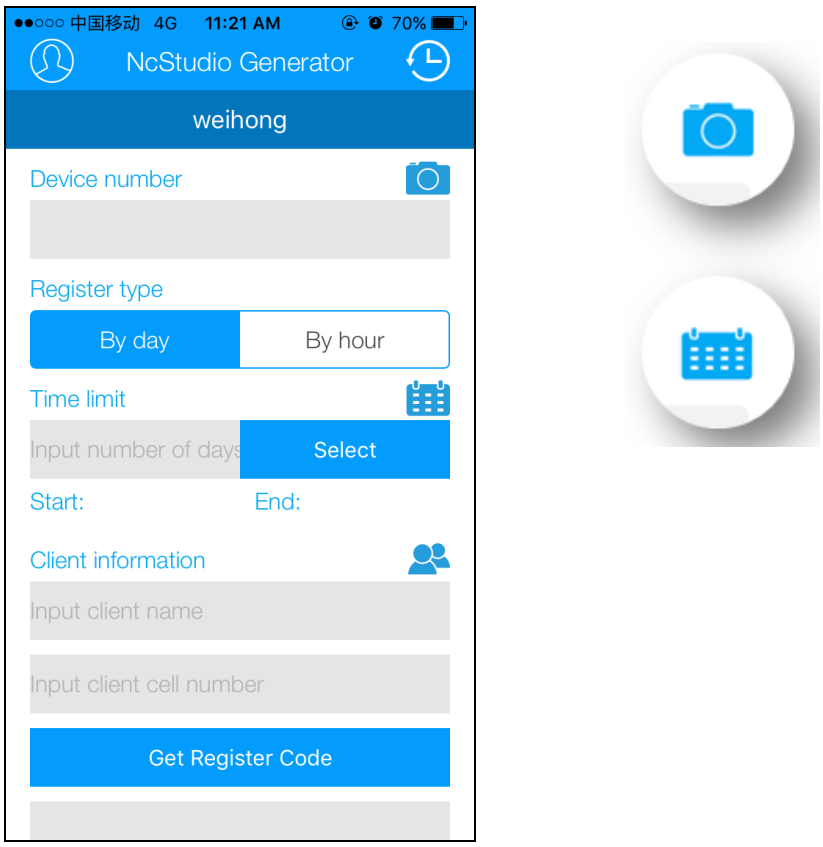


Fig. 3-87 Function Page

Device number: you can enter the device number here directly, or click the camera icon to open the camera on the phone and scan the device number on the device. However, the scanning function is not supported by now.

Register type: If you select “By day” as registration type, you can click on “Select” under “Time limit” to choose a time limit from range “1 week, 1 month, 6months, 12 months, permanent; or you can click on the calendar icon to choose specific days. Register time is calculated from the day you register. If you choose registering by hour, you can enter the time in the input box under “Time limit”.

Get Registration Code: click on “Get Registration Code”, and the grey box under “Get Registration Code” will load until the writing code appear in the box automatically. When the grey box is loading, three continuously dynamic dots load repeatedly.

Send SMS: click on “Send SMS”, and the interface will jump to interface “Edit SMS”. The writing code you have received will be added to the SMS automatically.

CAUTION

The system supports registering by hour or by day. If you choose to register by day, service time will be counted according to system internal clocking, no matter the system is power off or not. And if you choose to register by

hour, service time will be counted according to system internal clocking. However, after the system is power off, the service time will not get less until the system is power on.

- **User Information**

Click the head icon in the upper left of the navigation bar in function page to open user information page, as shown in Fig. 3-88.

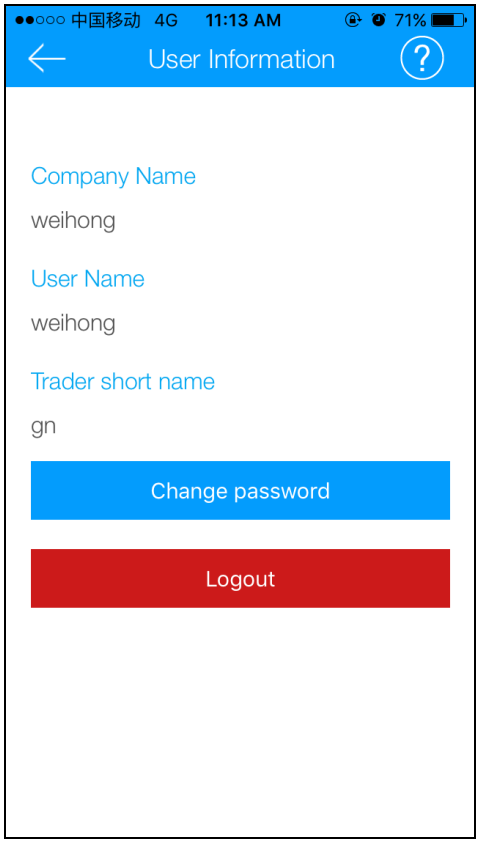


Fig. 3-88 User Information Page

- **History**

Click the clock icon in the upper right of the navigation bar in function page to open user information page, as shown in Fig. 3-89.

Fig. 3-89 History Page

3.18.4.2 Registration with Registration Code

After obtaining a registration code in the APP, you can register in NK300CX.

When the machine tool is in non-machining state, i.e., idle or E-stop state, you can register as the steps below. Do not register when the machine tool is in machining state, i.e., machining or stop state. Otherwise, the software will warn that “Unable to perform the action under the current mode”.

The steps to register are as follows:

1) In interface 「System (1)」 under functional area [System], view the current device number. Or you can view the device number in dialog box “Register” after pressing F1 in the interface 「System(1)」, as shown in Fig. 3-90. The number is also the device number shown in Fig. 3-87.

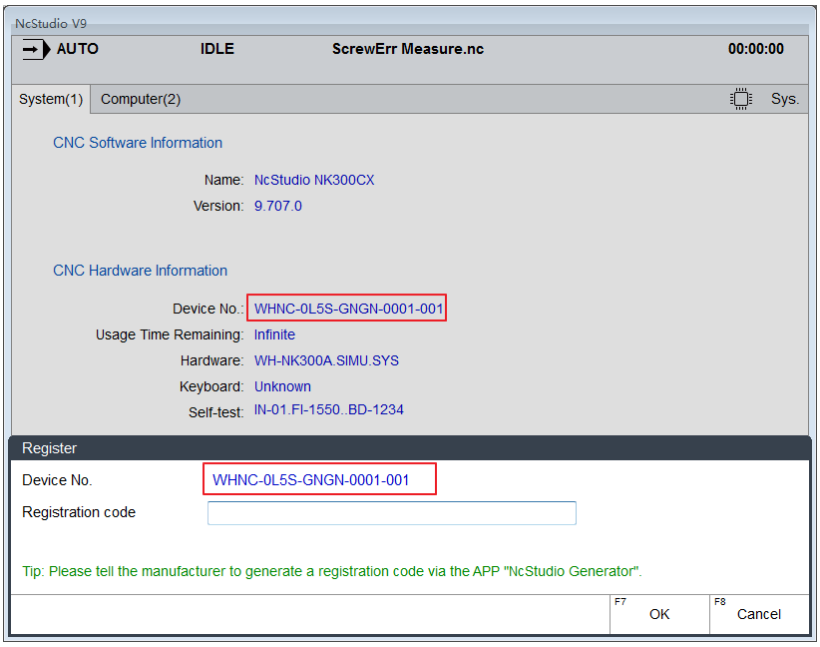


Fig. 3-90 View Device Number

2) Input the registration code generated in the APP into the input box “Registration code” in dialog box “Register”, as shown in Fig. 3-90. Then press F7 to confirm.

3) The system prompts “Register successfully. Please restart software!” Restart the machine. After restart, you can view the current left time for use in interface 「System (1)」 in functional area [System].

CAUTION

ID of board card varies as the registration times increases, which can be tell from the last three number of the serial number. For example, when registration times is 0, the last three number of the SN is 000. While when registration times is 1, the last three number of the SN is 001.

After registration, the normal service time of the software is limited. When the service time is nearly expired or already expired, different prompt info will pop up according to the current registration state and service time. And please note some functions will be disabled when the software expired. An introduction of software when it is expired or will be expired soon is shown as below.

- **Registration Time Close to Expiration Time**

Software State	Time Left	Software Tips		Remarks
		Sign of Tips	Content of Tips	
Open	Longer than 7 days		No tips.	
	1~7 days	A dialog box popping up	“The software will expire after n d. Please contact with the manufacturer!”	
	Less than 1day			

Software State	Time Left	Software Tips		Remarks
		Sign of Tips	Content of Tips	
Running	Longer than 7 days		No tips.	
	4~7 days	Yellow Warning Sign.	"The software will expire after n day(s)." Prompt every 6 hours.	Any other warning will clear the tip. And you can clear it by pressing any key.
	1~3 days		"The software will expire after n day(s)." Prompt every 2 hours.	
	Less than 1 day		"The software will expire after n day(s)." Prompt every hour.	

- **The software expires.**

1) When you open the software,

Software interface will not show. Only a dialog box reminding users to register will pop up.

2) When the software is running,

In non-machining state, the machine tool will enter expired state. A red warning "The software is expired. Please contact with the manufacturer!" will appear in the info bar. And the warning will not disappear until e-stop info covers it.

If the software expires during machining, the system will stop machining immediately and enter expired state. A red warning "The software is expired. Please contact with the manufacturer!" will appear in the info bar. And the warning will not disappear until e-stop info covers it.

If the software expired, commands such as "Start", "Advanced Start", and "Resume" are disabled, but manually moving the machine tool is enabled.

3.19 Auxiliary Function

3.19.1 Single Block Execution

You can set the machining task to be executed in single step mode, facilitating error diagnosis and failure recovery. Once in single block mode, the system stops machining when resultant velocity of each axis is "0".

When the single block key on the operation panel is pressed, the system will only execute the machining file for one line each time the START button is pressed, and then enter into the pause state. To go to the next line, you need to press the START button again.

3.19.2 Back to Workpiece Origin

The origin of WCS (workpiece coordinate system), i.e. workpiece origin, is fixed with respect to a certain point on the workpiece, while mobile with respect to machine origin. The selection of workpiece origin should meet the demands of simple programming, easy dimension conversion and small caused machining error, etc.

To back to workpiece origin, press the general function selection key  , and then press F7 to make the spindle return to workpiece origin automatically from the current position.

3.19.3 Jiggle

If machining is found not in position in machining, suspend machining and execute manual jiggle. Jiggle result is only available for the current machining task, and becomes ineffective after machining stops.

Jiggle function can be found by pressing key  ,  and F6. After pressing key  in auto machining, press F6 to access the jiggle interface, as shown in Fig. 3-91. Press key “Select” to set a proper step. The system provides step values including 0.01, 0.05, 0.1, 0.2, 0.3, 0.4 and 0.5. And then press an axis direction key to jiggle the corresponding axis. After satisfying jiggle result is obtained, press key “START” to continue machining.

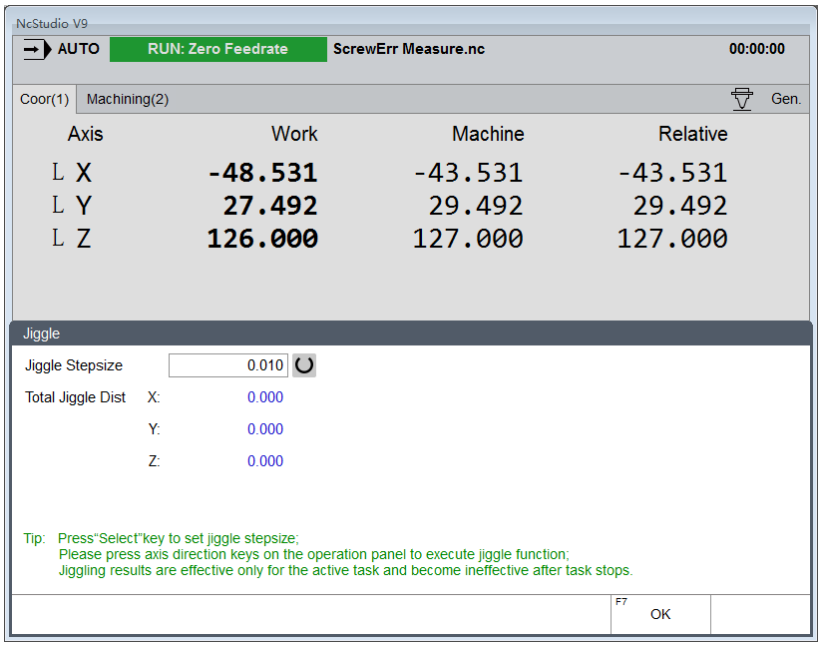


Fig. 3-91 Jiggle Setup

The steps to execute “Jingle” operation during machining are similar to those described above. And machining will not break during jiggle.

CAUTION

In manual mode, function “jiggle” is disabled and the “jiggle” button is inactive in grey.

3.19.4 Advanced Start

Function “Advanced Start” is used for select any blocks for machining, and it is classified into advanced start with selected lines and advanced start with selected tools.

In advanced start with selected lines, you can enter the start line number and end line number of a machining program block. Press key “Start”, and the system will machine from the start line to the end line of the program block and stop machining.

In advanced start with selected tools, you can enter a tool number. After finding the tool number, the program block will set the line where the tool number appears as the start line. Press key “Start”, and the system will start machining from the “start line” to the end line of the machining program and stop machining.

Press key  to access functional area [Machining], and press key “1” to open interface 「Coor(1)」. Then press F3, dialog box “Advanced Start” will pop up, as shown in Fig. 3-92. Press key “↑” and “↓” to switch between input boxes.

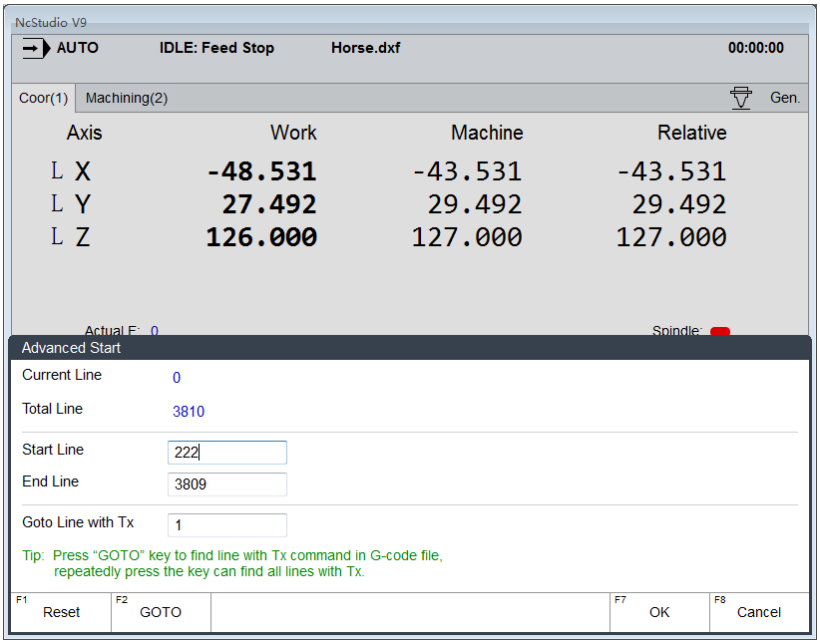


Fig. 3-92 Dialog Box of Advanced Start

Current Line: the line number of the current machining program command line.

Total Line: the number of total command lines in the program file.

Start Line: you can enter the start line of a machining program block. Effective range is {0, end line number}

End Line: you can enter the end line of a machining program block. Effective range is {start line number,

the maximal line number}

Goto Line with Tx: you can enter the tool number which will execute machining. The default for the item is the current tool number. If the tool number of the current spindle tool is 1, the default will be 1. And if the tool number of the current spindle tool is 2, the default will be 2.

Reset: if you press button “F1 Reset”, the system will automatically reset the start line number as “0” and end line number as the maximal line number.

GOTO: press button “F2 GOTO” after entering tool number, and the system will find the program line where the tool number appears and set it is the start line. At the same time, the info bar will show the result of finding. Press the button repeatedly to find different lines where the tool number exists in a program file.

After machining settings, you can view the start and end line number to be machined in the file name on each interface and the upper left part of interface 「Machining(2)」.

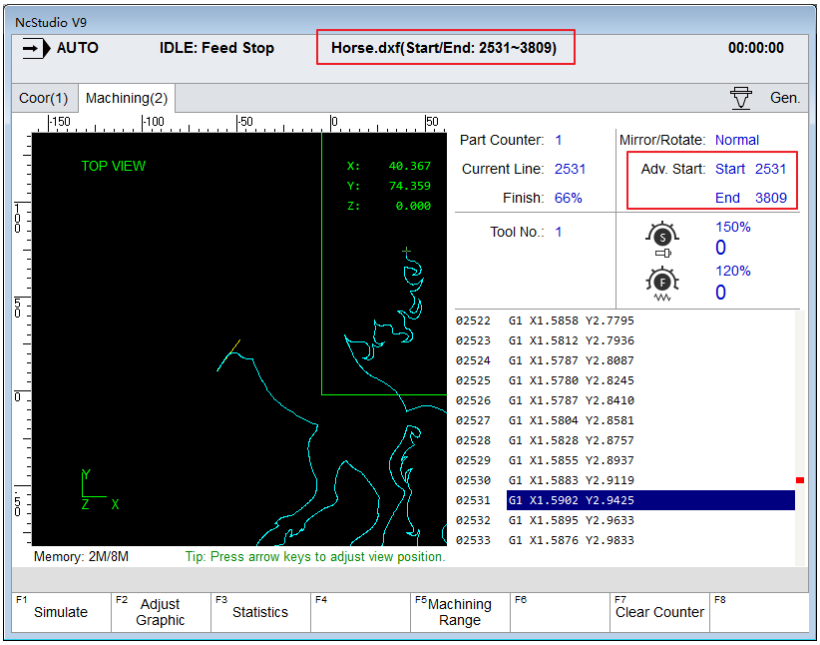


Fig. 3-93 View the Start and End Line Number of Advanced Start

CAUTION

Function “GOTO” can only be used for G-code program file.

3.19.5 Mirror and Rotation

This function can execute mirror and rotation on a machining file.

Mirror and rotation function can be set on the interface 「Machining」 by pressing key  and  . Press F4 to open the mirror and rotation machining dialog box, as shown in Fig. 3-94. Press “Select” button to select a machining mode and set the coordinate of the rotate centers on X\Y-axis. If

rotate centers are not set, the default rotate center is the workpiece origin.

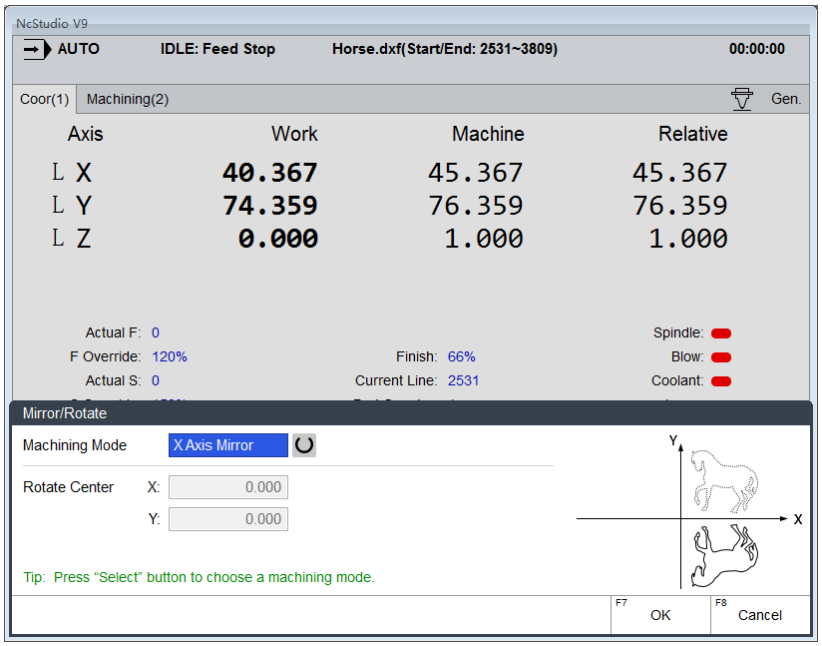


Fig. 3-94 Mirror and Rotate Setting on the Main Interface

After setting mirror and rotate information, you can check the information in the file names in all interfaces and in interface 「Machining (2)」, as shown in Fig. 3-95.

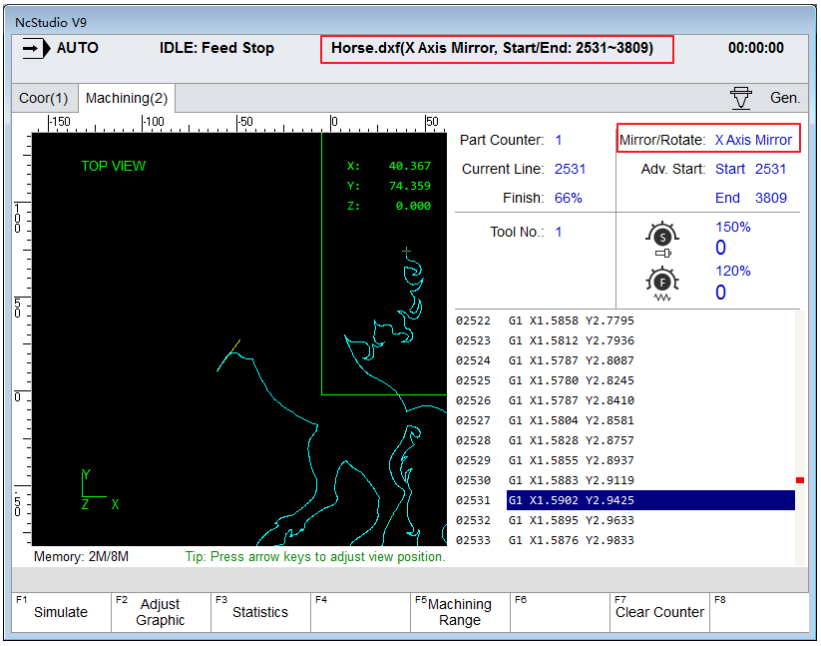


Fig. 3-95 Mirror and Rotation Interface

CAUTION

1) Under maching mode "Normal", "Mirror in X-axis" and "Mirror inY-axis", the value of rotation centers acannot be modified and in grey.

2) In manual mode, function “Mirror/Rotation” is disabled and the button is in grey.

3.19.6 Breakpoint Resume

This function is executed by pressing the breakpoint resume key on the operation panel, which will make the system resume machining from the stop line number of last time machining automatically.

If power failure or emergency stop occurs during machining, and you are sure about the accuracy of the workpiece coordinates, you can execute this function to make a machine tool rapidly move to the breakpoint and resume machining, to save machining time.

3.19.7 Workpiece Length Sensing

Workpiece length can be sensed by the system.

The system senses workpiece length by workpiece coordinates. For instance, to sense the workpiece length in the X direction, the steps are as following:

1) Press key  ,  and F1 to open the sub-bar 「Coor Display」;
2) Manually move the X-axis to one side of workpiece, and then press “F1” to switch to “Relative” Coordinate and press “F2” to clear the relative coordinate on X-axis.
3) Manually move the X-axis to the other side of workpiece. Workpiece length in X direction equals the X-axis “Relative” value displayed on the interface.
4) Workpiece length sensing of other axes is similar to the steps described above.

Clearing relative coordinates function has no effect on absolute coordinates and machine coordinates, so you can still use the original coordinates for machining.

3.19.8 Restore Parameters

The system boasts the function of parameter auto backup. If you forget to save the set parameters, you can switch to this screen, in which you can restore parameters from the ex-factory date to system last shutdown.

Press key  ,  and F1 to open dialog box “Restore Parameter”, as shown in Fig. 3-96. Press key “↑” or “↓”to select active backup parameters and press F1 to restore the selected parameters.

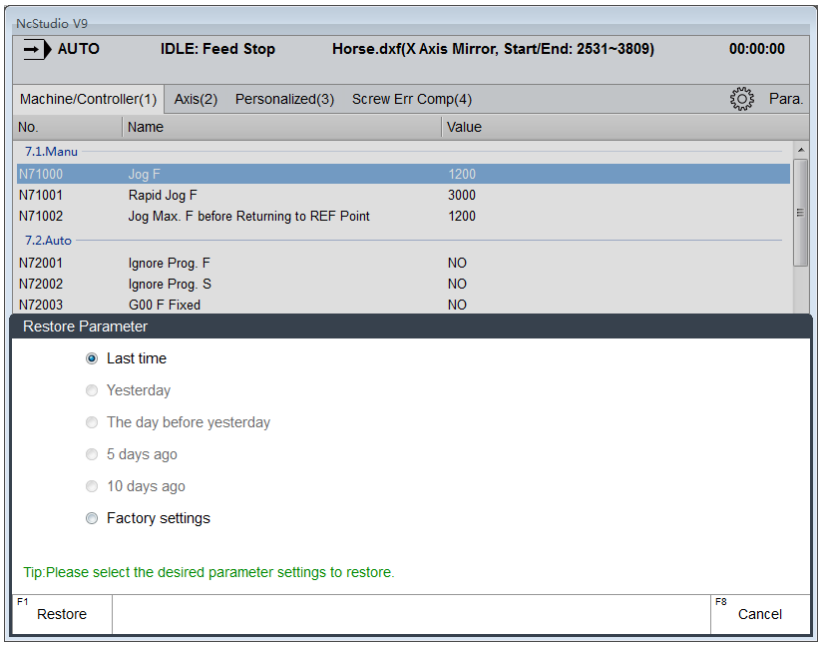


Fig. 3-96 Dialog Box “Restore Parameter”

3.19.9 Manual Data Input (MDI)

You can enter and execute commands freely in functional area [MDI].

Press key  and then press F8 in interface 「Coor(1)」 to access the MDI interface, as shown in Fig. 3-97

In the middle of the interface are command lines. The current line is highlighted in blue. Press key “↑” or “↓” to select a line and enter your commands in the input box. After entering the new commands, press key “Enter” to confirm your input.

The newly entered instruction is at the top. You can press “F1”~”F8” to execute commands in the corresponding input box.

Press key  next to exit the dialog box of MDI.

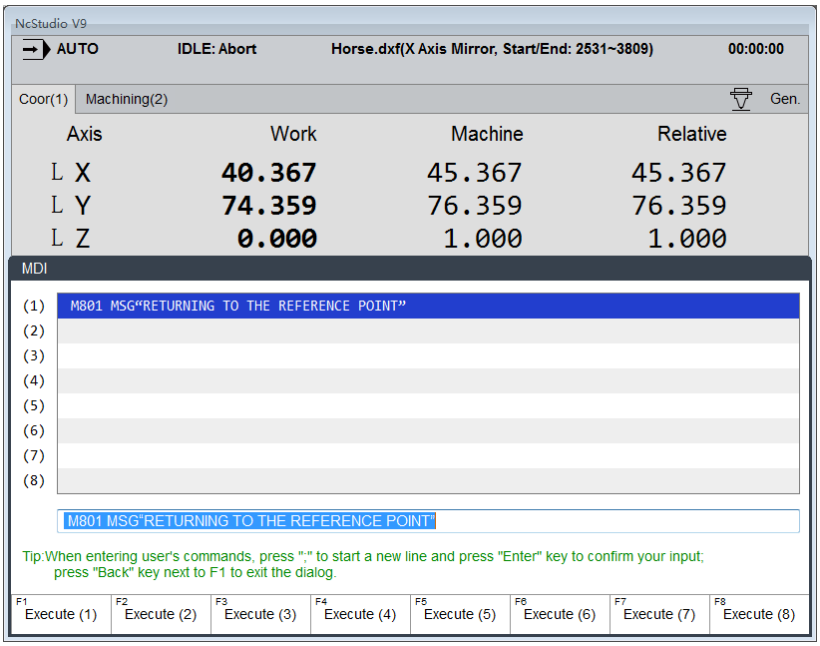


Fig. 3-97 Manual Data Input Screen

3.20 Tool Magazine

Tool magazine, one of key component of ATC unit, is used to store tools and can be designed to different types according to capacity and tool getting method. At present, three types of tool magazine are supported for NK300CX, namely, linear tool magazine, disk-type tool magazine and servo tool magazine.

There are two methods to change a tool, that is, manual and auto tool change. For machines without ATC unit, manual tool change is the only option; while for those with ATC unit, both manual and auto tool change are available, and auto tool change is better in terms of efficiency.

To activate auto tool change function, you need to specify the type of tool magazine first by setting parameter of N66031 “Tool Magazine Type” and N66030 “Automatic Tool Measure”.

3.20.1 Auto Tool Change for A Linear Tool Magazine

With relatively simple kinematics, linear tool magazine stores tools in the form of array. Procedure of tool change may vary slightly due to different magazine-machine locations. In general software, the system takes the linear tool magazine parallel to X-axis by default, as shown below.

Linear tool magazine
(The system takes linear tool magazine parallel to X-axis by default)

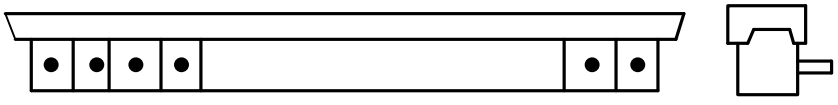


Fig. 3-98 An Example of Linear Tool Magazine

For example, if a customer has 12 tools, he can select a 1-line 12-row tool magazine, or a 2-line 6-row tool magazine, etc. Auto tool change is realized by programming in the public.dat file according to the related information learned from the customer. The process of auto tool change for a linear tool magazine is as following:

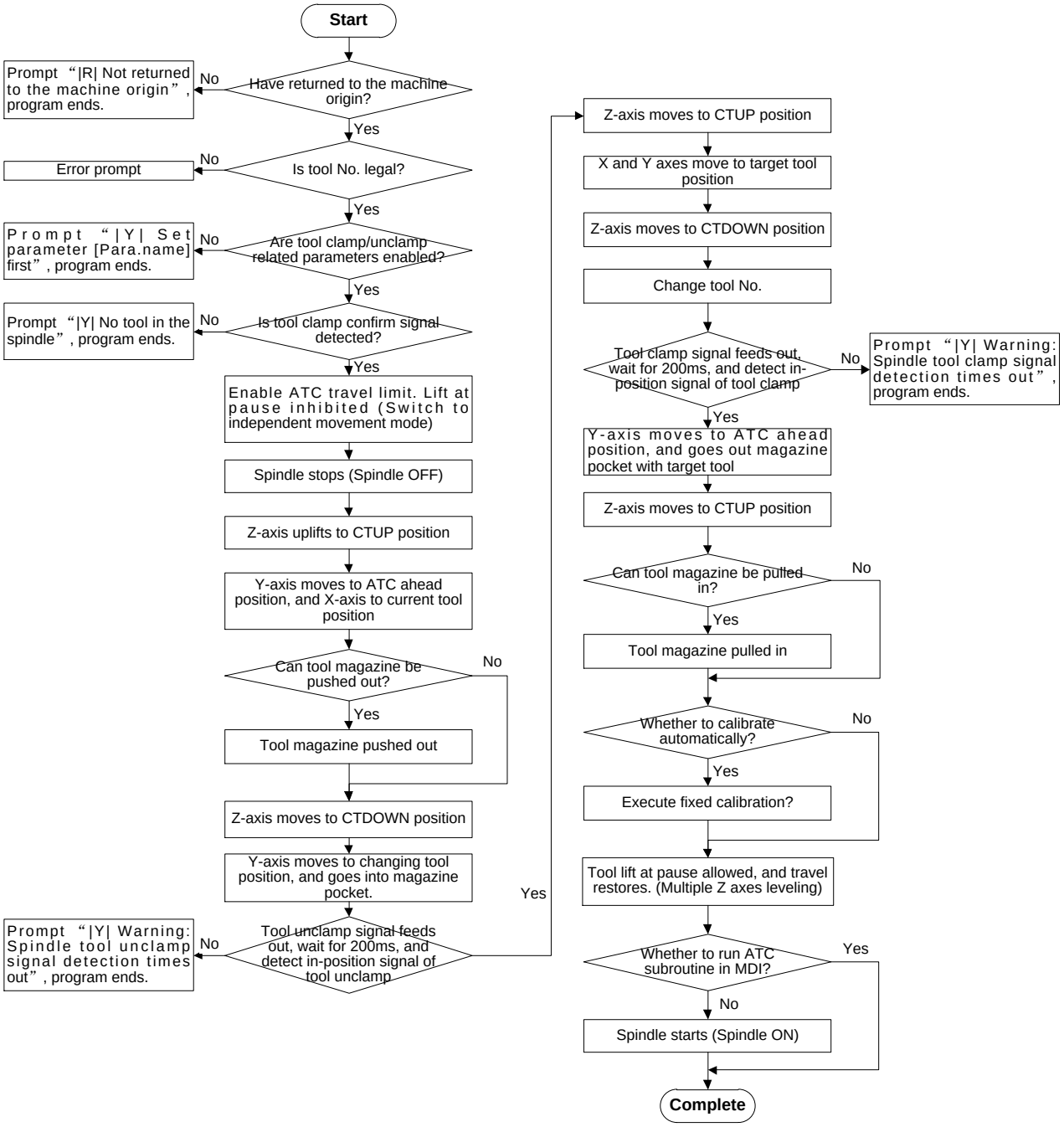


Fig. 3-99 Process of Auto Tool Change for A Linear Tool Magazine

3.20.2 Auto Tool Change for A Circular Tool Magazine

Circular tool magazine, or called disk-type tool magazine, stores more than 8 tools and occupies large space on most occasions. It is usually mounted on the upper end of the column or side of the spindle. In general software, the system takes circular tool magazine mounted on X-axis by default, as shown below.

Circular tool magazine
(The system takes the tool magazine fixed on X-axis by default)

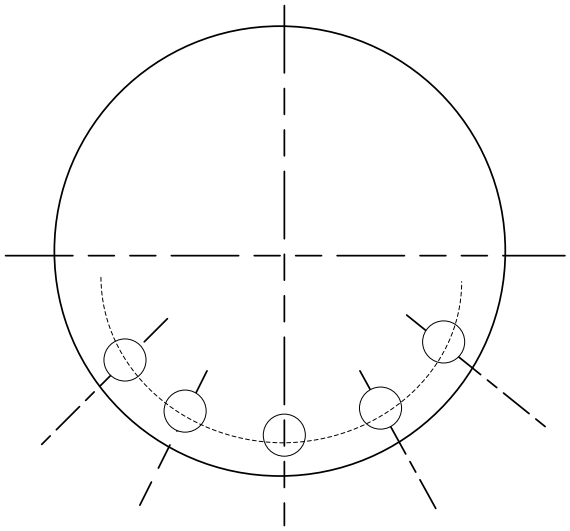


Fig. 3-100 An Example of Circular Tool Magazine

When a machine tool is with the function of a circular tool magazine and auto tool change is needed during file machining, the process of auto tool change is as following:

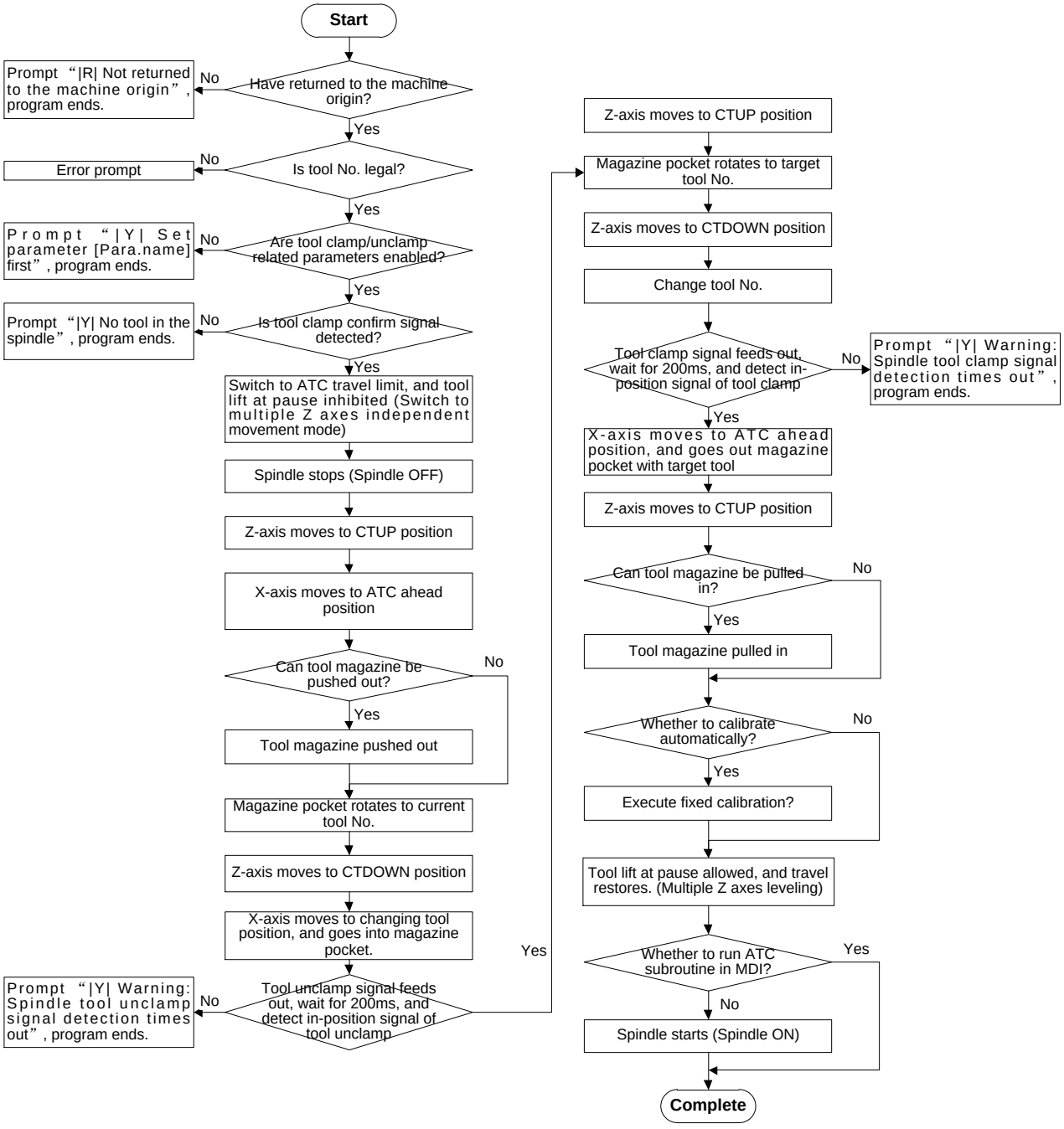


Fig. 3-101 Process of Auto Tool Change for A Circular Tool Magazine

3.20.3 Auto Tool Change for A Servo Tool Magazine

Servo tool magazine stores tools in form of disk-type magazine, with 16 tools distributed evenly. Rotation of the magazine is controlled by A-axis, whose rotation is controlled by servo motor. In general software, the system takes the magazine mounted on X-axis by default.

The process of tool change of servo tool magazine is shown as below:

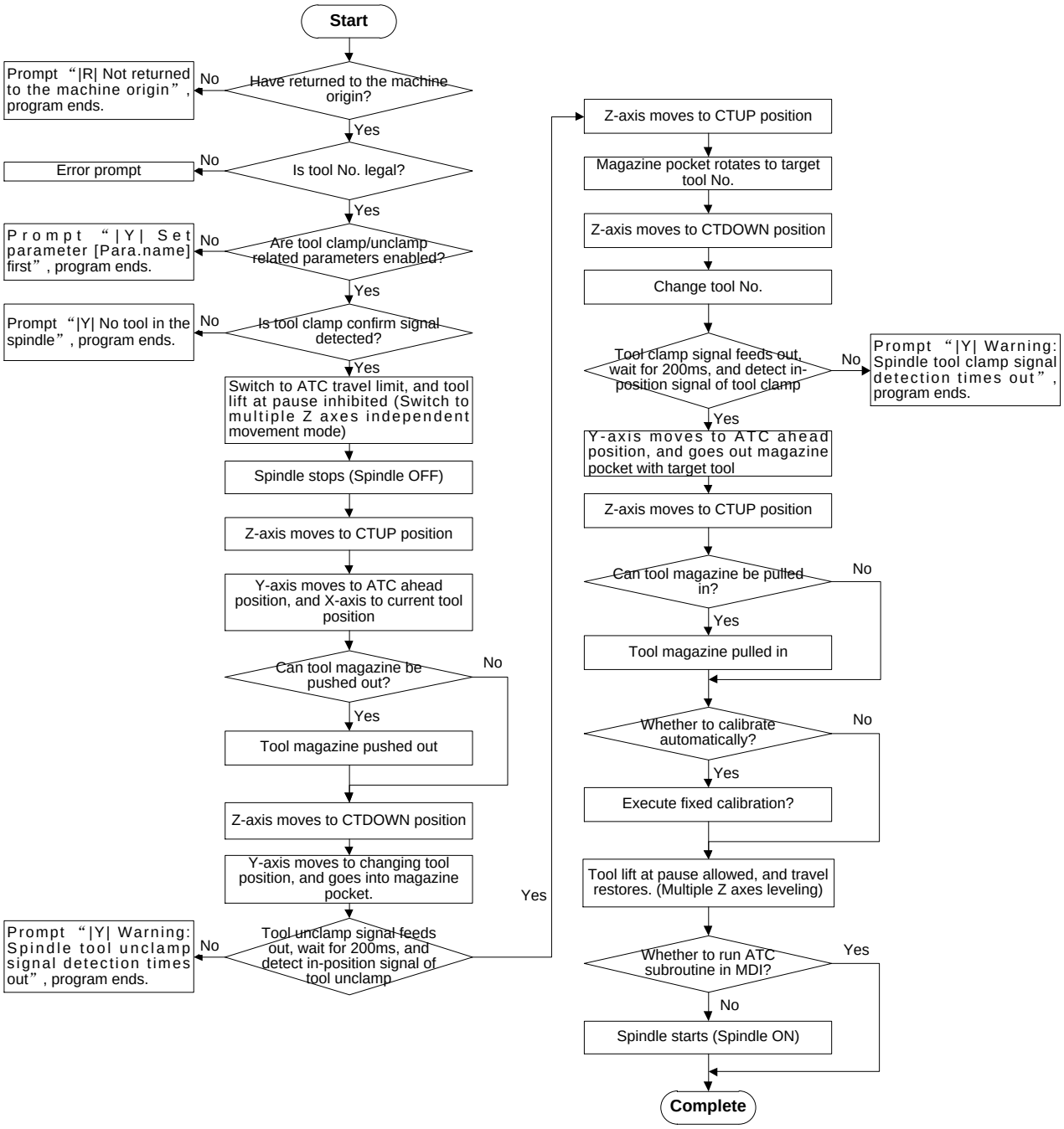


Fig. 3-102 Process of Auto Tool Change for A Servo Tool Magazine

CAUTION

When a servo tool magazine is activated,

1) A-axis stays motionless during process of returning to the fixed point or work zero;
2) When making all axes returning to the machine origin, A-axis is excluded;
3) Homing or returning status of A-axis will not be checked before machining.

3.20.4 Prompt for Tool Change

Prompts may pop up during the process of tool change, please refer to table below for details and countermeasures.

Table 3-4 Tool Change Prompts and Countermeasures

Prompt Type	Content	Countermeasure
|R| Prompt in red background	|R| Not returned to the machine origin	Return all axes to the machine origin.
|Y| Prompt in yellow background	|Y| Current tool No. is illegal	Set correct tool No.
	|Y| Target tool No. is illegal	Check following two aspects: 1) Is parameter setting for magazine capacity correct? 2) Is there any error in T command?
	|Y| Current tool No. of Z1 and Z2 are different	This prompt may appear in multi-Z configuration. Ensure correctness of tool No. first and then set same value for Z1 and Z2.
	|Y| Please set parameter[Name] first	Correctly set the parameter.
	|Y| No tool in the spindle, cannot change tool	Check following three aspects: 1) Tool has been well placed in the spindle; 2) Tool clamp signal port works well; 3) Wiring of tool clamp signal port is correct.
	|Y| In-position signal of dust cover not detected or it fell, please check!	Check in-position signal port of dust cover and its wiring.
	|Y| Warning: in-position detection of	Check in-position signal port of tool magazine pushed-out and

Prompt Type	Content	Countermeasure
	magazine pushed-out time out	its wiring.
	|Y| Warning: spindle tool unclamp signal detection time out	Check tool unclamp signal port and its wiring.
	|Y| Warning: spindle tool clamp signal detection time out	Check tool clamp signal port and its wiring.
	|Y| Warning: in-position detection of magazine pulled-in time out	Check the port and its wiring.
|M| Prompt without background color	|M| Tool No. of target tool and current tool are the same	Check following two aspects: 1) Is the tool correct? 2) Is T command repeated?

3.20.5 Parameter Specifications

Here are parameters related to auto tool change.

Class	Parameter		Details	Setting Range
Common Parameters	N66005	Upper Position	Z-axis machine coordinate when a tool moves to tool magazine for tool change, or CTUP position	-99999~99999
	N66006	Lower Position	Z-axis machine coordinate of tool change position when a tool moves downwards from Upper Position, or CTDOWN position	-99999~99999
	N66017~ N66018	Deceleration Position X/Y	Machine coordinate value of spindle position before tool change	-99999~100000
	N66028	Feedrate in Tool Changing	Movement speed of spindle during tool change	0~100000
	N66029	Feedrate in Moving from Upper Position to Lower Position	The moving speed of the spindle from the upper position to the lower position in Z direction during tool changing.	0~60000
	N66030	Automatic Tool Measure	Whether to execute auto tool measurement after tool change	NO: Not execute YES: To execute
	N66031	Tool Magazine Type	0: Null; 1: Disk Tool Magazine 2: Linear Tool Magazine	0; 1; 2; 3

Class	Parameter		Details	Setting Range
			3: Servo Tool Magazine (only available for A-type of 4 axes configuration)	
	N66032	Tool Magazine Capacity	The capacity of tool magazine	1~255
	N66033	Check Change ToolNo	Whether to check tool number in tool change is proper or not	YES: The tool number must be within (0, 256) in tool change command. NO: The range of the tool number is not limited, and the tool number remains the same.
	N66045	Tool Unclamp In-position Signal Port	PLC address of spindle in-position signal port when unclamping tool.	NA
	N66046	Tool Clamp In-position Signal	PLC address of spindle in-position signal port when clamping tool.	NA
	N66047	External Tool Control Signal Port	PLC address of the external signal port for controlling over tool clamp/unclamp.	NA
	N66048	Output Port of Tool Unclamp/Clamp	PLC address of output port for spindle unclamping/clamping tool.	NA
	N66049	Output Port of Mag. Out	PLC address of output port for tool magazine ejecting out.	-
	N67000~ N67002	Change Tool Workbench Range Lower Limit X/Y/Z	Machine coordinate value of worktable stroke lower limit of X/Y/Z during tool change	/
	N67010~ N67012	Change Tool Workbench Range Upper Limit X/Y/Z	Machine coordinate value of worktable stroke upper limit of X/Y/Z during tool change	/
	This group of parameters sets the worktable stroke range for tool change to avoid tool damage caused by over travel during tool change.			
Linear Magazine	Tool Position (X)	The machine coordinates of T1.		-99999~99999
	Tool Position (Y)			

Class	Parameter		Details	Setting Range
Circular Magazine	N66007~ N66008	Spindle Position in Tool Change	Position for the spindle to clamp/unclamp tool in tool change.	-99999~99999
	N66036	Tool Count Port	PLC address of the tool counting port.	NA
	N66037	Tool Mag. Back to Origin Port	PLC address of disk-type magazine back to magazine origin port.	NA
	N66038	Tool Mag. CW Port	PLC address of disk-type magazine CW rotation port.	NA
	N66039	Tool Mag. CCW Port	PLC address of disk-type magazine CCW rotation port.	NA
	N66040	Tool Count CW Delay	OFF delay when magazine rotates to the last pocket in CW direction during tool counting.	0~5000
	N66041	Tool Count CCW Delay	OFF delay when magazine rotates to the last pocket in CCW direction during tool counting.	0~5000
	N66042	Mag. CW to Origin Delay	Delayed time of the port when magazine turns to origin in clockwise direction.	0~5000
	N66043	Mag. CCW to Origin Delay	Delayed time of the port when magazine turns to origin in counter clockwise direction.	0~5000
Servo Magazine	N66050	Servo Magazine Rotate Speed	It refers to the rotational speed of servo tool magazine in tool change or magazine CW/CCW rotation.	0~100000
Comment	If parameter N66031 is set to “NULL”, tool change action will not be execute when T command is encountered, and only tool No. will be modified. Setting value of parameter N66032 should be consistence with actual magazine capacity of the machine. In case of linear tool magazine, each tool has its own position. In case of disk-type tool magazine, there may be deviation between the position where current tool stops in tool counting and the position of tool clamping. In order to minimize this kind of error, parameters N66040~66043 can be enabled.			

CAUTION

Since the machine structures of tool magzines vary, the above tool change flow charts apply only for general situations. Please contact with the manufacturer for part adjustment if the actual operation is diiferent fron the operation described above.

4 Introduction to Multi-Z Axes Software

This chapter focuses on introduction to operations of multi-Z axes software, especially to the difference compared with integral software.

4.1 Configuration Selection

Used together with multi-Z axes software, NK300CX can be a good solution to multiple Z axes motion control.

At present, “Union” configuration (also known as linkage configuration) and “Turn” configuration (also called alternative configuration) are supported. Similar to configuration switchover method in integral software, you can turn to 「System」 interface under [System] functional area to open a dialog box named “Config Manager”, where you can select configuration. Note that the software needs to be restarted to activate target configuration.

4.1.1 “Union” Configuration

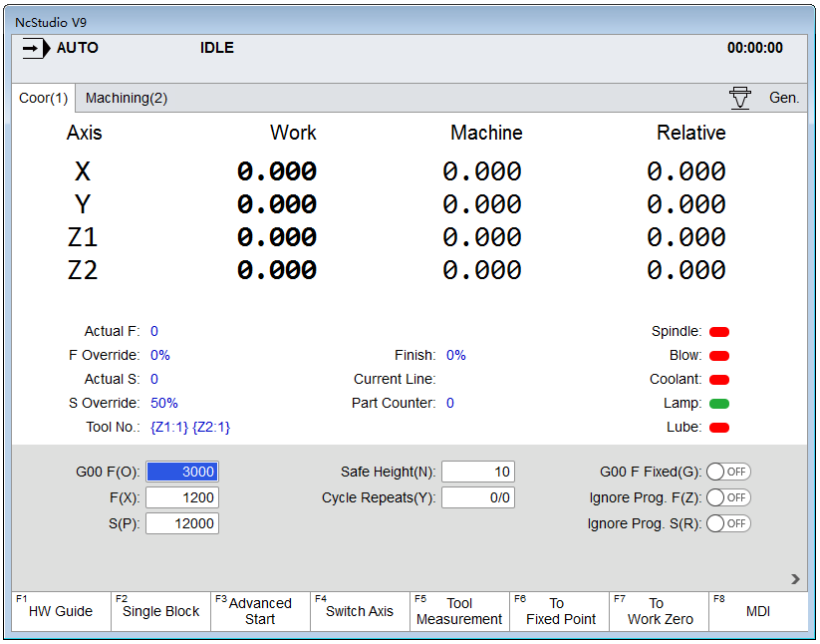


Fig. 4-1 「Coor」 interface under linkage configuration

With linkage configuration activated, as shown on above figure, both Z1-axis and Z2-axis are active.

◆ **Axis Switchover**

As shown in Fig. 4-1, pressing F4 “Switch Axis” button opens a subdivision manipulation button bar. F1~F3 correspond to “Select Z1”, “Select Z2” and “Select Z1Z2” respectively.

Note that if any type of tool magazine has been used, the system will detect in-position signal of tool clamp port of to-be-activated Z-axis, and prompt shown in yellow background will appear once the signal is not detected.

When switching Z-axis, Z1/Z2-axis will move to a safe position “Positive travel limit-1” first before activation of target axis. There are three situations in this process, as follows.

1) When Z1-axis is the target axis, the spindle stops first, and if Z2-axis has returned to the machine origin, Z2-axis moves to the position defined by parameter setting; if Z2-axis hasn't returned to the machine origin, Z2-axis stands still. At last, Z1-axis will be activated, and Z2-axis remains inactive.

2) When Z2-axis is the target axis, the spindle stops first, and if Z1-axis has returned to the machine origin, Z1-axis moves to the position defined by parameter setting; if Z1-axis hasn't returned to the machine origin, Z1-axis stands still. At last, Z2-axis will be activated and Z1-axis remains inactive.

3) When Z1Z2 axes need to be activated together, the spindle stops first, and if both Z axes have returned to the machine origin and parameter N75400 "Auto Leveling Z Axes" is set to "Yes", automatic leveling will be executed.

Auto leveling Z axes, as name implies, Z1Z2 axes will be adjusted to be same in terms of workpiece coordinates in Z direction. When automatic leveling is to be executed, final position of Z-axis is affected by the positive travel limit.

1) If the final position is lower than positive travel limit, Z1/Z2-axis will move to the position of larger coordinate with priority.

2) If the final position exceeds positive travel limit, Z1/Z2-axis will move the limit position.

◆ **Multiple Z Axes Automatic Leveling**

If both Z1Z2 axes are to be activated, the system will adopt prior strategy of moving up to make their Z coordinates the same. That is, leveling Z1Z2 axes automatically.

As shown in Fig. 4-1, press to turn to next manipulation button bar, as shown below. Press F5 "Leveling" to enable auto leveling function get the same workpiece coordinate in Z direction of both Z1Z2 axes.

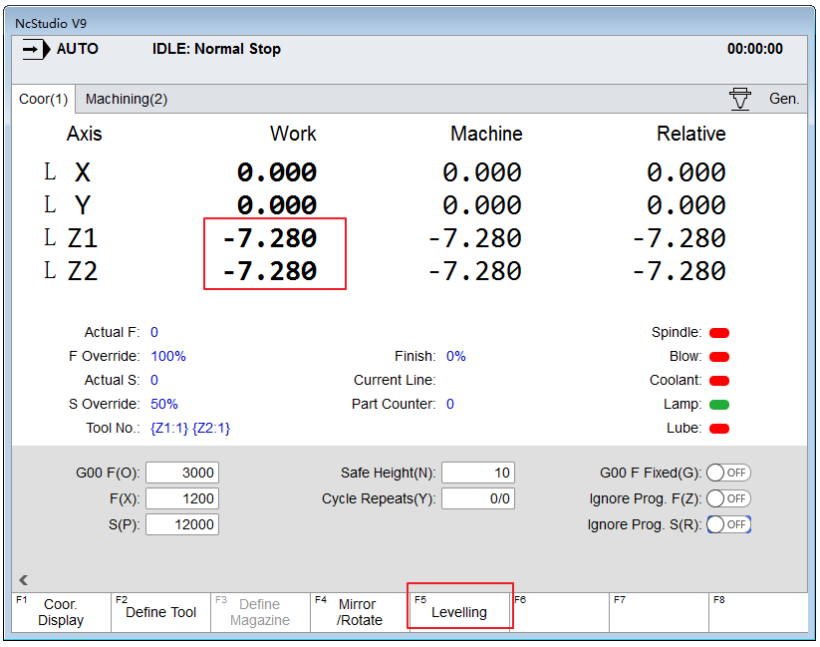


Fig. 4-2 Auto leveling Z axes

4.1.2 “Turn” Configuration

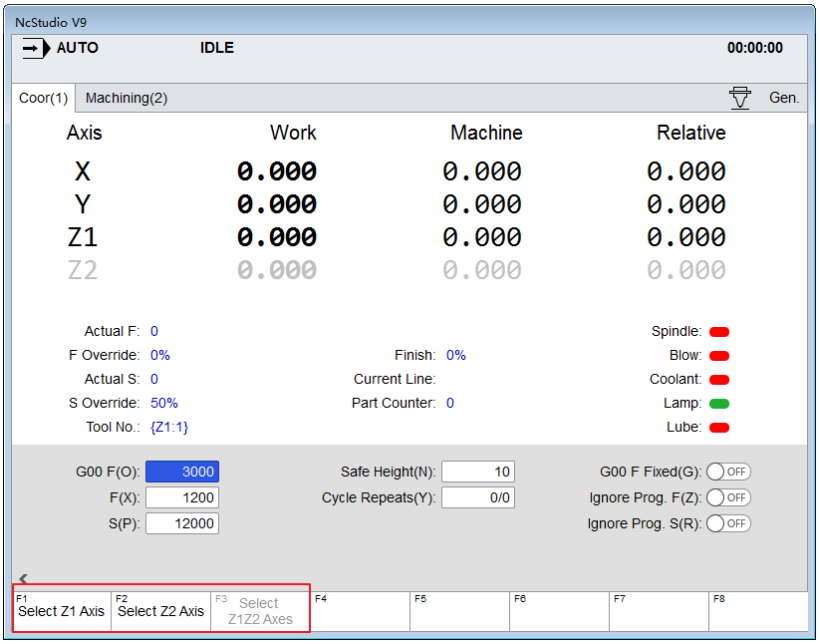


Fig. 4-3 「Coor」 interface under alternative configuration

With alternative configuration active, “Select Z1Z2” is unavailable. To put it in other words, there will be only one active Z-axis in any moment.

As shown in Fig. 4-3, press F1 or F2 to activate the target Z-axis. Current Z-axis will move to the position defined by parameter “Z1/Z2 position at axis switch” before activation of the target Z-axis. For instance, current Z-axis being Z1, and Z2-axis to be selected, Z1-axis will move to the position of “Z1 position at axis switch”, and Z2-axis will be activated, namely, switching Z1-axis to Z2-axis.

With alternative configuration active, the machine will do following actions before T command execution, taking changing T1 to T2 as an example.

1) If any type of tool magazine is used, tool clamping signal of Z1 and Z2 axes as well as related parameter settings will be detected firstly. The system will give yellow prompt if detection is negative and the program execution will be terminated.
2) Z1 and Z2 axes move to the position set by the parameters.
3) Z1-axis is deactivated and Z2-axis is to be activated.

4.2 Tool Measurement

Similar with that in integral software, tool measurement in multi-Z axes software is divided into mobile calibration, fixed calibration and first time/after changed calibration as well. Main difference lies in calibration process of an addition Z-axis. For this reason, please refer to section 4.2 for detailed information of calibration process. Here are corresponding calibration interfaces in multi-Z axes software, for reference only.

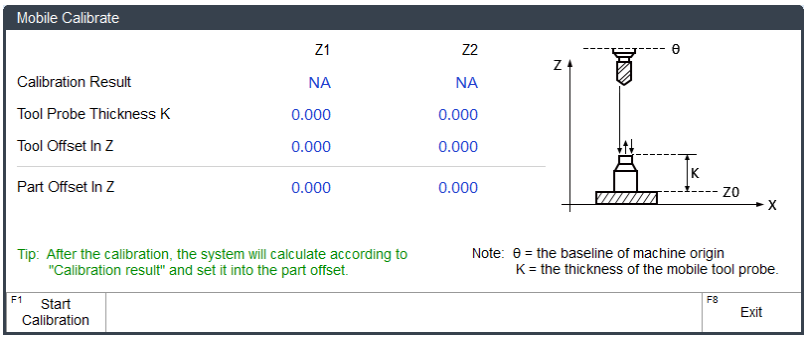


Fig. 4-4 Mobile calibration dialog box of multi-Z software

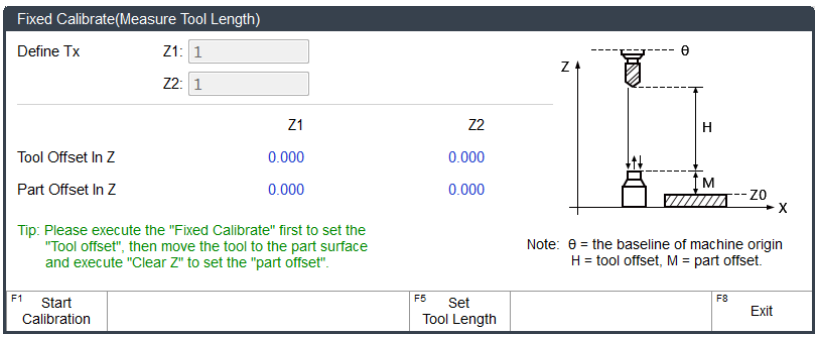


Fig. 4-5 Fixed calibration dialog box of multi-Z software

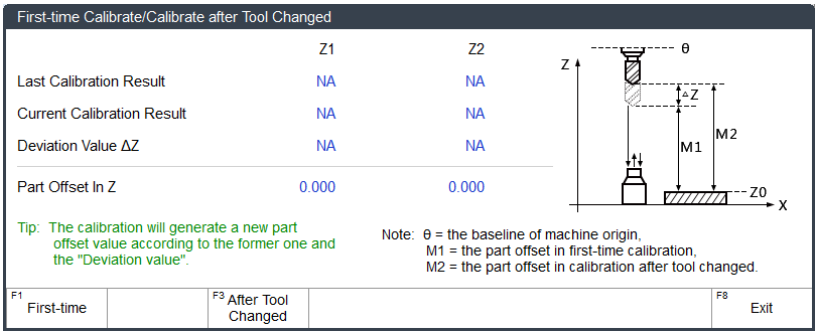


Fig. 4-6 First time/after TC calibration of multi-Z software

4.3 Related Parameters

On basis of parameters of integral software, there are some parameters exclusively owned by multi-Z axes software. Refer to following table for details.

Parameter		Meaning	Range
N41010	Auto Draining	Whether to drain automatically.	1: Yes; 0: No
N41011	Auto Draining Interval	The interval between twice drainage.	3.6~300000 sec
N41012	Auto Draining Duration	The duration of each automatic drainage.	1-100
N62090	G09 tolerance	Size of G09 exact stop window.	0~99mm
N64244	Optimize	Whether to make performance	1: Yes; 0: No

Parameter		Meaning	Range
	Performance	optimization or not.	
N66037	Automatic Tool Measurement	If yes, the machine automatically moves to the fixed tool presetter and conduct tool calibration when T command is being executed.	1: Yes; 0: No
N73002	Z-axis Lifting Mode on Pause	Z-axis lifting mode on pause. 0. Lift to the position set by parameter; 1. Lift to the workpiece coordinate set by parameter; 2. Lift to the machine coordinate set by parameter.	0; 1; 2
N75002	Z1 presetter input port address	The PLC address of the input port of tool presetter signal. For alternative calibration of multiple Z axes, only Z1 signal will be used.	-
N75003	Z2 presetter input port Addr		
N75024	ToolMea Overtravel Port Addr	The PLC address of input port on I/O terminal board, where the system gets overtravel signal from the presetter.	-
N75025	ToolMea Overtravel Alarm	Alarm will occur when overtravel in tool calibration.	1: Yes; 0: No
N75026	Tool calibration type	0: several tool probes available, multiple Z axes conduct tool measurement simultaneously. 1: only one tool probe available, multiple Z axes conduct tool measurement alternately.	0; 1
N79401	Z1 Pos when change spindle	The machine coordinates of Z1 while switching spindle.	-100~0mm
N79402	Z2 Pos when change spindle	The machine coordinates of Z1 while switching spindle.	-100~0mm
N79403	Switch to Z1	Whether to switch to Z1 when task ends. Only used in alternative configuration.	1: Yes; 0: No
N79404	Z1Z2 Spacing-Offset X	Distance between Z1 and Z2 axes in X direction.	-9999~9999mm
N79405	Z1Z2 Spacing-Offset Y	Distance between Z1 and Z2 axes in Y direction.	-9999~9999mm

5 Maintenance

5.1 Operation System Maintenance

When you receive product NK300CX, it is ready for use with all systems installed and setup. If any failure occurs, you can restore the software to leaving-factory state.

5.1.1 Preparation

1) An USB flash disk (above 1G);
2) The backup and restore toolkit

5.1.2 Creating OS Startup Disk

- **Creating an USB Startup Disk**

You can create an USB startup disk which will help the system access DOS interface, backup and restore the system SSD with DOS tools in the USB disk.

Steps to create USB startup disk are as follows:

1) Insert an USB into the PC, and double click file “HPUSBFW.EXE" in file folder “hpUpgsh” on the desktop of PC. An interface shown as below will appear.

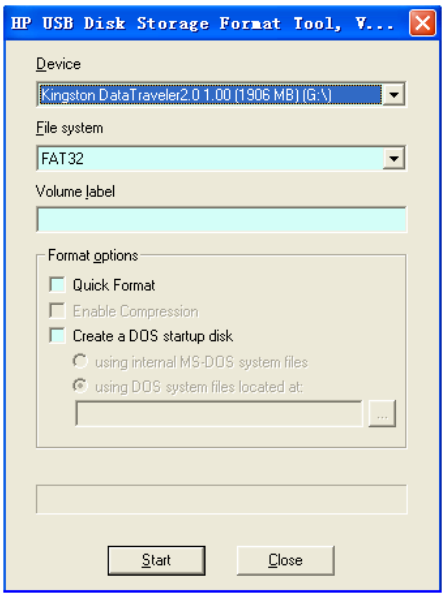


Fig. 5-1 USB Format Tool

- Under “Device”, select the USB flash disk which needs to be formatted;
- The default for “File system” is FAT32;
- Select items “Quick Format” and “Create a DOS startup disk;
- Below the item “using DOS system files located…”, specify and locate the path “Desktop\hpUpgsh\boot”;
- Click “Start” to initiate formatting. After two successive confirmations, creation of USB startup disk is successfully completed.

2) On the desktop of PC, double click file “USB backup and restore tool kit”, and interface shown as Figure 5-1 will appear.

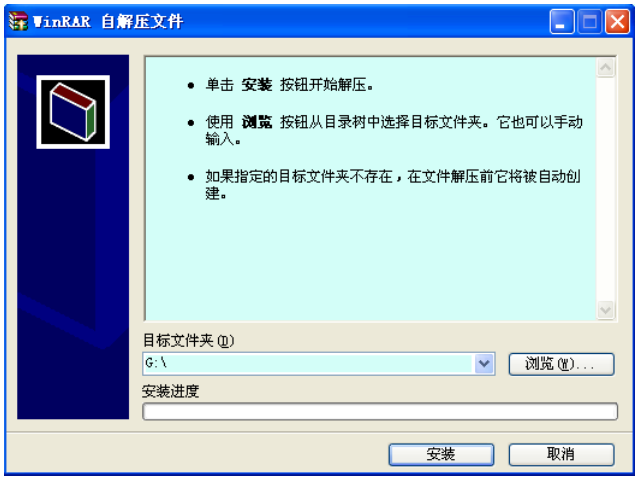


Figure 5-1 USB Restore Tool Kit

3) Locate the USB startup disk which has been created successfully in the target file box, and then click “Install”. After installation, all files contained in “USB backup and restore tool kit” will be unzipped into this USB.
4) Conduct anti-virus check on the USB to secure it is safe from viruses.

5.1.3 OS Restoration

Operating system restoration is the mirror image installation of the system SSD. An USB setup disk and a system SSD will be needed.

1) Insert the USB flash disk to the USB slot.
2) Restart the system, and press [Delete] key to enter BIOS interface. Accessing “Boot→Hard Drive BBS Priorities→Boot Option #1”, and set USB setup disk as “Boot Option #1”.
3) After setting start orders, press F4 to save the setting and restart. After normal startup, interface shown as below will appear.

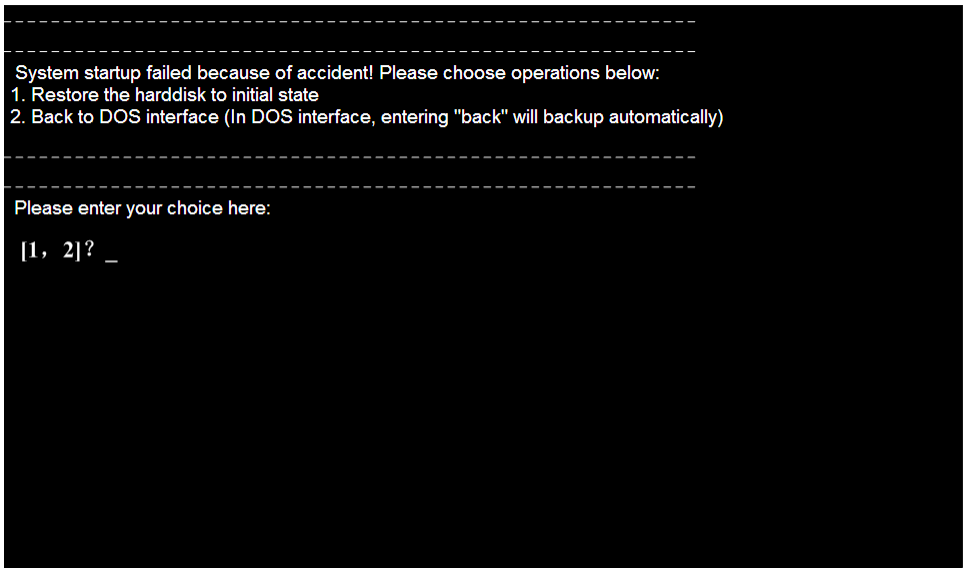


Fig. 5-2 DOS Startup Interface

- Enter “1”, and an interface shown as Fig. 5-3 DOS Confirmation Interface will appear.

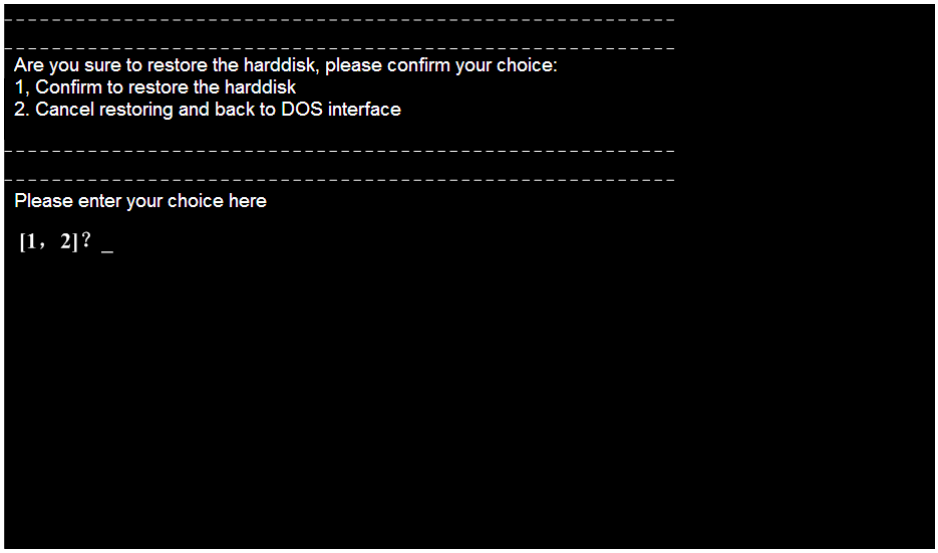


Fig. 5-3 DOS Confirmation Interface

- Enter “1” again, and the system will execute Ghost restoration. Pull out the USB disk the moment the system restarts. System installation is completed.

5.1.4 Backup OS to USB disk

Steps to back up operation system to USB disk are as follows.

1) Insert the USB flash disk to the USB slot.
2) Restart the system, and press [Delete] key to enter BIOS interface. Accessing “Boot→Hard Drive BBS Priorities→Boot Option #1”, and set USB setup disk as “Boot Option #1”.
3) After setting start orders, press F4 to save the setting and restart. After normal startup, interface shown as below will appear.

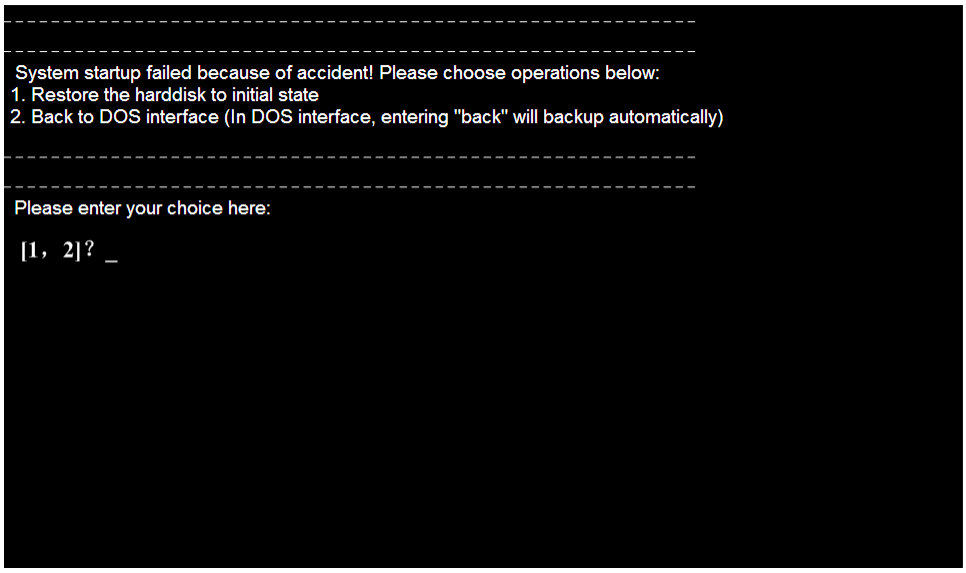


Fig. 5-4 DOS Startup Interface

4) Enter “2 → back → 2”, and the system will conduct Ghost backup process. Pull out the USB disk and turn off the power when the process finishes. System backup is completed.

5.1.5 Backup and Restoration of the Integrated System

You can use “Windows Ghost” Restoration function to restore the system when problems occur. Backup of operating system of NK300CX has been completed before leaving factory. In addition, the software has also been installed in CNC system, but its backup is not done in CNC system. We suggest that you immediately back up the BIOS system and the software again the first time the machine is power on or after debugging is completed.

- **System Re-Backup**

Steps to back up the system again are as follows.

1) Power on the machine, access the operating system choice interface, and select “Windows Ghost”, as shown in Fig. 5-5.

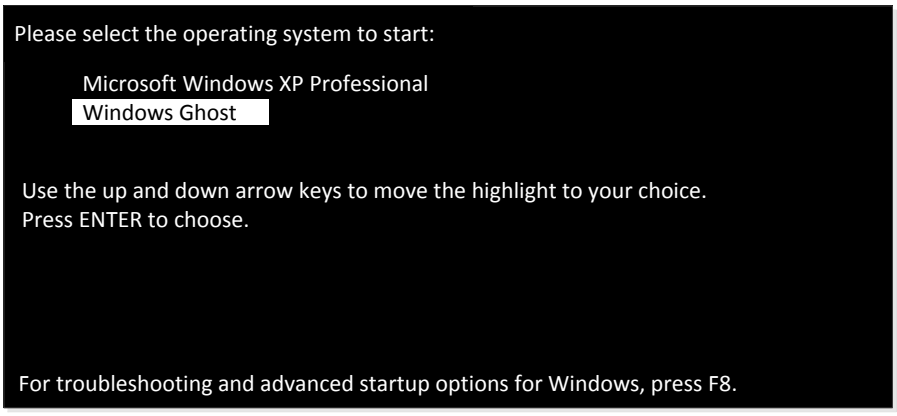


Fig. 5-5 Select “Windows Ghost” operating system

2) It jumps to restoration confirmation interface, as shown below.

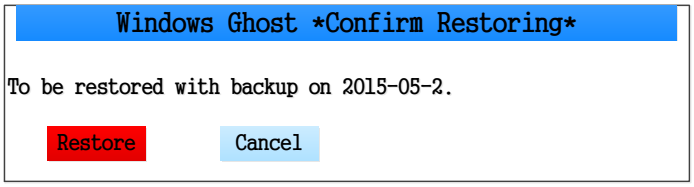


Fig. 5-6 Restoration confirmation

3) Press [Cancel] to access “Windows Ghost” interface, as shown in Fig. 5-7. Press “2” to select re-backup option.

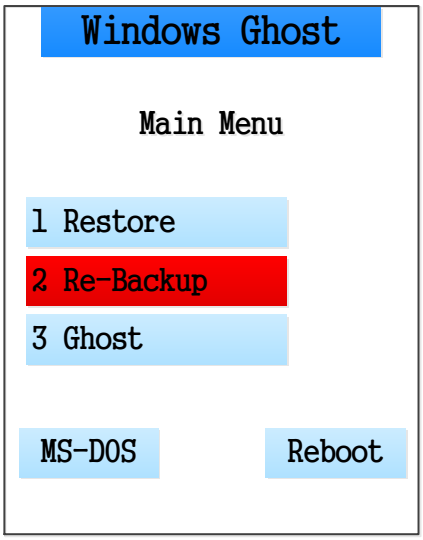


Fig. 5-7 “Window Ghost” interface

4) A re-backup confirmation dialog box will appear, as shown in Fig. 5-8.

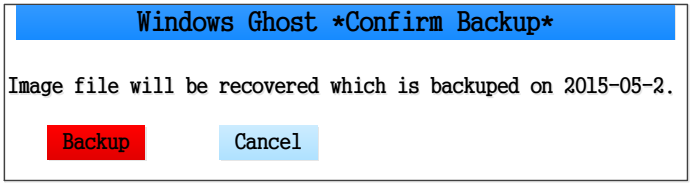


Fig. 5-8 Re-backup confirmation

5) Press [Backup] for confirmation. And the interface jumps to re-backup progressing interface. When re-backup finishes, the system will be restarted automatically.

- **System Restoration**

After re-backing up the system, if you need to use windows ghost restoration function, you can select “1 Restoration” in a dialog box shown as Fig. 5-6 or Fig. 5-7. And you can restore the system as directed.

Note that if you use windows ghost restoration function while you have not re-back up the system, the system will be re-started automatically. A dialog box titled with “FirstRun” will pop up noticing you that you should install the software, as shown in Fig. 5-9 and Fig. 5-10. You can select a disk to install the software as directed.

Fig. 5-9 FirstRun Notice 1

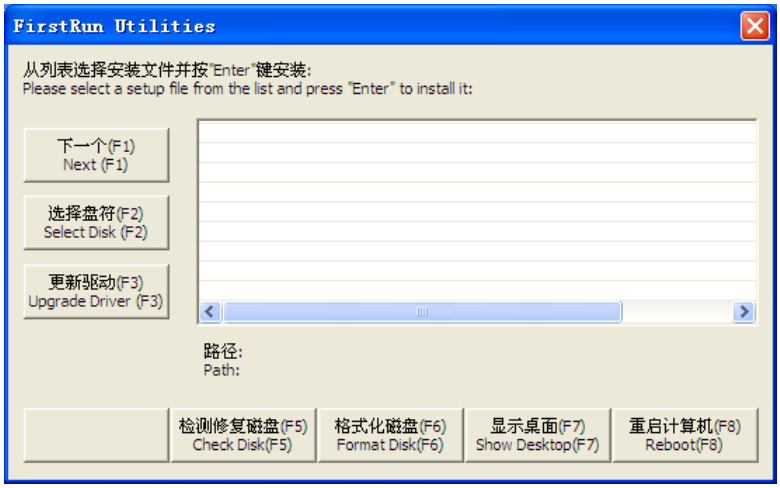


Fig. 5-10 FirstRun Notice 2

CAUTION

1) When exceptions occur during NcStudio system backup and restoration, you should consider the following causes at priority.
   - ✓ Is the guidance order of hardware in BIOS correct?
   - ✓ Is there any problem occurred during system backup?
   - ✓ Is the storage of USB disk enough during system backup?
   - ✓ The backup progress will exit automatically if image file exists in the USB disk.
   - ✓ During system backup, if there is mirror image file in USB disk, the process will exit.

To avoid the problems listed above, it is recommended that you conduct system disk security check and rerpair before proceeding with system backup and restoration. Otherwise, system performance may be influenced. So as data disk.

- ✓ Do not power off the PC during backup of NcStudio system. Otherwise, the system can be damaged.
- ✓ When the prompt about installing software in FirstRun dialog box appears, only .exe file is supported. Compressed files of format such as .zip, .rar are not supported. They must be unzipped for installation.

5.2 NcStudio System Maintenance

5.2.1 Package and Update

Software backup function is supported in NcStudio system. When software installation completes and parameters corresponding to a machine tool are set, you can pack up and back up the software with parameter settings as the original data. And the backup software can be directly installed on a machine tool of the same type. The function is realized in system maintenance. Steps to pack up and backup software are shown as follows.

When the system is in idle state, press  to enter the [System] functional area, and then press button F2 “System Maint.” to enter the system maintenance screen, as shown in Fig. 5-11.

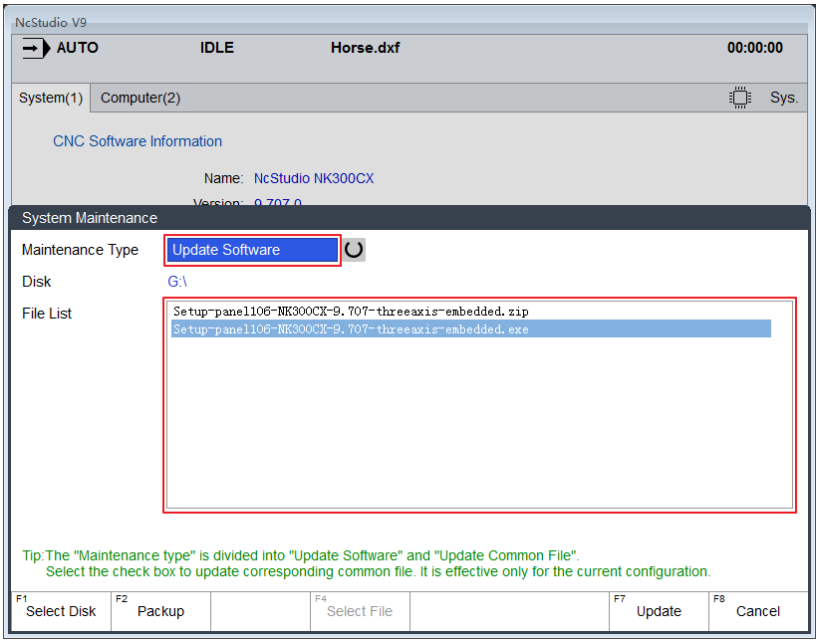


Fig. 5-11 System Maintenance- Upgrade Software

Maintenance type includes update software and update common software. The default is “Update software”. Select “Update software”, and the current parameters can be saved after the software is updated. Select “Update Common File” and then you can select Public, Plc, Amend, String files in “File List” for backup, which is only valid for the current configuration. Fig. 5-11 is the dialog box for “Update software”. In updating software,

- **Select Disk**

Press F1 “Select Disk”, and an input box for entering the drive letter of the removable disk with the update package will pop up. Select the disk where you will save the update package. Then all software update package in the disk will show in “File List”.

- **Packup**

Press F2 “Packup” to pack the current software automatically and save the packed software to the

selected disk.

- **Update**

Press key “ ↑ ” or “ ↓ ” to move the cursor and press space key to select the software , and then press F7 “Update” to start software installation. If the file cannot be opened normally, please refer to 5.2.2 for detailed installation information.

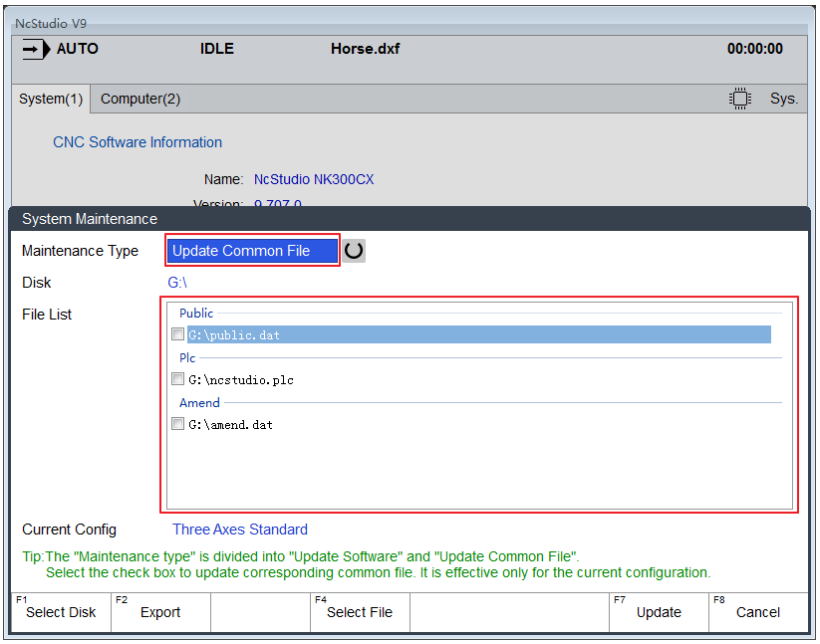


Fig. 5-12 Public File Upgrade

- **Select Disk**

Press F1 “Select Disk”, and an input box for entering the drive letter of the removable disk with the update package will pop up. Select the disk where you will save the update package. Then all files in the disk will show in “File List”, including Public, Plc, Amend, and String files.

- **Export**

Pressing F2, and the system will automatically export the public file under current configuration to USB disk. Prompt for successful exportation will pop up if export process completes.

- **Select File**

Press key “ ↑ ” or “ ↓ ” to move the cursor and press F4 to select the file. Check in the check box in front of the common files you need to update. You can update several files each time.

- **Update**

Press F7 to update. After confirmation, upgrading succeeds after restart the system.

If the public files to upgrade contains amend.dat file, before final upgrading, a prompt dialog box will pop up, as shown below. Choose “Yes” to confirm and continue updating and choose “No” to cancel it.

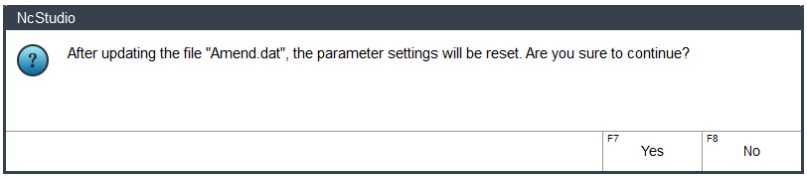


Fig. 5-13 Prompt Before amend.dat File Updating

CAUTION

Function “Update Common File” is only enabled for the current configuration.

5.2.2 Software Installation

Steps to install software are shown as follows:

1) Accessing the desktop. Press combination key “Ctrl + Alt + Delete” to enter the task manager interface ==> press “Alt + F” key to select “New Task” ==> in the new task dialog, input “explorer” and press “Enter” ==> press “Alt + Tab” to switch to NcStudio ==> press “Alt + F4” to close it and enter the desktop.

2) Insert the USB flash disk with the software NK300CX into the USB slot on the operation panel of NK300CX host. Access the desktop as directed in step 1. Find the software to install in [My Computer] and double click it to initiate installation. The first dialog box popping up is about language selection, as shown in Fig. 4 8. Switchover between languages while the software is running is supported in NcStudio. Choose the interface language you need.

Fig. 5-14 Language Selection Dialog Box

3) To avoid the interference of old version software to the current software installation, before formal installation begins, a prompting dialog box about saving previous parameter setting will pop up, as shown in Fig. 4 9. Choose “Yes” to save the parameters and delete the old version software before current software installation begins.

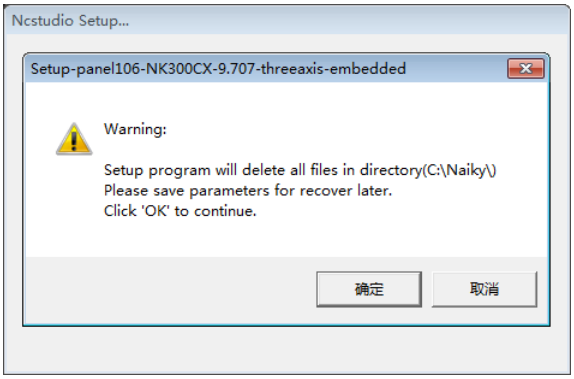


Fig. 5-15 Prompt for parameter saving

4) Click [确定] (means [Yes]) to continue. If software of other version has been installed before and its parameters have been modified, there will be a prompt dialog box for confirmation of parameter settings saving, as shown below. Note that if it is the first time to install the software, this prompt will be omitted. Jump to step 5) directly to go ahead.

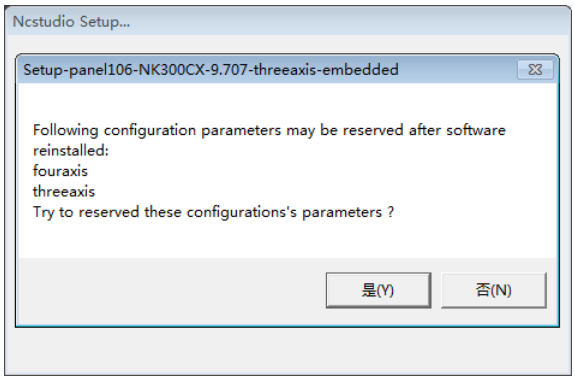


Fig. 5-16 Prompt for parameter settings saving

5) Click [是] (means [Yes]). The system will be installed under directory C:\Naiky. Installation progress is shown in progress bar, as shown in below.

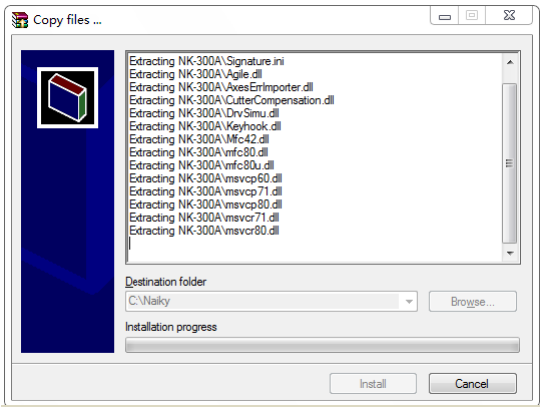


Fig. 5-17 Installation progressing

6) Software installation is completed.

CAUTION

The above installation introduction is for situation where the software has been damaged and cannot work normally. If the software can be launched normally, please refer to 4.2.2 to upgrade software instead of newly installing one.

5.3 Warning Information

Type	Warning Content	Causes	Solution
Warning	"Simulation results showed that the motion of the program exceeded machining bounds."	The scope of machining file exceeds upper and lower limits of worktable, decided by setting "N10020"& "N10030".	Modify the value of "N10020" and "N10030" to expand stroke range of worktable (see section 3.3.3).
	"Returning machine home was not finished!"	The system has not returned to machine origin yet. If "N74001" is set to "YES", returning to machine origin before machining is required.	Back to machine origin before using this function.
	"The result error of returning machine home for X/Y/Z-axis was out of range"	An error in the precision of home switch	Detect the precision of home switch.
		An error in the precision of encoder origin	Detect if the system encoder zero signal is correct.
	"Unable to perform the action under the current mode"	An illegal operation is executed in machining, such as changing the setting of a parameter.	Stop machining, and execute the operation under idle state.
	"Unable to perform the action under simulation mode"	An illegal operation is executed in simulation mode, such as changing the setting of a parameter or pressing a shortcut key	Quit simulation mode, and execute the operation under idle state.

Type	Warning Content	Causes	Solution
Limit alarm	"Limit of X+/X-/Y+/Y-/Z+/Z-"	The polarity of port Positive/Negative Limit of X-/Y-/Z-axis is not right.	Enter [I/O Port] screen under [Diagnosis], and modify the port polarity (refer to section 3.2).
		X-/Y-/Z-axis runs into the limit switch directly in movement.	Manually move X-/Y-/Z-axis away from the limit switch.
		There is an error in the limit switch itself.	Check if the limit switch works normally.
Back to REF. point alarm	"The distance of coarse/fine positioning switch for X/Y/Z-axis was too close"	The actual installation distance between coarse and fine positioning switches is smaller than the setting value of parameter "N74110".	Re-adjust the actual position of home switch and encoder zero to make the space within the range (0 + "N74110", screw pitch – "N74110") (see section 3.2).
Servo alarm	"Servo alarm of X/Y/Z-axis"	The polarity of port Axis X/Y/Z Servo Alarm is wrong.	Enter [I/O Port] screen under [Diagnosis], and modify the port polarity (refer to section 3.2).
		There is an error in the X/Y/Z-axis servo driver itself.	Check if the X/Y/Z-axis servo driver works normally.
E-stop alarm	"ESTOP button pressed"	The polarity of port Emergency Stop is wrong.	Enter [I/O Port] screen under [Diagnosis], and modify the port polarity (refer to section 3.2).
		The E-stop button is pressed down.	Turn the E-stop button clockwise to make it pop-up.
Oil level alarm	"lube level low alarm"	The polarity of port Lubrication Position Test Alarm is wrong.	Enter [I/O Port] screen under [Diagnosis], and modify the port polarity (see section 3.2).
		When the oil level line in the	Check if the oil mass is too

Type	Warning Content	Causes	Solution
		oil pump is below a certain value, a signal will be sent to the system to give an alarm.	small in the oil pump.
Spindle alarm	"Spindle alarm"	The polarity of port Spindle Alarm is wrong.	Enter [I/O Port] screen under [Diagnosis], and modify the port polarity (see section 3.2).
		There is an error in the inverter.	Check if the inverter works normally.
File error	"Machining program not loaded yet"	Start file machining with no file loaded in advance.	Load a machining file in advance.
Pulse feedback alarm	"Axis X/Y/Z Encoder Steady/Dynamic Error"; "Axis X/Y/Z Serious Following Error"	It is used to detect if the D-value between sent pulses and received pulses exceeds the setting value of the corresponding parameter.	Check if the servo system is stable or if the motor encoder is damaged.
Change tool over-travel alarm	Alarm for over-travel in tool change	Alarm signal occurs in tool change over-travel protection port	Check if the tool presetter works normally. During tool changing, Z-axis keeps moving downward for receiving no calibration signal, and triggers the over-travel protection port. Hardware faulty, which may result in continuous signal of the port.
Terminal board not connected	The terminal board is not well connected with the NK300CX system	Wiring is not well or hardware fault of Lambda controller.	Re-plug the connection wire and restart the software. Something wrong with the port polarity. Invert the polarity and restart the software. Analyze possible causes

Type	Warning Content	Causes	Solution
			according to the state of SYSTEM LED indicator. Change a new Lambda controller.
Panel not connected	Operational panel is not well connected	Something wrong with the port polarity. Wiring is not well. Operation panel fault.	Something wrong with the port polarity. Invert the polarity and restart the software. Re-plug the connection wire and restart the software. change a new operational panel.

CAUTION

Some alarms shown in table above are alarms added for the the machine structure of of a machine tool, and it is not discribed in general warning information. Please consult with the machine tool manufacturer if you have any questions.

5.4 Common Troubleshooting

5.4.1 What should you do if the spindle does not rotate?

1) Start the spindle, and check whether the spindle start indicator lamp on the controller is on.
2) If the lamp is on, measure whether the SPIN port, i.e. port Y00 in integral software, or port Y03/Y0 in multi-Z-axis embedded software, is conducted with a multimeter. If the port is conducted and works normally, check whether the parameter setting of the inverter is right, whether the spindle and the inverter have been damaged, or whether the wiring of the spindle and the inverter is correct.
3) If the lamp is off, close the host machine and power off the machine tool, and then re-plug the connection cable on the controller. If it still does not light up, please change the Lambda controller or the NK300CX host machine.

5.4.2 What should you do if an axis does not move?

1) Check whether the polarity of output port “Servo Enable of any axis”, in [Port(3)] screen under [Diagnosis] is correct. Normally it should be “NO”.

2) Check whether the parameters about the servo driver, including control mode, pulse input form, and electronic gear ratio, are set correctly. The control mode should be set as position control. The pulse input form should be pulse+direction.
3) Check whether the servo cable of this axis is well connected with the system host machine and the servo driver.
4) Check whether the motor is enabled.
5) Move the machine tool manually, and check whether the driver receives pulses. If it receives pulses and the machine tool has no output, check the transmission is loose. If it does not receive pulses, please change the host machine or the driver.

5.4.3 What should you do if servo motor brake in Z-axis does not work?

1) Check whether there is signal in input port “Brake”. If there is no signal, check whether servo driver is enabled, and whether the parameter about brake of servo driver is set correctly.
2) If there is signal, remove the cables connecting with output port brake, Y00-C00, start the system, power on the machine tool with system alarm signal removed, and measure whether the port is conducted with a multimeter.
3) Power off the machine tool, reconnect the two cables, and reconnect the 24V power in the former circuit. Power on and measure whether there is 24V voltage between the ends of the brake cable with a multimeter. If there is 24V voltage, the motor is damaged.
4) If the brake still does not work, please change the Lambda controller.

5.4.4 What should you do if homing is abnormal?

1. Limit alarm or servo driver alarm appears during homing, i.e. backing to machine origin.

1) Press button  to enter the [Diagnosis] functional area, press key “3” to open the [Port] interface. Make sure the polarity of input port “Reference Point of n-axis” is in accordance with the signal type of the port. “NO” represents “nomally open”, and “NC” represents “normally close”.
2) Move the machine tool to home switch position manually. Check whether the color of the dot in front of the “Reference Point of n axis” changes from red to green. If there is no color change, the software can’t receive the reference point signal. Check if there is any problem in the home switch or in the wiring of home switch. To check whether the system failure occurs, conduct the reference point signal with COM port on the controller with a conducting wire, and check whether the color of the dot before “Reference Point of × axis” changes.
3) Enter [Axis(2)] interface of [Parameter] functional area, and check whether parameter “Coarse Positioning Dir.”, “Fine Positioning Dir.” and “Back Off Distance” are set correctly. The direction of parameter “Fine Positioning Dir.” should be the same as that of parameter “Back Off Distance”, and opposite to the direction of “Coarse Positioning Dir.”.
4) Check whether the position of home switch is appropriate to avoid the following three situations: the distance between home switch and limit switch is too short; the home switch is installed

behind the limit switch; or the position of home switch is out of the mechanical stroke of a machine tool.

2. When backing to machine origin, the machine tool motions towards a certain direction at a relatively low speed (ten percent of the speed of coarse positioning) until limit is triggered.

   Press button  to enter the [Diagnosis] functional area, press key “3” to open the [Port] interface. Check whether the polarity of input port “Reference Point of n-axis” is correct]. When the home switch is triggered, i.e., when there is an input signal, the dot in front of the port number should be green. Otherwise, it is red.

3. A certain axis moves a very long distance at a rather low speed or keeps moving in the opposite direction after coarse positioning during backing to machine origin.

   The reason why the above problem occurs is that the system can’t detect zero signal of the encoder on the axis. The solutions are as below.

   1) Move the machine tool manually, and check whether there is any signal on input port “Encoder Zero of n-axis” in [Port] interface.
   2) Check whether the servo cable of this axis is well connected with the system host machine and servo driver.
   3) Check whether there is any problem in the driver, motor, encoder cable, servo cable, and the control system. e.g., you can exchange the servo cable and the servo driver separately with those of axes which return to machine origin normally.

5.4.5 What should you do if a machine tool moves upward after arriving at the position of tool presetter during tool presetting?

1) Press button  to enter the [Diagnosis] functional area, press key “3” to open the [Port] interface. Check whether the ploarity of input port “Cut”, i.e. “X24”, is in accordance with the signal type of the port.
2) Manually press down the tool presetter and check whether the polarity of port “Cut” in [Port] interface changes. If it the polarity does not change, the tool presetter must have been damaged.

5.4.6 What should you do if software failed because of automatic write number identification?

For NK300CX integrated CNC system, different configurations may be embedded in one software. Hardware of different configurations is the same, which can be told from write number of board card. Specifically, 00, GN and I5 are three matchable write number for NK300CX general software, with 00/GN representing for 4 or less axes configurations, while I5 representing 5 or less axes configurations. T4

and T5 are write number for denture machine of NK300CX series, with T4 for 4 axes configuration and T5 for 5 axes configuration separately. To conclude, matchable write number for NK300CX series software are 00, GN, I5, T4 and T5.

Write number of the board card will be automatically identified during software startup. If write number is mismatched, there will be installation error. You can refer to following contents for countermeasures for different failure situations.

5.4.6.1 Write file tag not exist

Prompt: Write file tag not found, software exit!

Solution: Install legal software.

If the software is illegal, prompt as above will pop up and software installation aborts. You can solve this problem by installing legal software.

5.4.6.2 Write type not matched

Prompt: Device type is not matchable

Solution: ①Register the software and write the board card again; ②Re-install the software.

If write number of board card is not the same with the write number in configuration file, above prompt will pop up, as shown in Fig. 5-18. There are two ways to solve this problem, register the software and write the board card again, or install software whose write type is matchable with write type of board card.

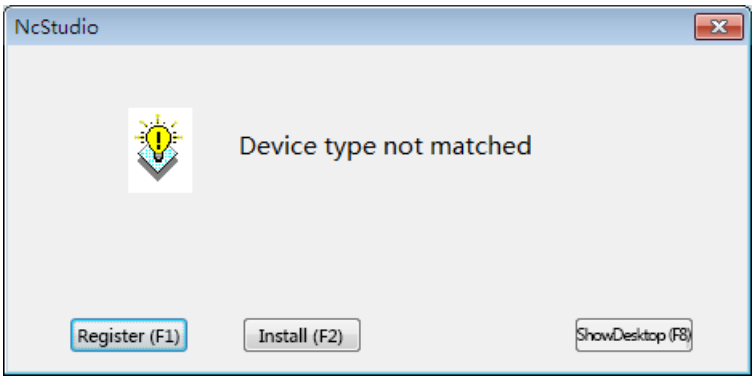


Fig. 5-18 Prompt for unmatchable device type

(1) Register the software again

As shown in Fig. 5-18, press F1 to open a dialog box titled “Registration”, see Fig. 5-19. You need to generate a registration coder with help of “NcStudio Generator” APP first, and type it into the box in. Press F7 to confirm and complete write to hardware. Note that software needs to be restarted to validate the new write number.

Please refer to section 3.18.4 for detailed instructions to NcStudio generator.

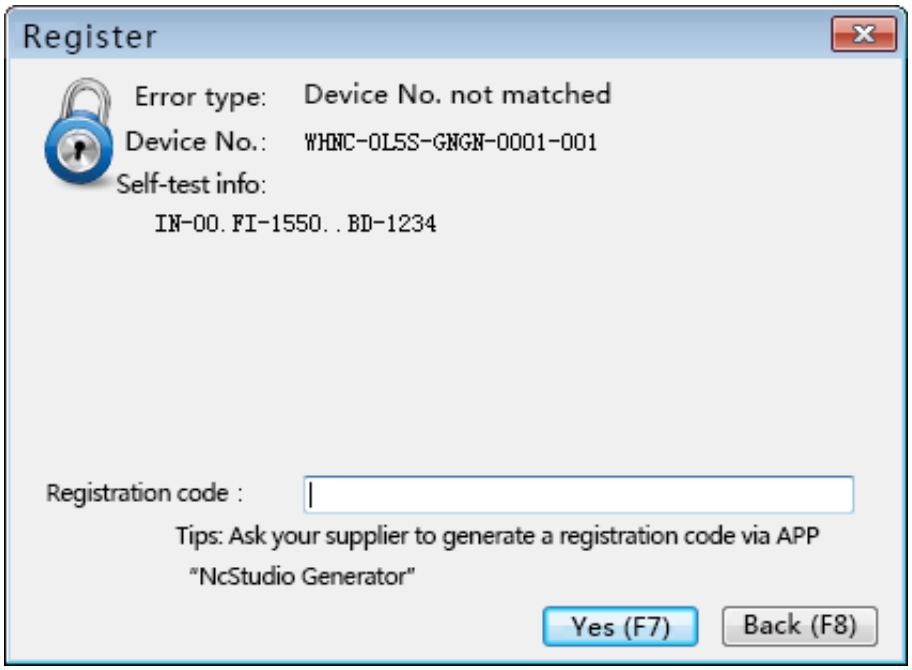


Fig. 5-19 "Register" dialog box

(2) Re-install software

As shown in Fig. 5-18, press F2 to open a dialog box titled "Install software", see Fig. 5-20. Select an installation package whose write number is matchable, and press F7 to install it.

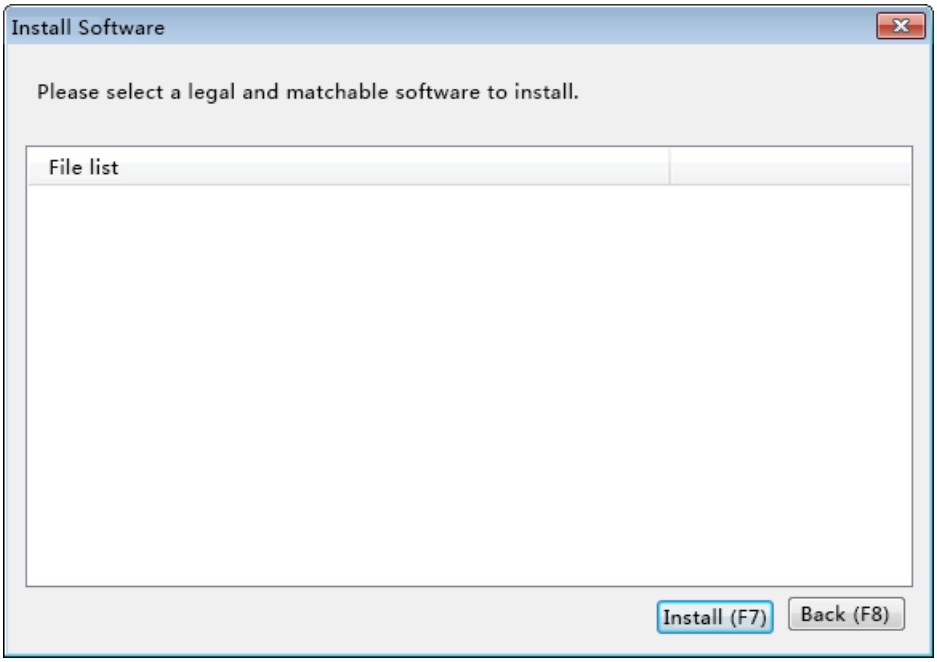


Fig. 5-20 "Install software" dialog box

(3) Show desktop

As shown in Fig. 5-18, press F8 to return to the desktop.

5.4.6.3 Active configuration not matched

Prompt: The board card does not support current configuration, please choose another configuration in flowing list.

Solution: Choose another proper configuration.

If active configuration of the software is found inconsistent with the configuration written by board card, prompt as above will pop out. In this case, all you need to do is to choose a proper configuration and restart the software.

See Fig. 5-21 for configuration list, where you can select a matchable one and press F7 to activate it. Press F8 to exit the software and show the desktop.

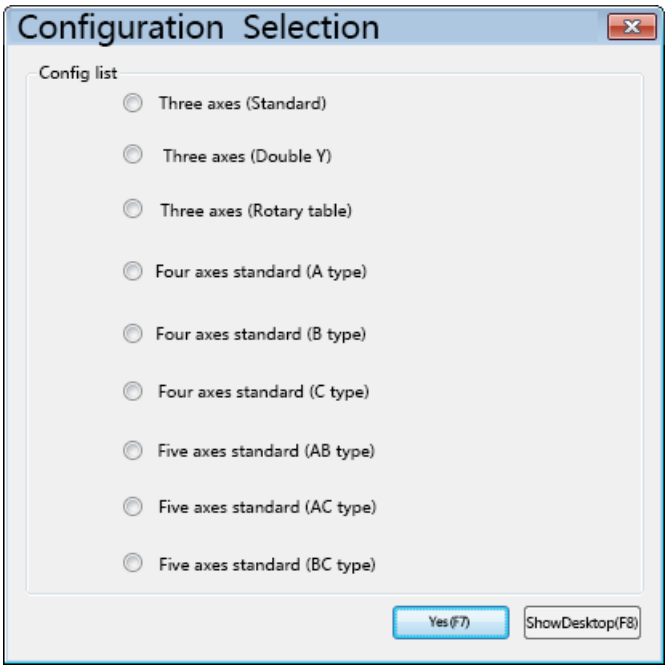


Fig. 5-21 Configuration list

5.4.6.4 Active axes number and configuration not matched

Prompt: The software is illegal, you can report by calling at 021-33587550.

Solution: Re-install a legal software.

If number of actual axes is larger than that supported by hardware writing, prompt as above will appear and the software exits. To solve this problem, please contact with the supplier and install legal software.

6 Driver

6.1 Driver Parameters

Parameters listed in this chapter can only make a machine work normally instead of ensuring the best machining results. Relevant parameters need adjusting according to the specific machine type.

6.1.1 Parameters Setting of WISE Servo Driver

Para. No.	Function	Value	Description
Pr528	LED initial status	6	Monitor if the number of sent and received pulses is correct by setting this parameter. In Weihong control system, the correct quantity of pulses sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pr008	Command pulse No. per motor circle	0	When it is set to "0", parameters Pr009 and Pr010 are valid.
Pr009	1st numerator of command pulse frequency division/ multiplication	Need calculation	Range: 0~2^{30} Typical value: pitch 5 mm, encoder resolution 10000, deceleration ratio 1:1, pulse equivalent 0.001 mm/p: Pr009＝10000 Pr010＝pitch 5mm/ pulse equivalent 0.001mm＝5000 Pr009/Pr010=10000/5000=2/1
Pr010	Denominator of command pulse frequency division/multiplication	Need calculation	
Pr011	Output pulse No. per motor circle	2500 (default)	Typical value: pulse equivalent 0.001mm/p, deceleration ratio 1:1, pitch 10mm/p, sets this parameter to 2500; pitch 5mm/p, sets this parameter 1250.
Pr100	1st position loop gain	480 (default)	Unit: 0.1/s. Set it according to the actual situation.
Pr101	1st velocity loop gain	270 (default)	Unit: 0.1Hz. Set it according to the actual situation.
Pr102	1st velocity loop integrated time constant	210 (default)	Unit: 0.1ms. Set it according to the actual situation.

When the value of Pr008 is not "0", it can be calculated in terms of the following formula:

$$\text{Command pulse No. per motor circle} = \frac{\text{Screw pitch}}{\text{Pulse equivalent} \times \text{Mechanical deceleration ratio}} = \frac{5\text{mm}}{0.001\text{mm}/p} = 5000$$

When screw pitch is 5mm and pulse equivalent 0.001, the value of Pr008 is "5000".

◆ **Attachment List: the relationship among parameters Pr008, Pr009 and Pr010.**

Pr008	Pr009	Pr010	Description
1~2^{20}	_ (no influence)	_ (no influence)	Command Pulse Input → Encoder Resolution / Setting Value of Pr008 → Position Command As shown above, the process is undergone in terms of the setting value of Pr008, not affected by the settings of Pr009 and Pr010.
0	0	1~2^{30}	Command Pulse Input → Encoder Resolution / Setting Value of Pr010 → Position Command When the values of Pr008 and Pr009 are both set to “0”, as shown above, the process is undergone in terms of the setting value of Pr010.
	1~2^{30}	1~2^{30}	Command Pulse Input → Setting Value of Pr009 / Setting Value of Pr010 → Position Command When the value of Pr008 is “0”, but the value of Pr009 is not “0”, as shown above, the process is undergone in terms of the setting values of Pr009 and Pr010.

6.1.2 Parameters Setting of YASKAWA Σ－Ⅱ Servo Driver

Para. No.	Function	Value	Description
Fn010	Set password (to prevent arbitrary modification to parameters)	0000	Set [0000]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] permitted; Set [0001]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] prohibited.
Un00C	Pulse counter of input command	LXXXX (Hexadecimal system)	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pn000	Direction selection Control mode selection	0010	Bit 0: Set 0, “CCW” is forward rotation (viewed from the load end of screw ball); Set 1, the rotation direction of the motor is reversed. Bit 1: Set 1, position control mode (calculate pulse instruction all the time).

Para. No.	Function	Value	Description
Pn200	Select pulse instruction mode	0005	Bit 0: Set 5, select the instruction input mode as “pulse + direction”, negative logic. Bit 3: Set 0, input differential signal into filter.
Pn50A	Selection function	8100	Bit 1: Set 0, Servo ON /S-ON, input from 40th pin; Set 7, Servo ON all the time. Bit 3: Set 8, positive rotation not used and signal input (P-OT) prohibited.
Pn50B	Selection function	6548	Bit 0: Set 8, reverse rotation not used and signal input (N-OT) prohibited.
Pn50F	Selection function	0300	Set it when servo motor with brakes. Bit 2: Set 3, brake interlock signal “/BK” is output from CN1-29, CN1-30 to control 24V relay for brake
Pn50E	Selection function	0211	Set it when servo motor with brakes To avoid of CN1-29 and CN1-30 being used for other function and leading to brake ineffective, “3” is not allowed to appear in the 4 digits.
Pn506	Servo off, time delay of brake when motor stops	Depended	Set it when motor with brakes Default setting is “0”, setting unit is 10ms.
Pn201	PG divider	Need Calculation	Range: 16 ~2^{14}. Set it according to actual PG divider ratio. Typical value: pulse equivalent 0.001mm/p, without reduction box, pitch 10mm, set this parameter to 2500; pitch 5mm, set it to 1250.
Pn202	Electronic gear ratio (numerator)	Need Calculation	Pn202 = pulse No. of each encoder circle × 4 × mechanical deceleration ratio. Pn203 = (screw pitch/ pulse equivalent). Typical value: pitch 5mm, encoder 17-bit, coaxial connection between motor and screw, pulse equivalent 0.001mm, Pn202＝16384; Pn203＝625. Pitch 5mm, encoder 17-bit, coaxial connection between motor and screw, pulse equivalent 0.0005mm, Pn202＝8192; Pn203＝625.
Pn203	Electronic gear ratio (denominator)	Need Calculation	

6.1.3 Parameter Setting of YASKAWA Σ-Ⅴ Servo Driver

Para. No.	Function	Value	Description
Fn010	Parameter input prohibition setting	0000	Set [0000]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] permitted; Set [0001]: modification to user parameters [PnXXX] and part of auxiliary function parameters [FnXXX] prohibited.
Pn000	Function selection basic switch 0	0010	Bit 0: Set 0, positive rotation at positive rotation command Bit 1: Set 1, position control mode (pulse sequence command)
Pn200	Format selection switch of position control command	0005	Bit 0: Set 5, select the instruction mode as “pulse + direction”, negative logic.
Pn50A	Input signal selection 1	8100	Bit 1: Set 0, Servo ON /S-ON, input from the 40th pin; Set 7, Servo ON all the time. Bit 3: Set 8, positive rotation not used and signal input (P-OT) prohibited.
Pn50B	Input signal selection 2	6548	Bit 0: Set 8, negative rotation not used and signal input (N-OT) prohibited.
Pn50F	Output signal selection 2	0300	Set it when servo motor with brakes. Bit 2: Set 3, brake interlock signal “/BK” is output from CN1-29, CN1-30 to control 24V relay used for brake
Pn50E	Output signal selection 1	0211	Set it when servo motor with brakes To avoid of CN1-29 and CN1-30 being used for other function and leading to brake ineffective, 3 is not allowed to appear in the 4 digits.
Pn506	Brake instruction-servo OFF and time delay	Depended	Set it when motor with brakes Default setting is “0”, setting unit is ms.
Pn20E	Electronic gear ratio (numerator)	Need Calculation	$\frac{Pn20E}{Pn210}=\frac{Encoder\ resolution \times Pulse\ equivalent \times Deceleration\ ratio}{Screw\ pitch}$ For example, screw pitch 5mm, 20-bit encoder, coupling direct drag, pulse equivalent 0.001mm,
Pn210	Electronic gear ratio (denominator)	Need Calculation	

Para. No.	Function	Value	Description
			$\frac{PN20E}{PN210}=\frac{2^{20}\times 0.001}{5}=\frac{1048576}{5000}=\frac{131072}{625}\approx\frac{210}{1}$ When screw pitch is 10mm, $\frac{PN20E}{PN210}=\frac{1048576}{10000}=\frac{65536}{625}\approx\frac{105}{1}$ For a rotary axis with 13-bit encoder and deceleration ratio as 60, $\frac{PN20E}{PN210}=\frac{2^{13}\times 0.001\times 60}{360}=\frac{8192}{6000}=\frac{512}{375}$

6.1.4 Parameter Setting of PANASONIC MINAS A4 Servo Driver

Para. No.	Function	Value	Description
Pr01	LED initial status	12	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pr02	Select control mode	0	0: position mode 1: velocity mode 2: torque mode
Pr40	Selection of command pulse input	1	1: input by differential exclusive circuit
Pr42	Select command pulse input mode	3	Set command pulse input mode: command pulse + command direction, negative logic
Pr44	Feedback pulse divider (numerator)	Need Calculation	Range: 1 ~ 32767. Set it according to actual PG divider ratio. Pulse equivalent 0.001mm/p, deceleration ratio 1:1, pitch 10mm, sets this parameter to 2500; pitch 5mm, set it to 1250.
Pr48	1st numerator of command pulse frequency	Need calculation Range:	Typical value: pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm:

Para. No.	Function	Value	Description
	multiplication	1~10000	Pr48＝10000 Pr4B＝pitch 5mm / pulse equivalent 0.001mm＝5000 Pr48/Pr4B=10000/5000=2/1
Pr4B	Denominator of the command pulse frequency multiplication	Need calculation Range: 1~10000	

6.1.5 Parameter Setting of PANASONIC MINAS A5 Servo Driver

Para. No.	Function	Value	Description
Pr5.28	LED initial status	6	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
Pr0.01	Select control mode	0	0: position mode 1: velocity mode 2: torque mode
Pr0.05	Selection of command pulse input	XX	0: Photo-coupler input (PULS1,PULS2,SIGN1,SIGN2) 1: Exclusive input for line driver (PULSH1, PULSH2, SIGNH1,SIGNH2) Note: generally, “1” is selected for this parameter.
Pr0.07	Command pulse input mode setup	3	Set command pulse input mode: command pulse + command direction, negative logic.
Pr0.08	Command pulse No. per motor circle	0	When it is set to “0”, parameters Pr0.09 and Pr0.10 are valid.
Pr0.09	1st numerator of command pulse frequency multiplication	Need calculation Range: $0\sim2^{30}$	Typical value: pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm: Pr0.09＝10000 Pr0.10＝pitch 5mm/ pulse equivalent 0.001mm＝5000 Pr0.09/Pr0.10=10000/5000=2/1
Pr0.10	Denominator of the command pulse frequency	Need calculation Range:	

Para. No.	Function	Value	Description
	multiplication	$0\sim2^{30}$	
Pr0.11	Output pulse No. per motor circle	2500	Range: 1 ~ 262144. Set it according to actual PG divider ratio. Pulse equivalent 0.001mm/p, without reduction box, pitch 10mm, sets this parameter to 2500; pitch 5mm, set it to 1250.
When the value of Pr0.08 is not "0", it can be calculated in terms of the following formula: $\text{Command pulse No. per motor circle} = \frac{\text{Screw pitch}}{\text{Pulse equivalent} * \text{Mechanical deceleration ratio}} = \frac{5\text{mm}}{0.001\text{mm/p}} = 5000$ When screw pitch is 5mm and pulse equivalent 0.001, the value of Pr0.08 is "5000".			

◆ **Attached List: the relationship among parameters Pr0.08, Pr0.09 and Pr0.10.**

Pr0.08	Pr0.09	Pr0.10	Description
$1\sim2^{20}$	– (no influence)	– (no influence)	Command Pulse Input → Encoder Resolution / [Setting Value of Pr0.08] → Position Command As shown above, the process is undergone in terms of the setting value of Pr0.08, not affected by the settings of Pr0.09 and Pr0.10.
0	0	$1\sim2^{30}$	Command Pulse Input → Encoder Resolution / [Setting Value of Pr0.10] → Position Command When the values of Pr0.08 and Pr0.09 are both set to "0", as shown above, the process is undergone in terms of the setting value of Pr0.10.
	$1\sim2^{30}$	$1\sim2^{30}$	Command Pulse Input → [Setting Value of Pr0.09] / [Setting Value of Pr0.10] → Position Command When the value of Pr0.08 is "0", but the value of Pr0.09 is not "0", as shown above, the process is underdone in terms of the setting values of Pr0.09 and Pr0.10.

6.1.6 Parameter Setting of MITSUBISHI MR-JE Servo Driver

Para. No.	Code	Function	Value	description
PA01	*STY	Operation mode	XXX0	_ _ _x: select position control mode.
PD24	MBR	Output assignation to CN1-23 pin	XX05	_ _ xx: select MBR (electromagnetic brake interlock).
PA06	CMX	Electronic gear numerator	Need calculation	CMX/CDV=command unit × servo motor resolution × mechanical deceleration ratio / pitch of screw. E.G., pitch 5 mm, encoder resolution 10000, deceleration ratio 1:1, pulse equivalent 0.001 mm, CMX/CDV=10000×0.001/5 = 2/1; When pulse equivalent = 0.0005mm, CMX/CDV = 1/1. Electronic gear ratio range: 1/50 ~ 500
PA07	CDV	Electronic gear denominator	Need calculation	
PC36	*DMD	Status display selection	00XX	_ _xx: status display selection at power-on. This is used to select a status display shown at power-on. 00: cumulative feedback pulses 01: servo motor speed 02: droop pulses 03: cumulative command pulses 04: command pulse frequency
PA13	*PLSS	Command pulse input form	0011	Set command pulse input form: pulse train+ sign, negative logic.
PA15	*ENR	Encoder output pulses	Need calculation	Range: 1~65535, set according to the parameter setting of “Frequency Division Pulses of PG (X4)”. Typical value: pulse equivalent 0.001, screw pitch 10mm without a reduction box, PA15=2500; screw pitch 5mm, PA15=1250.
PD03	*DI1L	Input assignation to CN1-15 pin	XX02	_ _xx: select SON under position control mode.

6.1.7 Parameter Setting of MITSUBISHI MR-E Servo Driver

Para. No.	Code	Function	Value	Description
0	*STY	Control mode selection and regenerative fittings	X0X0	Bit 0: set 0: select position control mode. Bit 1, select motor series: 0: HC-KFE; 1:HC-SFE; Bit 3, select regenerative apparatus, set 0: not use. Bit 4, select motor power.
1	MBR	Function selection 1	001X	Bit 0: input signal filter. If external input signal causes chattering due to noises, etc., input filter is used to suppress it. Bit 1: CN1-12 function selection, set “1”: electromagnetic brake interlock (MBR); set “0”: zero speed detection signal.
3	CMX	Electronic gear numerator	Need calculation	CMX/CDV=command unit × servo motor resolution × mechanical deceleration ratio / pitch of screw. E.G., pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm, CMX/CDV=10000×0.001/5 = 2/1; When pulse equivalent = 0.0005mm, CMX/CDV = 1/1. Electronic gear ratio range: 1/50 ~ 500
4	CDV	Electronic gear denominator	Need calculation	
18	*DMD	Status display selection	00XX	3: cumulative command pulses E: load inertia When the parameter is set [3], monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection to determine if there is electrical interference.
21	*OP3	Function selection 3 (command pulse format selection)	0001	Set pulse command input form: pulse train+ sign, negative logic
27	*ENR	Encoder output pulse	Need Calculation	Range: 1 ~ 65535. Set it according to actual PG divider ratio. Pulse equivalent 0.001mm/p, without reduction box, pitch 10mm, sets this parameter to 2500; pitch 5mm, set it to 1250.
41	*DIA	Signal input SON-ON,	0110	Bit 0: Servo-ON selection. [0]: servo on by external input; [1]: servo on all the time inside.

Para. No.	Code	Function	Value	Description
		LSP-ON and LSN-ON automatically selection		Bit 1: last signal of positive rotation range (LSP): [1]: auto servo on inside, without external wiring. Bit 3: last signal of negative rotation range (LSN): [1]: auto servo on inside and no need of external wiring.

6.1.8 Parameter Setting of DELTA ASDA-A Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection to determine if there is electrical interference.
P1-00	External pulse input type	ZYX	102	X=2: pulse + direction; Z=1: negative logic
P1-01	Control mode setup	ZYX1X0	0000	Z=0: during control mode switching, DIO is maintaining the set value. Since switching control mode is not used, Z=0 Y=0: forward rotation (CCW) (in terms of load); Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-32	Motor stop mode selection	YX	00	Y=0: when there is no servo enabled, motor dynamic brake occurs; Y=1: motor is free. X=0: motor stops instantly, X=1: motor stops with deceleration.
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	Need calculation	N1/M= encoder pulses × 4× pulse equivalent× mechanical deceleration ratio/ pitch. Representative value: encoder pulses =2500, pitch=5mm, pulse equivalent=0.001mm/p, deceleration ratio=1, calculation as below: N1/M= 2500×4×0.001/5 = 2 / 1, N1=2, M=1; When the multi-electronic gear ratio is not used, P2-60~ P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	
P2-10	Digital Input Pin DI1	X2X1X0	101	X1X0=01: digital input (DI1=SON) corresponds to 9th pin of CN1. X2 = 1: set DI1 input as NO (normally open)

Para. No.	Function	Format & Range	Value	Description
				a-contact point.
P2-15	Digital Input pin DI6	X2X1X0	100	Default factory setting of DI6 and DI7 are NC (normally closed) limit signal input pins; driver can't run without being connected to pin 32 and pin 31 of CN1. X2=1: set DI6 and DI7 inputs as NO (normally open) a-contact points; X1X0=00, limit signal input of the driver is not used.
P2-16	Digital Input Pin DI7	X2X1X0	100	
P2-17	Function setting for digital input pin DI8	X2X1X0	100	External EMG stop input is not used.
P2-21	Function setting for digital output pin DO4	X2X1X0	108	DO4 corresponds to pin 1 & pin 26, used as clamping-position brake signal of Z-axis; X2=1: set DO4 output as NO (normally open) a-contact point; X2=0: set DO4 output as NC (normally closed) b-contact point; X1X0=08: set pin 1 and pin 26 as BK+ and BK- respectively.
P2-22	Function setting for digital output pin DO5	X2X1X0	007	DO5 corresponds to pin 28 & pin 27, used as servo alarm signal. X2=0: set DO5 output as NC b-contact point. X1X0=07: set pin 28 and pin 27 as ALRM+ and ALRM- respectively.
P2-51	Servo ON (SON) setup		0	0: Servo ON must be triggered by numerical input signal. 1: when servo is powered, if there is no alarm signal, servo will be automatically on. Set 1 when there is no SON signal wire.

6.1.9 Parameter Setting of DELTA ASDA-A2 Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity

Para. No.	Function	Format & Range	Value	Description
				of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
P1-00	External pulse train input type	ZYX	102	X=2: pulse + direction; Z=1: negative logic
P1-01	Set control mode	ZYX1X0	0000	Z=0: during control mode switching, DIO is maintaining the set value. Since switching control mode is not used, Z=0; Y=0: forward rotation (CCW) (from the view of load); Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	Need calculation	$\frac{P1-44}{P1-45}=\frac{\text{Encoder resolution}*\text{Pulse equivalent}*\text{Deceleration ratio}}{\text{Screw pitch}}$ When encoder resolution is 1280000, screw pitch 5mm, pulse equivalent 0.001, in direct coupling, $\frac{P1-44}{P1-45}=\frac{1280000\times 0.001}{5}=\frac{256}{1}$ When the multi-electronic gear ratio is not used, P2-60 ~P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	
P1-46	Detector output pulse No. setting	20 ~ 320000	Need calculation	Set output pulse number for the detector according to actual PG divider ratio. Pulse equivalent 0.001mm/p, without reduction box, pitch 10mm, sets this parameter to 10000; pitch 5mm, sets it to 5000.
P2-10	Digital Input Pin 1 (DI1)	X2X1X0	101	X1X0=01: digital input (DI1 = SON) corresponds to 9th pin of CN1. X2=1: set DI1 input as NO (normally open) a-contact point.

Para. No.	Function	Format & Range	Value	Description
P2-15	Function setting for digital input pin DI6	X2X1X0	100	Default factory setting of DI6 and DI7 is NC (normally closed) limit signal input; driver can't run without being connected to pin 32 and pin 31 of CN1. X2=1: set DI6 and DI7 inputs as NO a-contact points. X1X0=00, limit input of driver is not used.
P2-16	Function setting for digital input pin DI7	X2X1X0	100	
P2-17	Function setting for digital input pin DI8	X2X1X0	100	External EMG stop input is not used.
P2-21	Function setting for digital output pin DO4	X2X1X0	108	DO4 corresponds to pin 1 & pin 26, used as clamping-position brake signal of Z-axis; X2=1: set DO4 output as NO (normally open) a-contact point; X2=0: set DO4 output as NC (normally closed) b-contact point; X1X0=08: set pin 1 and pin 26 as BK+ and BK- respectively.
P2-22	Function setting for digital output pin DO5	X2X1X0	007	DO5 corresponds to pin 28 & pin 27, used as servo alarm signal. X2=0: set DO5 output as NC b-contact point. X1X0=07: set pin 28 and pin 27 as ALRM+ and ALRM- respectively.

6.1.10 Parameter Setting of DELTA ASDA-B Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
P1-00	External pulse train input type	ZYX	102	X=2: pulse + direction; Z=1: negative logic

Para. No.	Function	Format & Range	Value	Description
P1-01	Set control mode	YX1X0	000	Y=0: forward rotation (CCW) (from the view of load); Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-32	Motor stop mode	YX	00	Y=0: when there is no servo enabled, motor dynamic brake occurs; Y=1: motor is free. X=0: motor stops instantly, X=1: motor stops with deceleration.
P1-44	Electronic Gear Ratio (Numerator)(N1)	1~32767	Need calculation	N1/M= mechanical deceleration ratio × 4 × encoder pulses × pulse equivalent / pitch. Representative value: encoder pulses=2500, pitch =5mm, pulse equivalent=0.001 mm/p, deceleration ratio = 1, calculation as below: N1 / M = 2500×4×0.001/5 = 2/1, N1=2, M=1; When the multi-electronic gear ratio is not used, P2-60 ~P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator)(M)	1~32767	Need calculation	
P2-10	Function setting for digital input pin DI1	X2X1X0	101	X1X0=01: digital input (DI1 = SON) corresponds to 17th pin of CN1. X2=1: set DI1 input as NO (normally open) a-contact point.
P2-15	Function setting for digital input pin DI6	X2X1X0	100	Default factory setting of DI6 is NC (normally closed) limit signal input; driver can't run without being connected to pin 32 and pin 31 of CN1. X2=1: set DI6 input as NO a-contact point. X1X0=00, limit input of the driver is not used.
P2-18	Function setting for digital output pin DO1	X2X1X0	108	DO1 corresponds to 16th pin, as clamping-position brake signal of Z-axis; X2=1: set DO1 output as NO a-contact point; X2=0: set DO1 output as NC b-contact point; X1X0=08: set 16th pin as BK+.
P2-20	Function setting for digital output pin DO3	X2X1X0	007	DO3 corresponds to pin 1, used as servo alarm signal. X2=0: set DO3 as NC b-contact point. X1X0=07: set pin 1 as ALRM+.

6.1.11 Parameter Setting of DELTA ASDA-B2 Servo Driver

Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display		02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine whether there is electrical interference.
P1-00	External pulse train input type	ZYX	102	X=2: pulse + direction; Z=1: negative logic
P1-01	Set control mode	ZYX1X0	0000	Z=0: during control mode switching, DIO is maintaining the set value. Since switching control mode is not used, Z=0; Y=0: forward rotation (CCW) (from the view of load); Y=1: the rotation direction is reversed. X1X0=00: position control mode
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	Need calculation	N1/M= mechanical deceleration ratio × 4 × encoder pulses × pulse equivalent/ pitch Representative value: encoder pulses =40000, pitch =5mm, pulse equivalent=0.001, deceleration ratio = 1, calculation as below: N1 / M = 40000×4×0.001/5 = 32/1, N1=32, M=1; When the multi-electronic gear ratio is not used, P2-60 ~P2-62 are not required.
P1-45	Electronic Gear Ratio (Denominator) (M)	1~32767	Need calculation	
P2-10	Function setting for digital input pin DI1	X2X1X0	101	X1X0=01: digital input (DI1 = SON) corresponds to 9th pin of CN1. X2=1: set DI1 input as NO (normally open) a-contact point.
P2-15	Function setting for digital input pin DI6	X2X1X0	100	Default factory setting of DI6 and DI7 is NC (normally closed) limit signal input; driver can't run without being connected to pin 32 and pin 31 of CN1. X2=1: set DI6 and DI7 inputs as NO a-contact points. X1X0=00, limit input of the driver is not

Para. No.	Function	Format & Range	Value	Description
				used.
P2-16	Function setting for digital input pin DI7	X2X1X0	100	
P2-17	Function setting for digital input pin DI8	X2X1X0	100	External EMG stop input is not used.
P2-18	Function setting for digital output pin DO1	X2X1X0	108	DO1 corresponds to pin 6 & pin 7, used as clamping-position brake signal of Z-axis; X2=1: set DO1 output as NO (normally open) a-contact point; X2=0: set DO1 output as NC (normally closed) b-contact point; X1X0=08: set pin 6 and pin 7 as BK- and BK+ respectively.
P2-22	Function setting for digital output pin DO5	X2X1X0	007	DO5 corresponds to pin 28 & pin 27, used as servo alarm signal. X2=0: set DO5 output as NC b-contact point. X1X0=07: set pin 28 and pin 27 as ALRM+ and ALRM- respectively.

6.1.12 Parameter Setting of SANYO PY Servo Driver

Para. No.	Abbr.	Name	Standard Value	Setting Range	Unit	Remark
1-2	EGER	Electronic gear ratio	4/1	1/32767 to 32767/1		Depends on the specific encoder resolution. The formula of electronic gear ratio of servo driver is as below: Electronic gear ratio numerator =mechanical deceleration ratio × 4× pulse No. per encoder circle; Electronic gear ratio

Para. No.	Abbr.	Name	Standard Value	Setting Range	Unit	Remark
						denominator = (screw pitch / pulse equivalent) E.G. In Weihong system, the default pulse equivalent is 0.001mm/p, screw pitch is 5mm, pulse number per encoder circle is 2000 shaft coupling direct drag, currently the numerator of the electronic gear ratio is 8, and the denominator is 5. (Select an incremental type encoder)
1-16	MENP	Pulse amount of the motor encoder 1. Set the pulse amount of the motor encoder; 2. Standard configuration of the encoder pulse No. is as below. Incremental encoder omitting wiring: --2000P/R Absolute encoder:--2048P/R		500 to 65535	P/R	
2-0	PMOD	Pulse format of position command: Our system uses: direction + pulse format, the parameters are shown as following:				

<table>
<tr><th>Para. No.</th><th>Abbr.</th><th>Name</th><th>Standard Value</th><th>Setting Range</th><th>Unit</th><th>Remark</th></tr>
<tr><td></td><td></td><td colspan="5">
PMOD
<table>
<tr><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td></tr>
</table>
When bit 7=0
<table>
<tr><th>Bit 1</th><th>Bit 0</th><th>Command Pulse Input Digital Filter
Min. Pulse Width</th></tr>
<tr><td>0</td><td>0</td><td>0.8μs</td></tr>
<tr><td>0</td><td>1</td><td>0.2μs</td></tr>
<tr><td>1</td><td>0</td><td>0.4μs</td></tr>
<tr><td>1</td><td>1</td><td>1.6μs</td></tr>
</table>
When bit 7=1
<table>
<tr><th>Bit 1</th><th>Bit 0</th><th>Command Pulse Input Digital Filter
Min. Pulse Width</th></tr>
<tr><td>0</td><td>0</td><td>3.2μs</td></tr>
<tr><td>0</td><td>1</td><td>0.8μs</td></tr>
<tr><td>1</td><td>0</td><td>1.6μs</td></tr>
<tr><td>1</td><td>1</td><td>6.4μs</td></tr>
</table>
<table>
<tr><th>Bit6</th><th>Bit5</th><th>Command Pulse Format</th></tr>
<tr><td>1</td><td>0</td><td>Direction + Pulse</td></tr>
</table>
Switch of Digital Filter
<table>
<tr><td>0</td><td>High Speed</td></tr>
<tr><td>1</td><td>Low Speed (1/4)</td></tr>
</table>
</td></tr>
<tr><td>4-3</td><td>TYPE</td><td colspan="3">Control mode:
*Select one control mode from position, velocity, and torque modes.
<table>
<tr><th>Selection Item</th><th>Content</th></tr>
<tr><td>Position</td><td>Position control mode</td></tr>
<tr><td>Velocity</td><td>Velocity control mode</td></tr>
<tr><td>Torque</td><td>Torque control mode</td></tr>
<tr><td>Velo ↔Torq</td><td>Velocity ↔ Torque switch mode</td></tr>
<tr><td>Posi↔Torq</td><td>Position↔Torque switch mode</td></tr>
<tr><td>Posi↔Velo</td><td>Position↔Velocity switch mode</td></tr>
</table>
Referring to the switch type, the requisite control mode can be selected from pin 36 or 35 of the CN1.
Func3, set Bit7 as 0: pin 36 is enabled.
set Bit7 as 1:pin 35 is enabled.
$ $ $: standard value varies with the reset setup (leave factory setting).</td><td>6 types</td><td>Our system selects position control mode.</td></tr>
</table>

6.1.13 Parameter Setting of SANYO R Servo Driver

Para. No.	Parameter Name	Set Value	Remarks
Group 0, parameter setting of tuning mode			
00	Setting of the tuning mode	00	Set as auto tuning mode
Group 8, setting of the control parameters			
00	Polarity of position input	00	Position command mode: positive rotation effective
11	Input command mode	02	Pulse + negative logic
15	Setting of electronic gear	8/5	It depends on the resolution of the specific encoder. E.G.: incremental encoder 2000, motor needs 2000 ×4=8000 pulses per circle. And pulse equivalent of Weihong control card is 0.001mm/p, it needs 1000 pulses to move 1mm along line, in other words, if the screw pitch is 5, so, to move 5mm along line needs 5000 pulses, so F=8000/5000=8/5.
Group 9, setting of function effective			
05	Servo ON selection	02	Select servo ON state.
02	Servo alarm elimination	10	Make the function of servo alarm effective
Setting of the system parameters			
02	Encoder selection	00	Standard incremental encoder. The parameter depends on the specific situation, what we list is only the representative one.
03	Encoder resolution	2000	500－65535, set the encoder resolution manually.
08	Control mode selection	02	Select position control mode

6.1.14 Parameter Setting of SANYO Q Servo Driver

Para. No.	Parameter Name	Set Value	Remarks
Group 1			
GER1	Electronic gear	1/1	Set electronic gear ratio for position command pulse.

Para. No.	Parameter Name	Set Value	Remarks
	ratio 1		E.G., incremental encoder 2000, motor needs 2000 ×4=8000 pulses per circle. And pulse equivalent of Weihong control card is 0.001mm/p, it needs 1000 pulses to move 1mm along line, in other words, if the screw pitch is 5, so, to move 5mm along line needs 5000 pulses, so F=8000/5000=8/5.
GER2	Electronic gear ratio 2	1/1	This setting is the same as that of electronic gear ratio 1 and activated during electronic gear switching.
Group 4			
PA400	Command pulse selection	00H	Set position command pulse as “pulse + direction”.
Group 8			
S-ON	Servo ON	02H	Select servo ON state.
AL-RST	Alarm reset	10H	Make the function of servo alarm effective
Setting of the system parameters			
01	Encoder selection	00	Standard incremental encoder. The parameter depends on the specific situation, what we list is only the representative one.
03	Incremental encoder resolution	2000	500－65535, set the encoder resolution manually.
08	Control mode selection	02	Select position control mode.

6.1.15 Parameter Setting of KT270 Servo Driver

Para. No.	Parameter Name	Value	Description
PA4	Control mode selection	0	The control mode of the driver can be set through this parameter: 0: position control mode; 1: speed control mode; 2: trial run control mode; 3: JOG control mode.

Para. No.	Parameter Name	Value	Description
PA12	Numerator of position command pulse ratio	2	Set the ratio of the position command pulse (electronic gear). Under position control mode, with the setting of the PA12 and PA13, it is convenient to match with pulse source of each type, which can reach the user's perfect control resolution (that is angle/pulse) Expression: $P \times G = N \times C \times 4$ P: pulse amount of the input command; G: electronic gear ratio, G=ratio numerator / ratio denominator. N: circle number that the motor rotates; C: each circle line number of photo electricity encoder, C of our system =2500. E.G.: input 6000 command pulses to make the servo motor rotate one circle, $G = \frac{N \times C \times 4}{P} = \frac{1 \times 2500 \times 4}{6000} = \frac{5}{3}$ So set PA12 as 5 and PA13 as 3. We recommend the range of electronic gear ratio as: $\frac{1}{50} \le G \le 50$
PA13	Denominator of position command pulse ratio	1	Refer to parameter PA12.
PA14	Input mode of the position command pulse	0	Set the input mode of the position command pulse; there are following three modes can be selected by setting the parameter: 0: pulse + symbol; 1: positive rotation pulse/negative rotation pulse; 2: two orthogonal pulses inputs Default setting is 0: pulse + symbol, negative logic.
PA20	Invalid input on the end of the stroke	1	0: Valid stroke end of LSP, LSN positive rotation, negative rotation. When switch LSP is connected, driving of the positive rotation is allowed; When switch LSP is disconnected, driving of the positive rotation is prohibited (torque of the positive direction is 0). LSN is the same as LSP. If LSP and LSN are all disconnected, the abnormal alarming of driving prohibited (NO.7) will occur.

Para. No.	Parameter Name	Value	Description
			1: Invalid stroke end of LSP, LSN positive rotation, negative rotation. No matter which state of the switch LSP and LSN is in, driving of positive rotation and negative rotation are all allowed. Simultaneously, even if LSP and LSN are all disconnected, abnormal alarming of driving prohibited (NO.7) will not occur. 2: Invalid stroke end of LSP, LSN positive rotation, negative rotation, and SON is forced to be effective. (Note: SON forcedly effective is only used for motor debugging. In normal use, we suggest controlling the state of SON by input port.) 3: Valid stroke end of LSP, LSN positive rotation, negative rotation. When switch LSP is connected, driving of the positive rotation is allowed; When switch LSP is disconnected, driving of the positive rotation is prohibited (the speed of positive direction is 0, but the torque is not 0). LSN is the same as LSP. When LSP and LSN are all disconnected, abnormal alarming of driving prohibited (NO.7) will not occur.

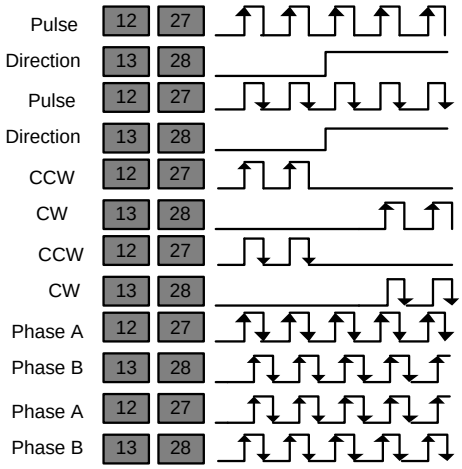
6.1.16 Parameter Setting of FUJI FALDIC-β Servo Driver

Para. No.	Name	Value	Description
01	Command pulse numerator α	Need calculation 1~32767	Command pulse numerator and denominator are also equal to those of the electronic gear ratio. α/ β=encoder resolution× pulse equivalent× mechanical deceleration ratio / screw pitch. Typical value: encoder resolution 65536, pitch 5mm, pulse equivalent 0.001, mechanical deceleration ratio 1, α / β=65536×0.001 / 5=8192 / 625, So α=8192, β=625.
02	Command pulse denominator β	Need calculation 1~32767	
03	Pulse string input form	0	Set the input mode of pulse string as: instruction + symbol, that is ‘pulse + direction’.

04	Direction of rotation switch	0 or 1	Set 0: Positive direction: Forward rotation (CCW); Set 1: Positive direction: Reverse rotation (CW).
10	CONT1 signal distribution	1	CONT1 is distributed as RUN (i.e. SON); if not distributed, CONT1 will be auto ON if there is no alarming when powered.
11	CONT2 signal distribution	2	CONT2 is distributed as RST (i.e. servo alarming clearance CLR). When 12, 13, 14 are 0, that is CONT3, CONT4 and CONT5 can't be distributed as OT over-travel or EMG (external emergency stop).
15	OUT1 signal distribution	1	Set 1, OUT1 is distributed as a-contact point of alarming output; Set 2, OUT1 is distributed as b-contact point of alarming detection.
27	Parameter write-protection	0 or 1	Set 0, write-enable. Set 1, write-protected.
74	CONT Always ON 1	1	Its initial value is 0, and it is set "1" here to enable servo (RUN).

6.1.17 Parameter Setting of STONE GS Servo Driver

Para. No.	Para. Name	Value	Description
F0f	Electronic gear ratio numerator	2	Electronic gear ratio of position mode: 4× pulse frequency fed back by servo encoder = command pulse frequency× F0f / F10; value of F0f / F10 must be within 1/100~100. (calculation with pitch 10mm)
F10	Electronic gear ratio denominator	1	
F00	Control mode selection	2	0: External speed running mode; make sure the value and direction of motor speed according to the external analog -10V ~ +10V signal of CN2-16, 17; 1: Internal speed running mode; make sure the value and direction of motor speed according to the setting of parameter F33, F35, F37, F39 and the port status of CN2-9, CN2-25; 2: Position pulse running mode; accept the input of external position pulse and direction level signal; 3: Jog mode; make sure the motor speed in terms of parameter setting of F3b, and control the rotation direction by the direction keystroke ▼ and ▲; 4: Torque mode; make sure the value and direction of motor torque according to the external analog -10V ~ +10V signal of CN2-43, 1;

<table>
<tr><th>Para. No.</th><th>Para. Name</th><th>Value</th><th>Description</th></tr>
<tr><td></td><td></td><td></td><td>5~10: Mixed mode; select mode according to the port input status of CN2-24:
<table>
<tr><th rowspan="2">F00 Value</th><th colspan="2">CN2-24 Interface Status</th></tr>
<tr><th>OFF (Mode One)</th><th>ON (Mode Two)</th></tr>
<tr><td>5</td><td>Position Pulse Mode</td><td>External Speed Running Mode</td></tr>
<tr><td>6</td><td>Position Pulse Mode</td><td>Internal Speed Running Mode</td></tr>
<tr><td>7</td><td>Position Pulse Mode</td><td>Torque Mode</td></tr>
<tr><td>8</td><td>Internal Speed Running Mode</td><td>External Speed Running Mode</td></tr>
<tr><td>9</td><td>Internal Speed Running Mode</td><td>Torque Mode</td></tr>
<tr><td>10</td><td>External Speed Running Mode</td><td>Torque Mode</td></tr>
</table></td></tr>
<tr><td>F2e</td><td>Pulse input mode selection</td><td>2</td><td>Command pulse string mode selection of position mode:
1 – Single pulse train positive logic
2 – Single pulse train negative logic
3 – Double pulse train positive logic
4 – Double pulse train negative logic
5 – Orthogonal pulse positive logic
6 – Orthogonal pulse negative logic
 
</td></tr>
</table>

6.1.18 Parameter Setting of TECO TSDA Servo Driver

<table>
<tr><th>Para. No.</th><th>Function</th><th>Value</th><th colspan="3">Description</th></tr>
<tr><td rowspan="6">Pn010-1</td><td rowspan="6">Set control mode</td><td rowspan="6">1</td><td rowspan="2">Value</td><td>Control mode</td><td></td></tr>
<tr><td>CN1 Pin12 open circuit</td><td>CN1 Pin12 closed circuit</td></tr>
<tr><td>0</td><td>Speed control</td><td>Speed control</td></tr>
<tr><td>1</td><td>Position control</td><td>Position control</td></tr>
<tr><td>2</td><td>Torque control</td><td>Torque control</td></tr>
<tr><td>3</td><td>Speed control</td><td>Speed control</td></tr>
</table>

Para. No.	Function	Value	Description		
			4	Position control	Position control
			5	Torque control	Torque control
Pn010-2	Set the pulse input format under position control mode	0	Value	The format of pulse input	
			0	Pulse + direction	
			1	Dipulse	
			2	A/B phase difference	
Pn010-3	Set rotation direction of motor	1	Value	Function	
			0	Motor rotates anti-clockwise with the input of positive command.	
			1	Motor rotates clockwise with the input of positive command.	
Pn021	Electronic gear ratio numerator	5	The input pulse amount will be multiplied by the ratio before output. Ratio range of parameter 21 to 22: 1/127＜parameter 21/ parameter 22＜127		
Pn022	Electronic gear ratio denominator	1			
Pn011-4	Set the value of Pin20 of CN1	1	Value	Function	
			0	Output of “0” speed signal	
			1	Output of brake signal	
Pn013-1	Set the maximum pulse frequency received by the driver under position control mode	7	It can correct the phenomenon of unauthorized over-travel. Received frequency is divided into 8 segments from 500Kpps to 200Kpps. “0” indicates 500Kpps while “7” 200Kpps.		

CAUTION

For the parameter setting of a specific driver, refer to the driver manual of the specific brand.

6.2 Wiring Diagram of Driver and Terminal Board

Wiring diagrams in this part are the wiring diagrams of CNC system-axes control-driver motion. When it is required to use one axis of the CNC system to control the motion of two drivers, the wiring diagram is as shown in Figure 2 in section 6.2.2 and Figure 4 in section 6.2.6 (taking YASKAWA driver and DELTA driver as an example; for YASKAWA server, its alarm signal wiring is NC type, while for DELTA server, its alarm signal wiring is NO type).

6.2.1 Wiring Diagram of WISE Servo Driver

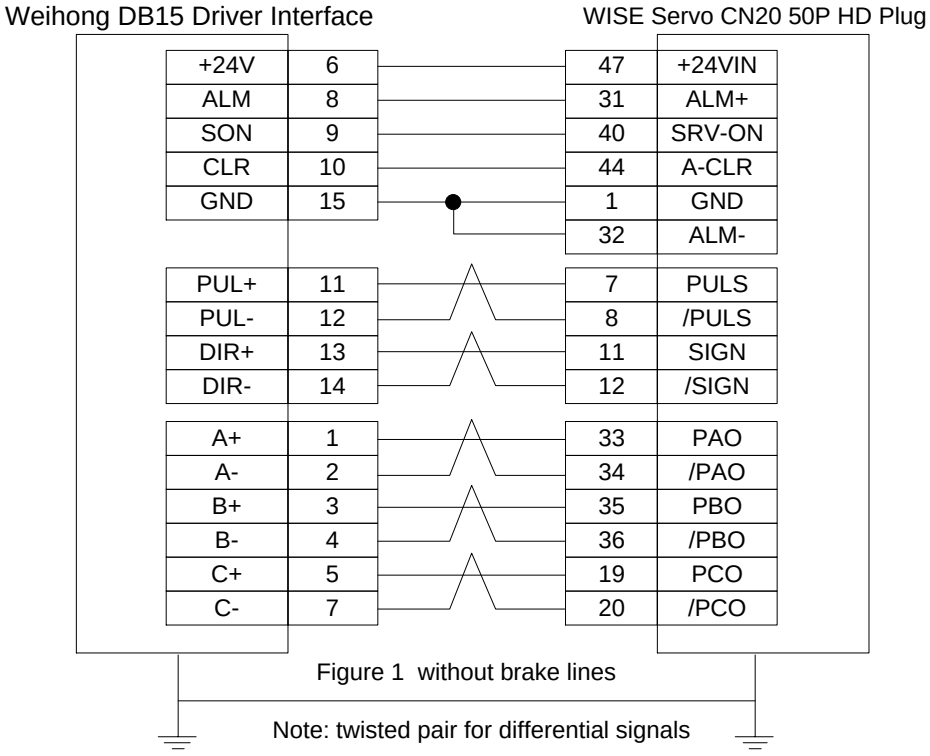


Figure 1 without brake lines

Note: twisted pair for differential signals

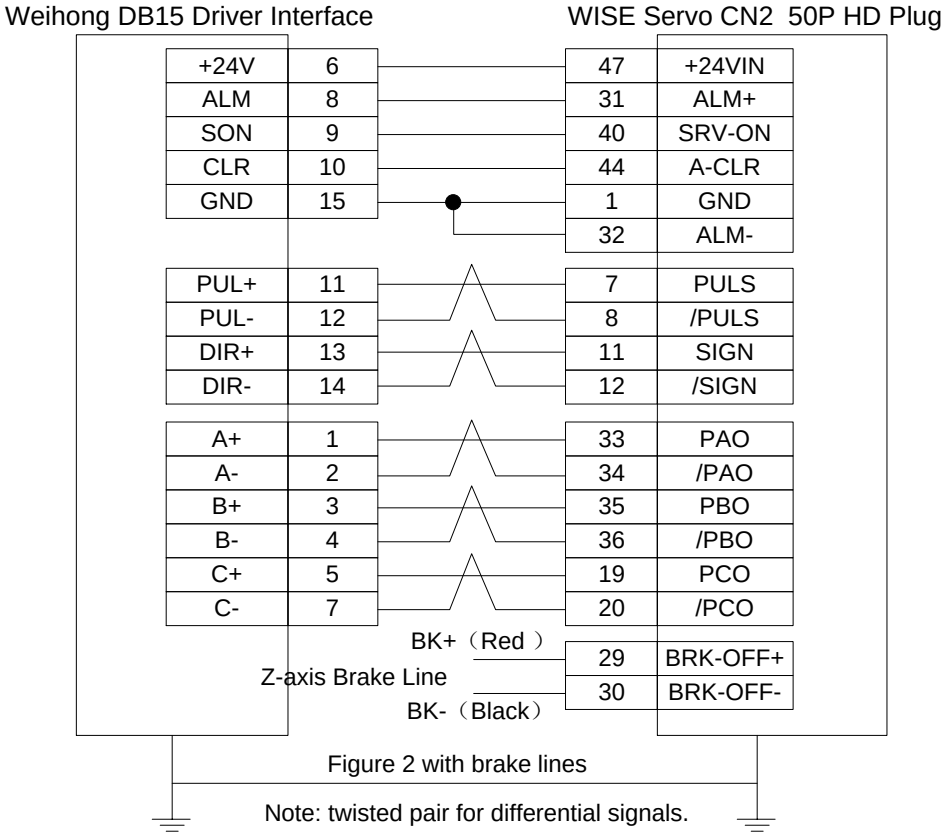


Figure 2 with brake lines

Note: twisted pair for differential signals.

6.2.2 Wiring Diagram of YASKAWA AC Servo Driver

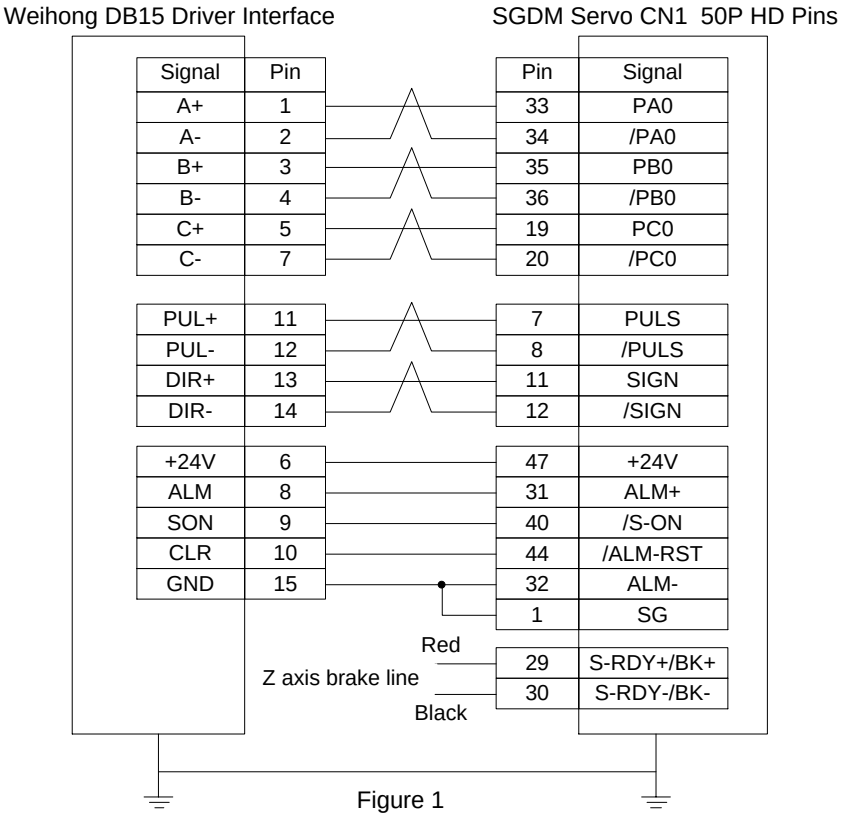


Figure 1

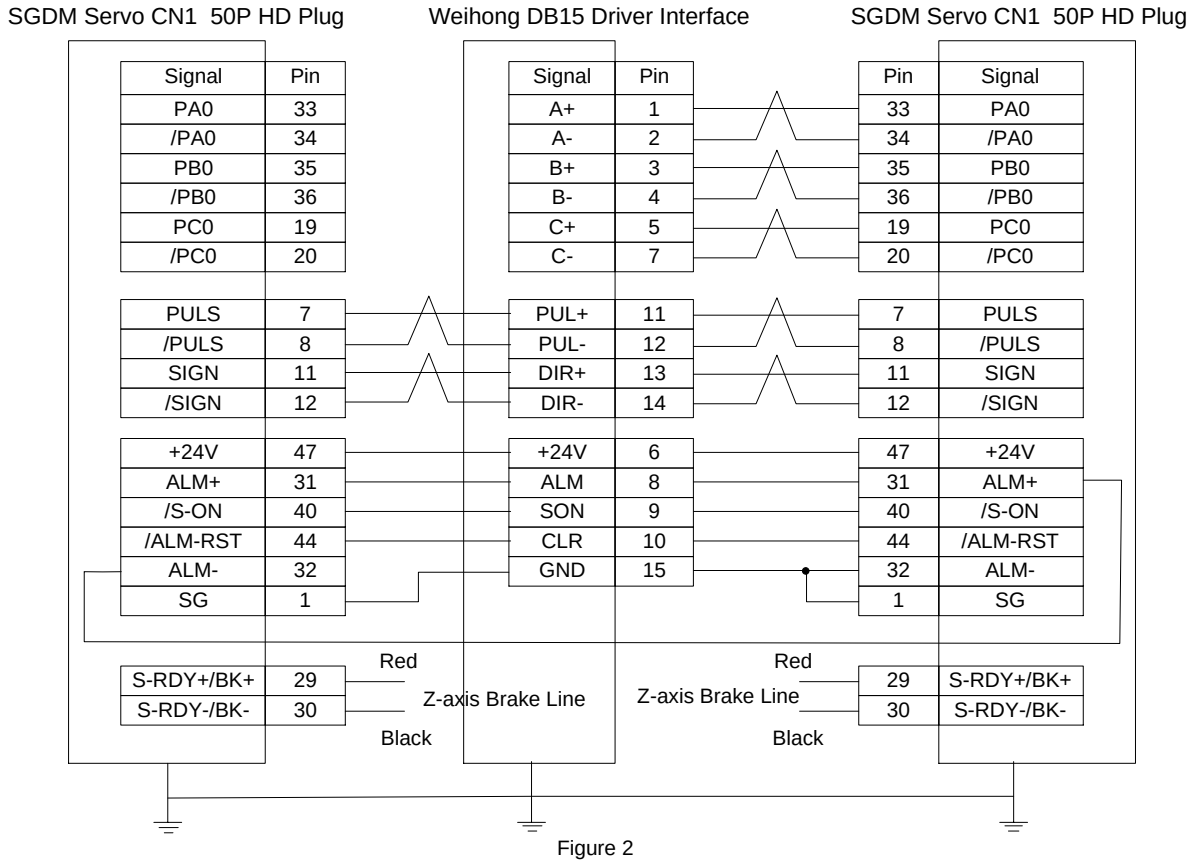
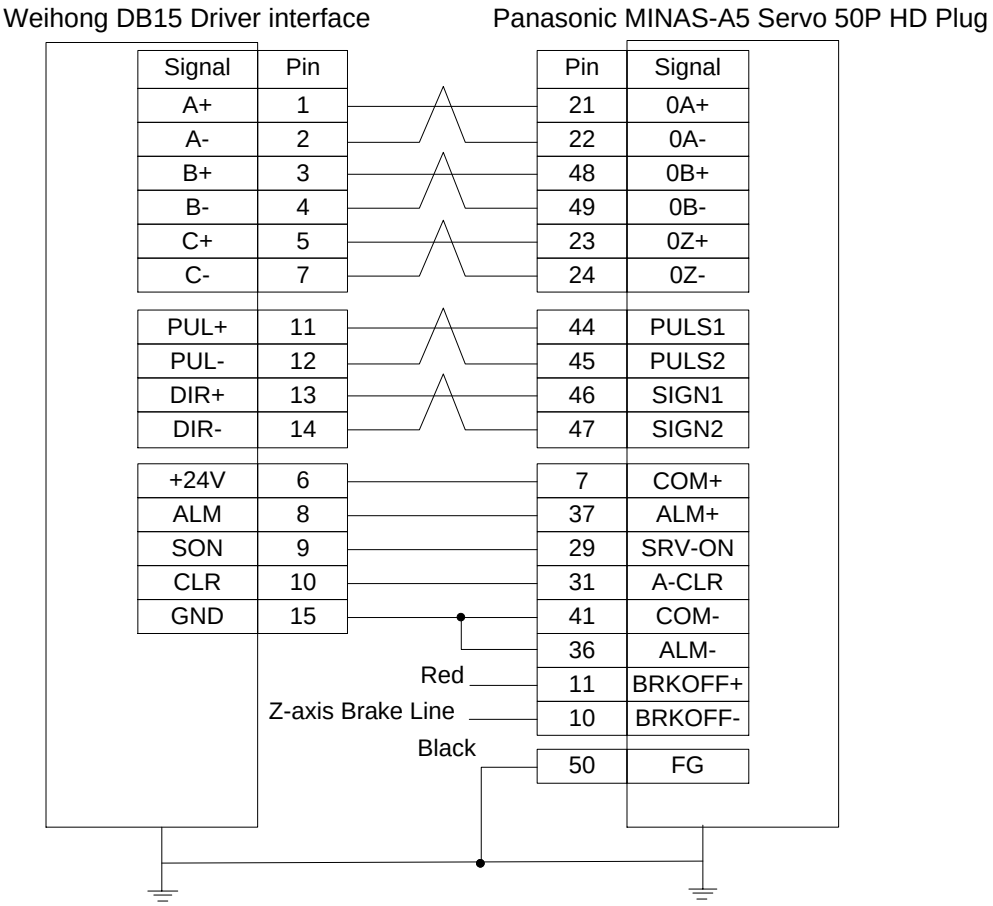


Figure 2

6.2.3 Wiring Diagram of PANASONIC AC Servo Driver



6.2.4 Wiring Diagram of MITSUBISHI MR-JE Servo Driver

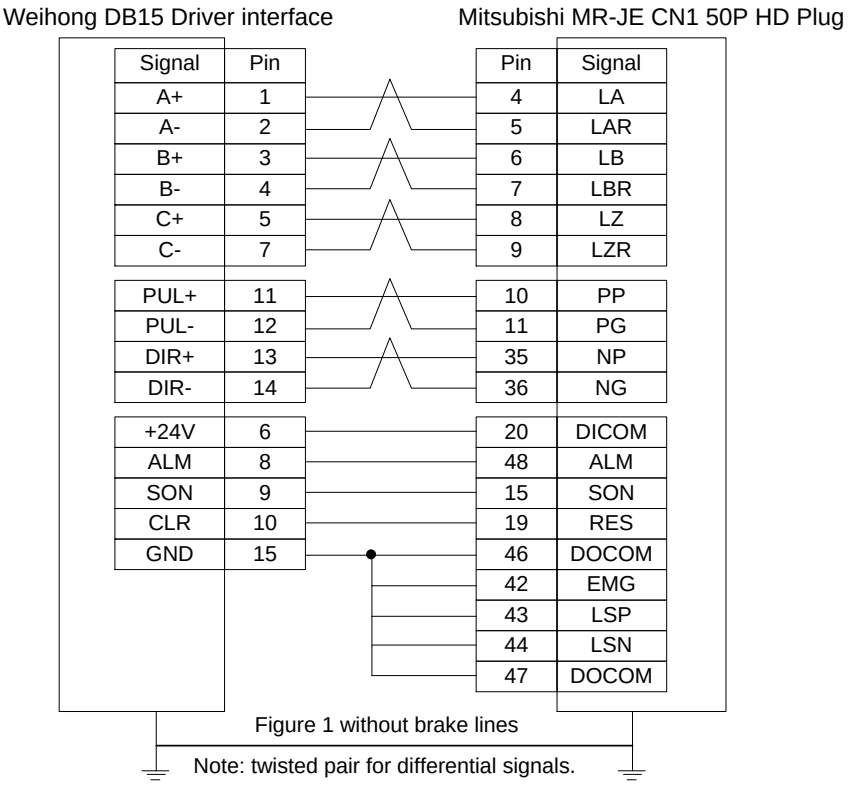


Figure 1 without brake lines

Note: twisted pair for differential signals.

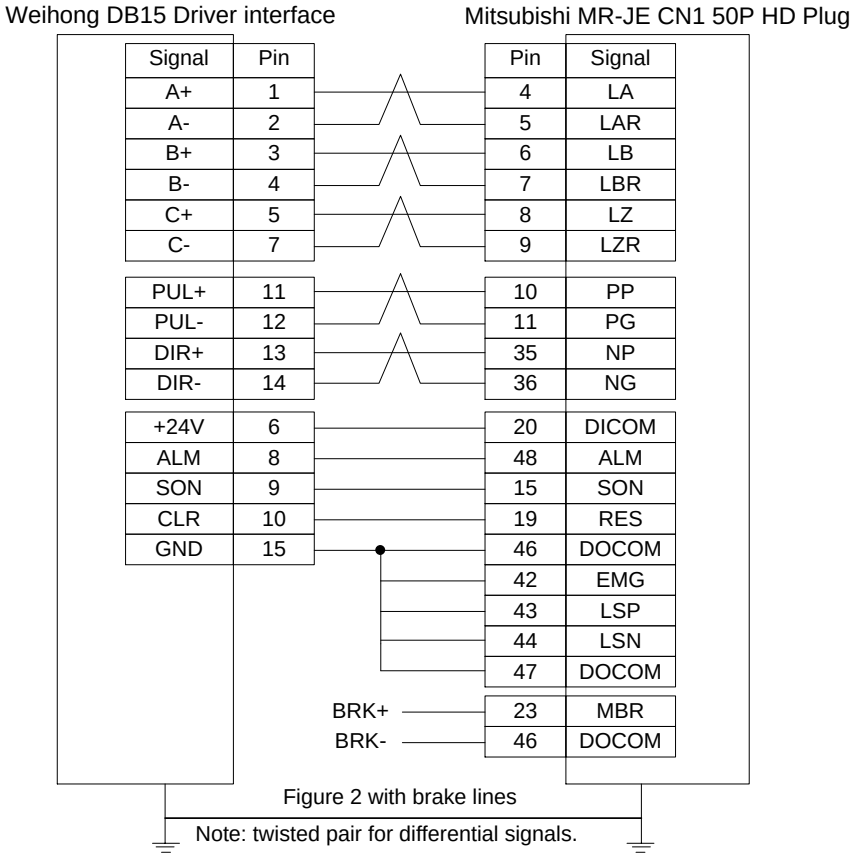
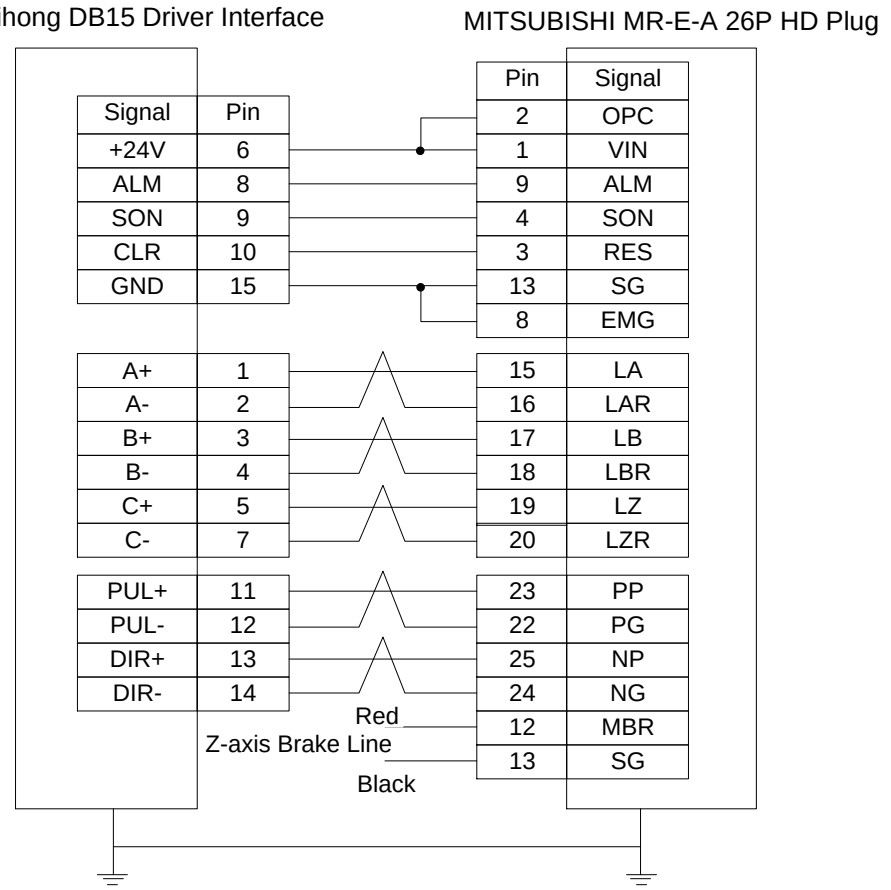


Figure 2 with brake lines

Note: twisted pair for differential signals.

6.2.5 Wiring Diagram of MITSUBISHI MR-E Servo Driver



6.2.6 Wiring Diagram of DELTA Servo Driver

DELTA ASDA-A, ASDA-A2 and ASDA-AB use the same cable. Among them, the wiring pins of ASDA-A2 and ASDA-AB are totally the same. As for ASDA-A, with PULSE as 41 and /PULSE as 43, its pulse signal pins are opposite to those of ASDA-A2 and ASDA-AB, but the other wiring pins are totally the same. For the detailed parameters settings, see section 6.1.9 and section 6.1.11.

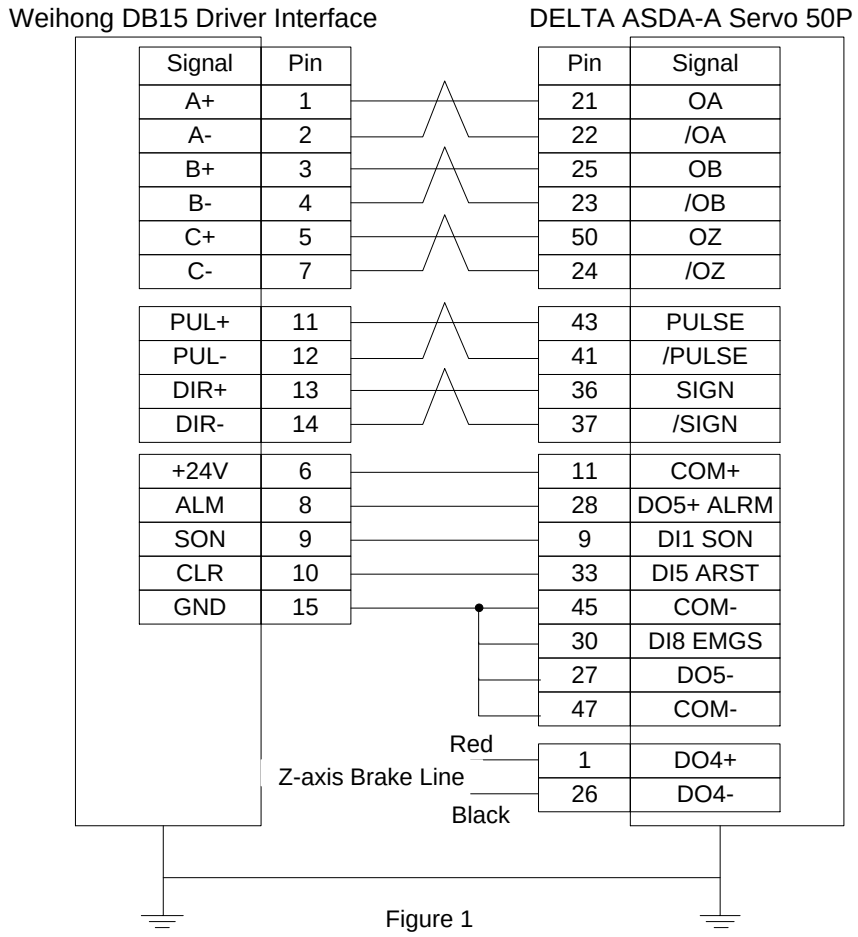


Figure 1

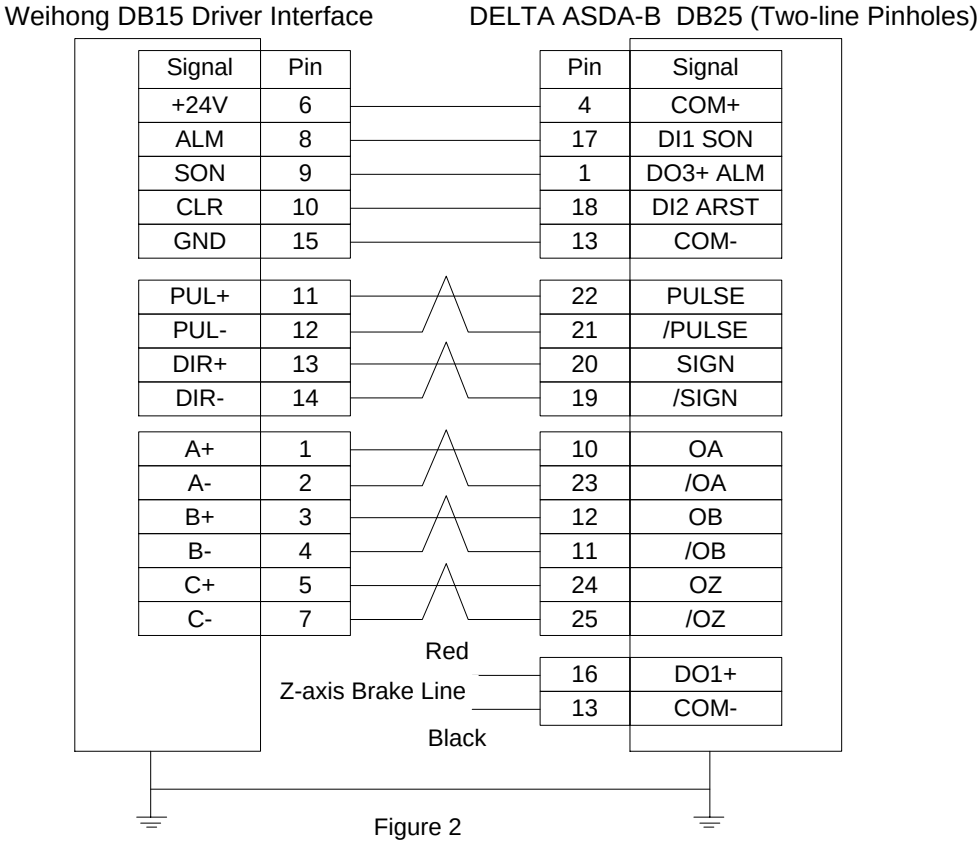


Figure 2

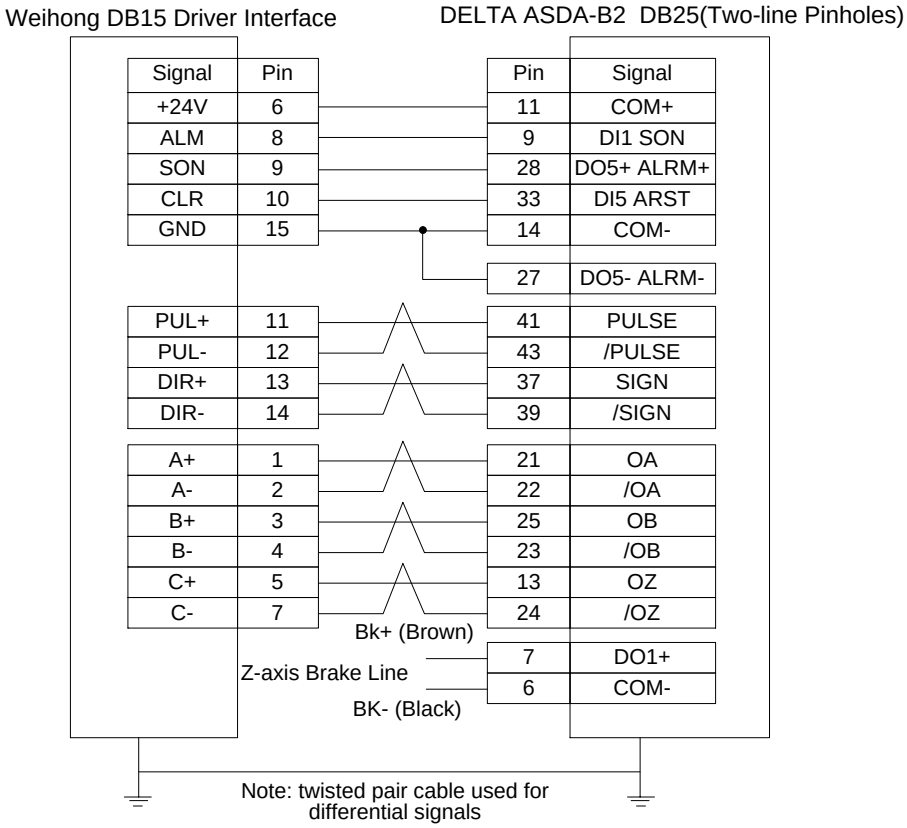


Figure 3

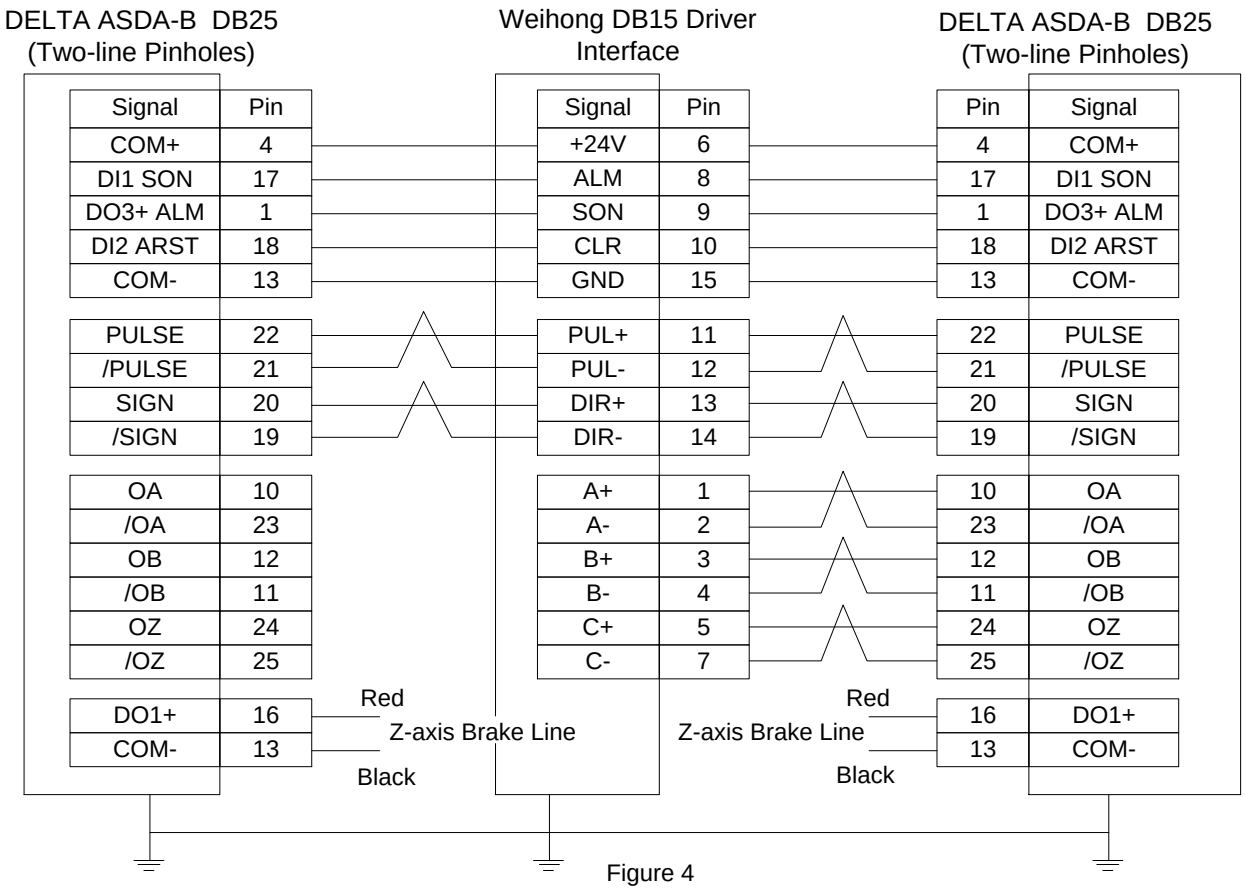
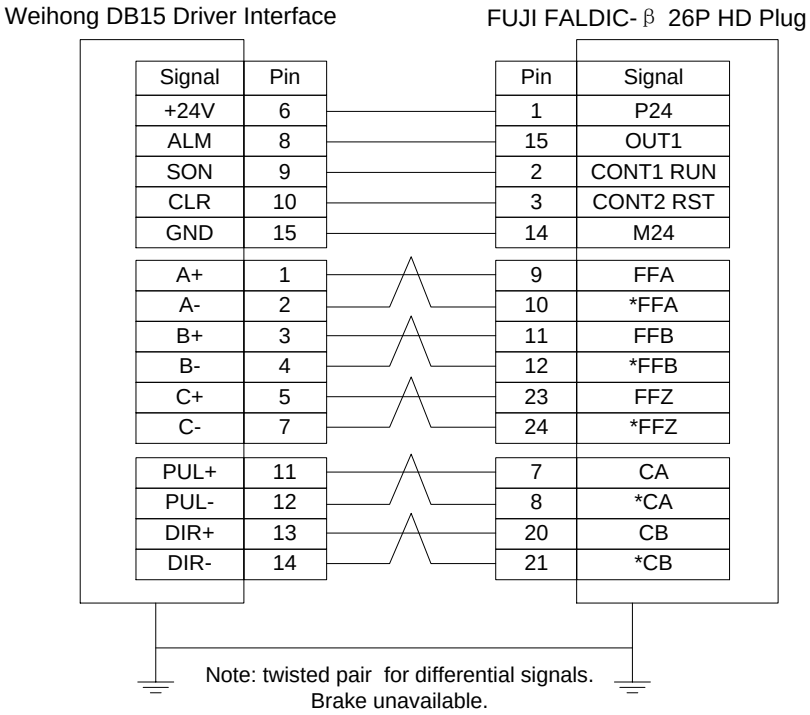
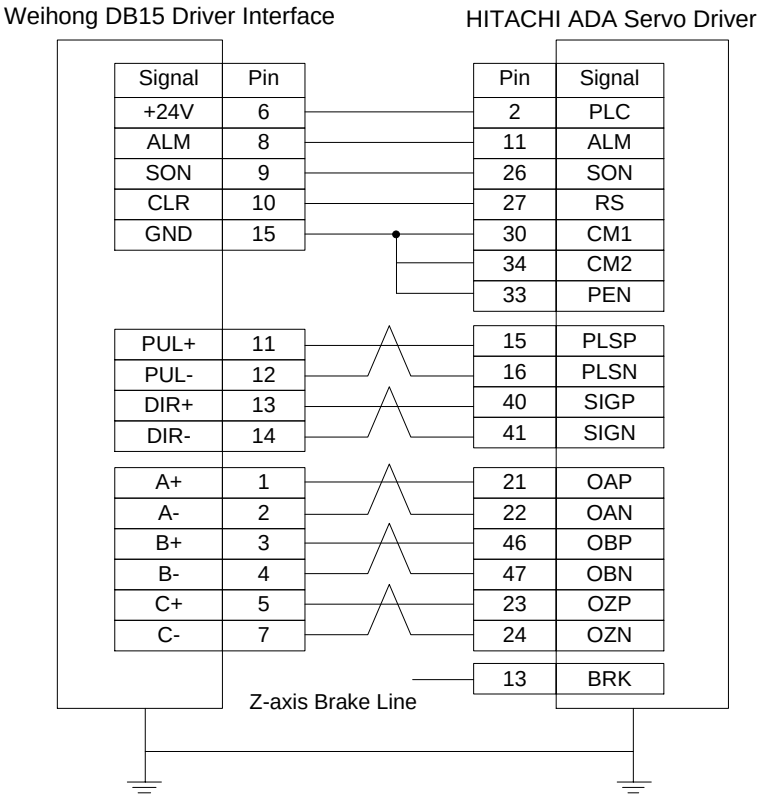


Figure 4

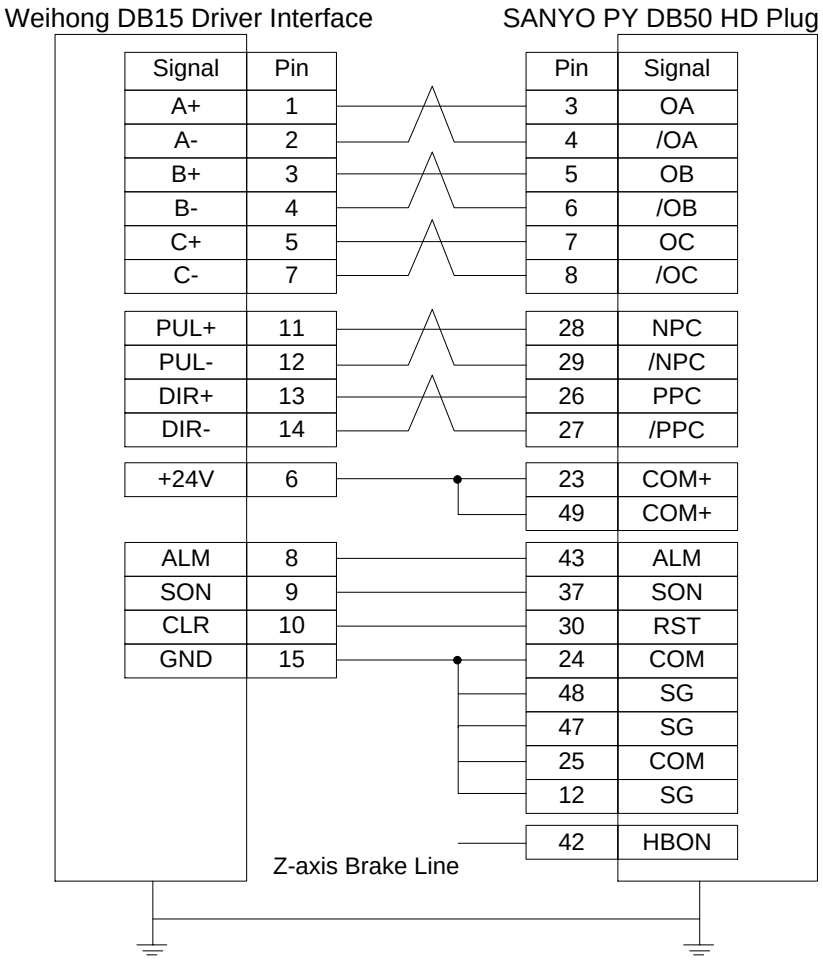
6.2.7 Wiring Diagram of FUJI Servo Driver



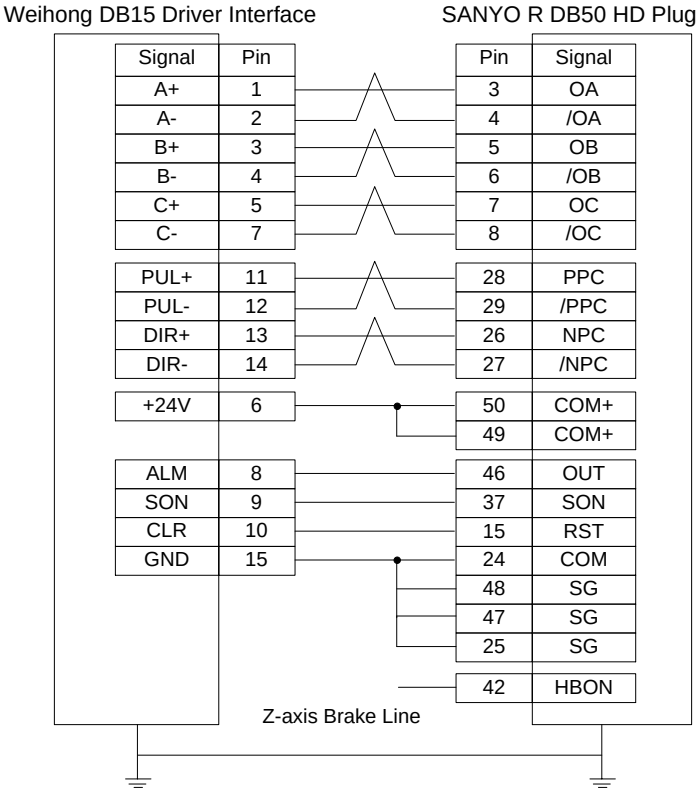
6.2.8 Wiring Diagram of HITACHI Servo Driver



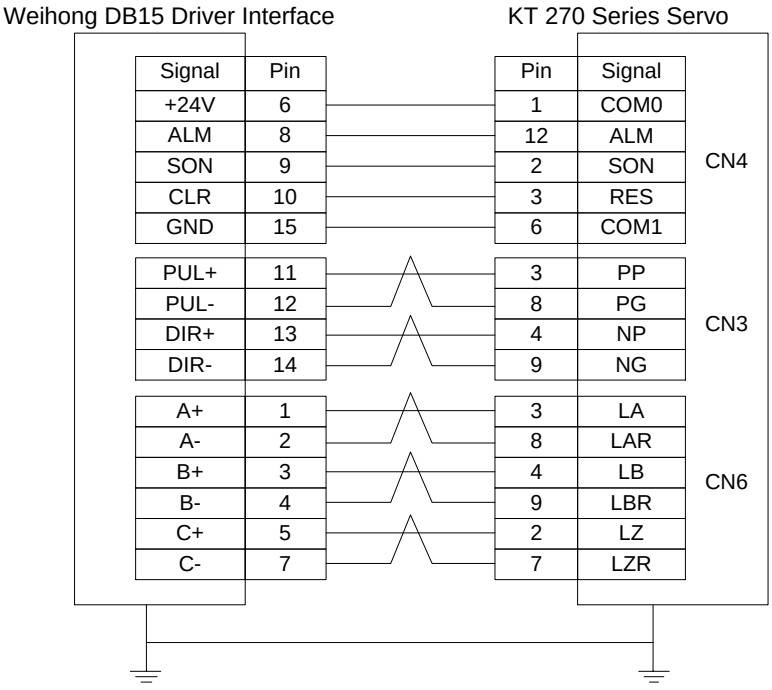
6.2.9 Wiring Diagram of SANYO PY Servo Driver



6.2.10 Wiring Diagram of SANYO R Servo Driver



6.2.11 Wiring Diagram of KT270 Servo Driver



6.2.12 Wiring Diagram of STONE GS Servo Driver

Weihong DB15 Driver Interface

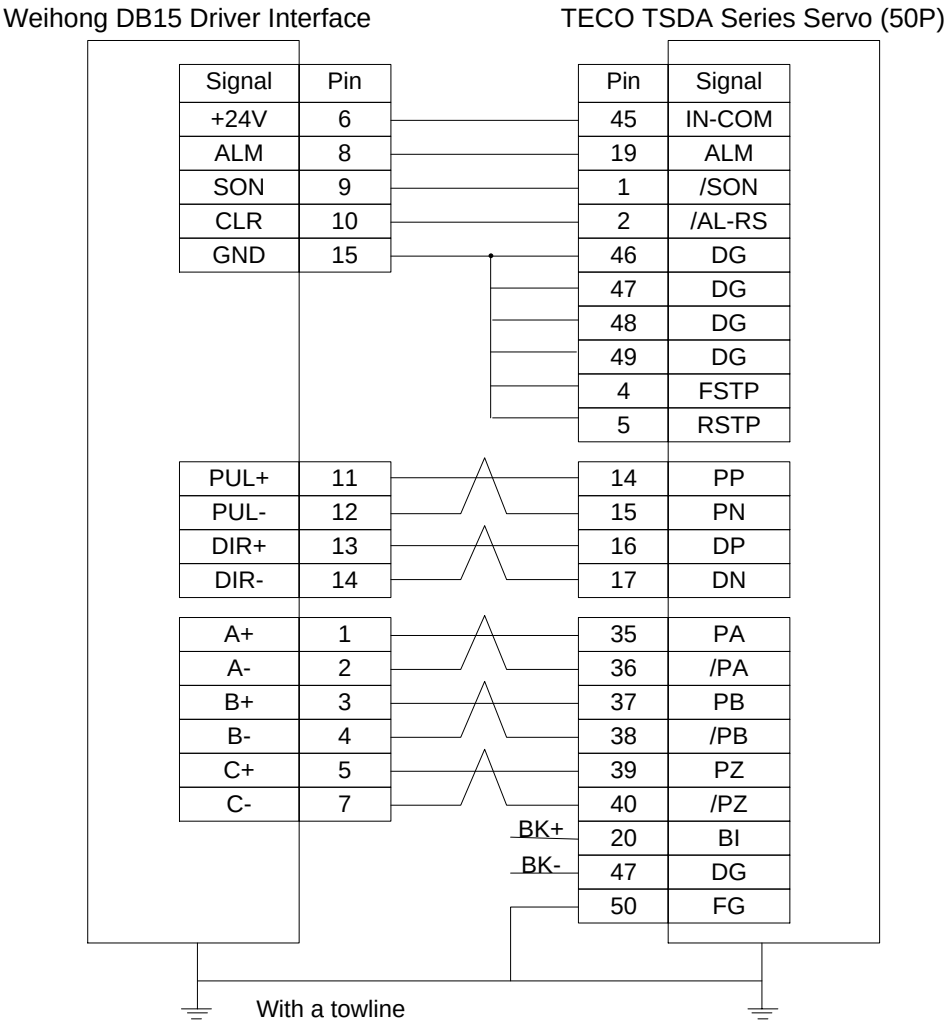
Signal	Pin
+24V	6
ALM	8
SON	9
CLR	10
GND	15
PUL+	11
PUL-	12
DIR+	13
DIR-	14
A+	1
A-	2
B+	3
B-	4
C+	5
C-	7

STONE GS Series Servo (3-line DB44 Pins)

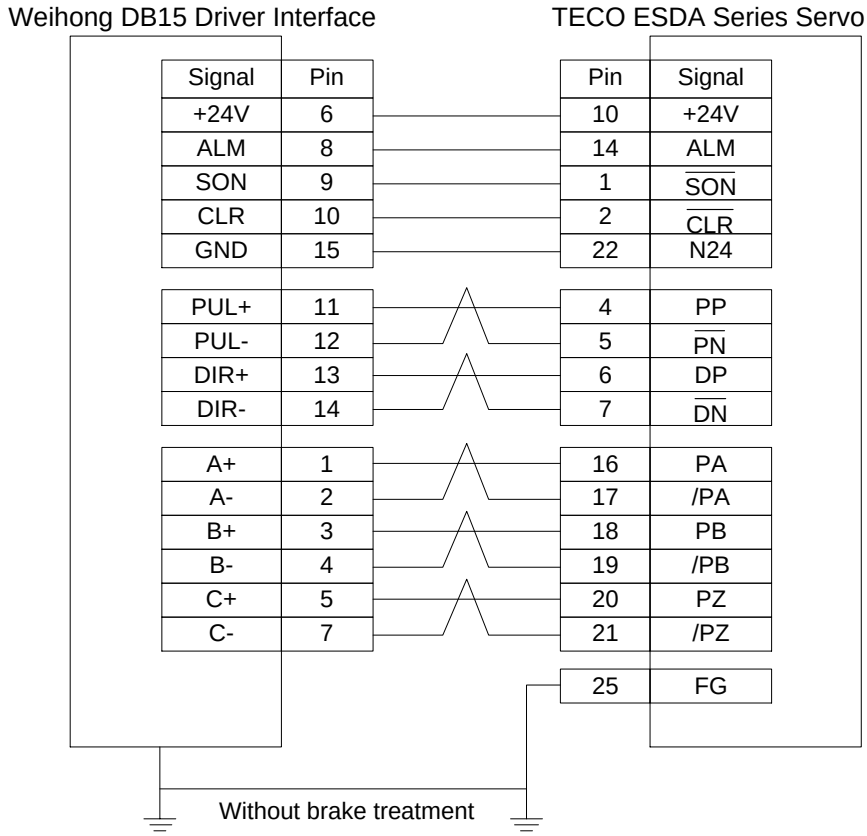
Pin	Signal
7	Input signal anode common port
22	Fault signal output+
23	Servo enable input
8	Alarm clear signal input
6	Fault signal output-
12	Pulse command signal input+
27	Pulse command signal input-
13	Direction/pulse command input +
28	Direction/pulse command input-
33	Signal differential output +
34	Signal differential output -
35	Signal differential output +
36	Signal differential output -
31	Signal differential output +
32	Signal differential output -
21	BRAKE+
5	BRAKE-

Red
Z-axis Brake Line
Black

6.2.13 Wiring Diagram of TECO TSDA Servo Driver



6.2.14 Wiring Diagram of TECO ESDA Servo Driver



7 Table of Parameters

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
1.0 Axis					
N10000	Axis Direction (X/Y/Z)	1: Positive -1: Negative	-1	Restart	3.3.1
N10010	Pulse Equivalent (X/Y/Z)	-0.0000009~999 (mm/p)	0.001	Restart	3.3.2
N10020	TravelLimits-Negative(X/Y/Z)	-99999~99999 (mm)	X: 0 Y: 0 Z: -100	Restart	3.3.3
N10030	TravelLimits-Positive (X/Y/Z)	-99999~99999 (mm)	X: 800 Y: 600 Z: 0	Restart	3.3.3
N10040	Enable Travel Limits (X/Y/Z)	YES: Valid; NO: Invalid	YES	Restart	3.3.3
N10050	Positive ToolMeas. Travel limits (X/Y/Z, or X/Y/Z1/Z2)	-99999~99999 (mm)	9999	Restart	3.9.1
N10060	Negative ToolMeas. Travel limits (X/Y/Z, or X/Y/Z1/Z2)	-99999~99999 (mm)	-9999	Restart	3.9.1
N10080	Enable ToolMeas. Travel limits	YES: Valid; NO: Invalid	NO	Restart	3.9.1
1.1 Encoders					
N11110	Axis Encoder Dir	1: Increasing encoder value; -1: Decreasing encoder value	1	Restart	3.3.3
N11130	Check Encoder Error	YES: Valid; NO: Invalid	YES	Restart	3.3.3
N11140	Static Tolerance	1~999999	500	Restart	3.3.3
N11150	Dynamic Tolerance	1~999999	500	Restart	3.3.3
N11160	Frequency Division Pulses of PG (X4)	1~999999	10000	Restart	3.3.3
N11303	Delay for Stopping On Estop	0.001~10 (s)	1	Immediate	-
	The time for a machine stops completely after E-stop.				
N11304	Encoder Feedback	YES: Valid; NO: Invalid	YES	Restart	3.3.3
N11309	Delay in Setting REF Sign	0.5~5	0.5	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
	The wait time for a machine tool stopping completely after backing to machine origin completed. The REF. Point mark will not be set until after the wait time.				
1.2 Compensation					
N12000	Screw Error Comp	0, 1, 2	1	Restart	3.14.1
N12001	Enable Backlash Compensation	YES: Valid; NO: Invalid	YES	Restart	3.14.1
N12010	Backlash	0~1000	0	Restart	3.14.1
N12020	Turn On AQE Compensation	YES: Valid; NO: Invalid	NO	Immediate	3.14.2
N12100	Time	0~10(sec)	0	Immediate	3.14.2
N12101	Distance of compensation	0~10(mm)	0	Immediate	3.14.2
	N (12101+10*n) is a group of 12 parameters with same setting. In the expression N(12101+10*n), “n” can be any number among 0, 1, 2…11.				
N12102	Delay	0~10(sec)	0	Immediate	3.14.2
	N (12102+10*n) is a group of 12 parameters with same setting. In the expression N(12102+10*n), “n” can be any number among 0, 1, 2…11.				
N12103	Intensity	0~1	0.75	Immediate	3.14.2
	N (12103+10*n) is a group of 12 parameters with same setting. In the expression N(12103+10*n), “n” can be any number among 0, 1, 2…11.				
1.3 Velo/Acc limits					
N13000	Max. Axis F(X/Y/Z)	0.001~100000 (mm/min)	48000	Immediate	3.12.1
2.0 Spindle					
N20001	Max. S	0~999999 (rpm)	24000	Restart	3.8
N20002	S	0~the maximal speed of spindle	12000	Immediate	3.8
N20005	Spindle Cool Off Delay Time	0~600 (sec)	5	Immediate	3.8
N20006	Spindle Speed when Centering	0~100000(rpm)	500	Immediate	3.8
N20010	Spindle On Delay Time	0~60(sec)	5	Immediate	3.8
N20011	Spindle Off Delay Time	0~60(sec)	5	Immediate	3.8
4.1 Lubricate					
N41000	Auto Lubricate	YES: Auto on; NO: Not auto on	NO	Immediate	-
	It sets whether the system automatically opens lubrication pump periodically and fills lube.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N41001	Lubricating Interval	1~1000000(sec)	18000	Immediate	-
	It is the time interval between two start-ups of lubrication pump.				
N41002	Lubricating Duration	1~100 (sec)	5	Immediate	-
	It is the filling time of lubrication pump each time. Default value is 5s in integral software while 10s in multi-Z software.				
4.2					
N42000	Inform Type when Cycle End	0; 1; 2	2	Immediate	-
	0: Red light not on; 1: Red light on for about 3s; 2: Red light always on until there is any input from mouse or keypad.				
N42001	Enable G28	YES; NO	YES	Immediate	-
N42002	Access Check for Modification	YES; NO	NO	Immediate	-
N42004	Machining Range Display Type	0; 1	0	Immediate	-
4.3 Coolant					
N43001	Coolant On when Task Start	YES; NO	NO	Immediate	-
N43002	Coolant Off when Task End	YES; NO	YES	Immediate	-
5.0 Controller					
N50011 ～ N50017	Enable connection with extended terminal board (Extended terminal board 1~7)	Yes; No	No: Extended TB 1; Yes: Extended TB 2~7	Restart	-
5.2 Handwheel					
N52001	Precise Pulse Counting	YES: Adopt; NO: Not adopt	NO	Restart	3.17.2
N52002	Handwheel Direction	1; -1	1	Restart	3.17.2
	1: Maintain the original motion direction of a machine tool in handwheel turning -1: Reverse the original motion direction of a machine tool in handwheel turning				
N52003	Multiple At X1	0.001~10 (mm)	0.001	Restart	3.17.2
N52004	Multiple At X10	0.001~10 (mm)	0.01	Restart	3.17.2
N52005	Multiple At X100	0.001~10 (mm)	0.1	Restart	3.17.2
N52006	HW Lead Gear (Numerator)	1~1000	1	Restart	3.17.2

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N52007	HW Lead Gear (Denominator)	1~1000	1	Restart	3.17.2
N52010	Handwheel Acceleration	1~1000 (mm/s^2)	200	Restart	3.17.2
N52012	Deceleration when Switching Axis	YES; NO	YES	Restart	3.17.1
N52013	Forbid HW Reverse Guide	YES: Ais stops when HW is turning reversele in HW Guide; NO:Axis moves normally when HW is turning reversely.	NO	Restart	3.17.2
N52030	HW Connection Mode	0: To terminal board 1: To operation panel	1	Restart	3.17.1
6.2 G code options					
N62000	Deceleration Distance	0~999 (mm)	2	Immediate	3.16.2
N62001	Approach F	0.001~99999 (mm/min)	300	Immediate	3.16.2
N62020	Enable Arc IJK Programming	YES: Valid; NO: Invalid	YES	Immediate	3.16.2
N62021	Arc Radius Tolerance	0~9999 (mm)	1	Immediate	3.16.2
N62022	Enable Tool Selection by G-code File	YES; NO	NO	Reload program	-
N62410	Enable Tool Compensation	YES: Valid; NO: Invalid	NO	Immediate	3.14.1
N62411	Tool Compensation Type	1: Normal type; 2: Intersect type; 3: Insert type	1	Immediate	3.14.1
N62412	Tool Compensation Direction	0: Null; 1: Left; 2: Right	1	Immediate	3.14.1
N62413	Interferometry Path Segments	1~5	3	Immediate	3.14.1
N62414	Enable Evade Interferometry	YES: enable NO: disable	NO	Immediate	3.14.1
N62730	G73_G83 Lifting Distance	-99999~99999 (mm)	0	Immediate	3.16.2

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N62760	G76_G87 Stop Orientation	0: G17 +X; 1: G17 –X; 2: G17 +Y; 3: G17 -Y	0	Immediate	3.16.2
6.3 Trajectory					
N63000	Look Ahead Distance for Interpolation	0~999	0.5	Immediate	-
N63002	Delay for Exact Stop	0~999 (s)	0	Immediate	3.12.1
N63006	Path Smoothing Time	0~0.064 (s)	0	Immediate	3.12.1
6.4 Velocity/Acc					
N64000	Startup F	0~600(mm/min)	0	Immediate	3.12.4
N64020	G00 F	Feed rate~Maximal speed of each axis /Maximal G00 speed supported by hardware	3000	Immediate	3.12.4
N64040	F	0~Rapid travel rate / Maximal G00 speed supported by hardware/Maximal feed rate	1200	Immediate	3.12.4
N64060	Max. F	0~100000 (mm/min)	48000	Immediate	3.12.4
N64101	Rapid Motion Axial Acceleration	0.001~100000 (mm/s^2)	800	Immediate	3.12.4
N64102	Z-axis Acceleration	0.001~100000 (mm/s^2)	800	Immediate	3.12.4
N64120	Acceleration for Corners	0.001~100000 (mm/s^2)	3800	Immediate	3.12.4
N64150	Axial Jerk	0.001~1e+011 (mm/s^3)	150000	Immediate	3.12.4
N64200	Smoothing The Path Velocity	YES: Enabled; NO: Disabled	YES	Immediate	3.12.4
N64201	MAX Angle Smooth Velocity	0~180	90	Immediate	3.12.4
	When the connection angle of two segments is larger than the value of the parameter, the system will start at startup speed, instead of smoothing the path velocity.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N64203	Path Interpolation Algorithm	1; 2; 3	3	Immediate	3.12.4
	Its setting range: 0: trapezoid algorithm 1: S-type algorithm 2: LEP algorithm 3: acceleration trapezoid algorithm.				
N64204	Acc or Dec time after Interpolation	0~99999	0.005	Immediate	-
	The longer the time is, the smoother the velocity will be. This parameter has no effect on the track precision.				
N64205	Min. F of LEP Interpolation	0~100000	60	Immediate	3.12.4
N64207	Arc Velocity Limit	YES: Enabled NO: Disabled	YES	Immediate	3.12.4
N64208	Max. F of Reference Circle	0.001~100000 (mm/min)	3600	Immediate	3.12.4
N64209	Min. F of Arc	0.001~100000 (mm/min)	180	Immediate	3.12.4
N64245	Pretreatment Path Number	1~2000	300	Immediate	3.12.4
N64249	Velocity Smooth for Single Axis	YES: enable NO: disable	YES	Restart	-
6.5 File translation					
PLT file translation					
N65000	Retract (PLT)	0~99999 (mm)	5	Reload program	3.16.4
N65001	PLT Units	0.001~99999	40	Reload program	3.16.4
N65002	Tool Offset	0.0001~99999 (mm)	0.025	Reload program	3.16.4
N65003	Cutting Depth	-99999~0 (mm)	0	Reload program	3.16.4
DXF file translation					
N65100	Retract (DXF)	0~99999 (mm)	5	Reload program	3.16.4
N65101	Cutting Depth	-99999~0 (mm)	0	Reload program	3.16.4
N65102	Layer Depth	-99999~0 (mm)	0	Reload program	3.16.4
N65103	First Point As Origin	YES: Use; NO: Not use	YES	Reload program	3.16.4
N65104	By Contour	YES: Valid; NO: Invalid	NO	Reload program	3.16.4

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N65105	Enable Bottom Cutting	YES: Valid; NO: Invalid	NO	Reload program	3.16.4
N65106	Use Metric	YES: Forcibly use; NO: Not forcibly use	NO	Reload program	3.16.4
ENG file translation					
N65200	Retract (ENG)	0~99999 (mm)	5	Reload program	3.16.4
N65201	Prompt For Tool Change	YES ; NO	YES	Reload program	3.16.4
N65203	Cutting By Tool No.	YES: Use; NO: Not use;	NO	Reload program	3.16.4
N65204	Deep Hole Cutting Type	0: Reciprocating chip removal 1: High-speed reciprocating chip removal	0	Reload program	3.16.4
N65205	Lifting Distance	0~99999	1	Reload program	3.16.4
N65206	Force To Use Tool Compensation	YES: Forcibly use; NO: Not forcibly use	YES	Reload program	3.14.1
N65207	Modify Tool No. in ENG File	YES: Enabled NO: Disabled	NO	Reload program	3.14.1
N65208	Z-axis Plunge Type	0; 1	1	Reload program	3.14.1
	The type of Z-axis downward feed at the beginning of machining an ENG file: 0: From safe height; 1: From the highest point (N10030 Table Travel Upper Limit -1)				
N65209	Lift when Change Tool	YES: Enabled NO: Disabled	YES	Reload program	3.14.1
N65210	Ignore Coordination System Instruction	YES: Enabled NO: Disabled	NO	Immediate	3.10.1
N65211	Z Lift Type after Drilling	0; 1	1	Immediate	-
	0: To R Plane 1: To specified position, exclusively of ENG file.				
N65212	Z Position after Drilling	-1000~1000mm	10	Immediate	-
	Lift Z-axis to this position when “Z Lift Type after Drilling” is set as 1.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N65213	Z Plunge Feedrate	0: Feedrate in machining 1: Feedrate in rapid traversing	0	Reload program	-
6.6 Change tool (Parameter No. may vary in multi-Z software and integral software, please note that.)					
N66005	Upper Position	-99999~99999 (mm)	0	Immediate	3.20.3
N66006	Lower Position	-99999~100000 (mm)	0	Immediate	3.20.3
N66007	Spindle Position in Tool Change X	-99999~99999 (mm)	9999	Immediate	3.20.3
N66008	Spindle Position in Tool Change Y	-99999~100000 (mm)	9999	Immediate	3.20.3
N66017	Deceleration Position X	-99999~100000 (mm)	0	Immediate	3.20.3
N66018	Deceleration Position Y	-99999~100000 (mm)	0	Immediate	3.20.3
N66028	F in Tool Changing	0~100000 (mm/min)	3000	Immediate	3.20.3
N66029	F in Moving from Upper Position to Lower Position	0~60000(mm/min)	1800	Immediate	3.20.3
N66030	Automatic Tool Measure	YES: enable Automatic tool measure NO: disable Automatic tool measure	YES	Immediate	3.20.3
N66031	Tool Magazine Type	0: Null 1: Disk Tool Magazine 2: Linear Tool Magazine	0	Restart	3.20.3
N66032	Tool Magazine Capacity	1~255	8	Immediate	3.20.3
N66036	Tool Count Port	-	NA	Immediate	-
N66037	Tool Mag. Back to Origin Port	-	NA	Immediate	4.3
N66038	Tool Mag. CW port	-	NA	Immediate	-
N66039	Tool Mag. CCW port	-	NA	Immediate	-
N66040	Tool Count CW Delay	0~5000	0	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N66041	Tool Count CCW Delay	0~5000	0	Immediate	-
N66042	Mag. CW to Origin Delay	0~5000	0	Immediate	-
N66043	Mag. CCW to Origin Delay	0~5000	0	Immediate	-
N66045	Tool Unclamp Position Signal Port	-	NA	Immediate	-
N66046	Tool Clamp Position Signal Port	-	NA	Immediate	-
N66047	External Tool Control Signal Port	-	NA	Restart	-
N66048	Output Port of Tool Unclamp/Clamp	-	NA	Restart	-
N66049	Output Port of Mag. Out	-	NA	Restart	-
N66064	Tool 1 Coordinate X (there are 21 tools in total.)	(mm)	0	Immediate	3.9
N66065	Tool 1 Coordinate Y (there are 21 tools in total)	(mm)	0	Immediate	3.9
6.7					
N67000 ～ N67002	Negative Change Tool Travel Limits(X/Y/Z)	(mm)	-10000	Restart	3.20.3
N67010 ～ N67012	Positive Change Tool Travel Limits(X/Y/Z)	(mm)	10000	Restart	3.20.3
N67020	Enable Change Tool Travel Limits(MCS)	YES: Check; NO: Not check	NO	Restart	-
7.1 Manu					
N71000	Jog F	0~N71001 (mm/min)	1200	Immediate	3.12.3
N71001	Rapid Jog F	0~N13000 (mm/min)	3000	Immediate	3.12.3
N71002	Jog Max. F Before Returning to REF Point	0 ~ “Rapid Jog Speed”	1200	Immediate	3.12.3
7.2 Auto					
N72001	Ignore Prog. F	YES: Ignore NO: Not ignore	No	Immediate	3.12.4
N72002	Ignore Prog. S	YES: Ignore NO: Not ignore	No	Immediate	-
N72003	G00 F Fixed	YES: Fix NO: Not fix	No	Immediate	3.12.4

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N72004	Spindle Off when Cycle Stop	YES: On; NO: Off	YES	Immediate	-
N72008	Spindle On when Cycle Start	YES: On; NO: Off	YES	Immediate	-
N72009	Cycle Machining Interval	0~1000	10	Immediate	3.16.2
N72010	Enable Work Coordinate Limits	YES: Enable; NO: Disable	NO	Immediate	3.10.1
N72020	Negative Work Coordinate Limits	(mm)	-99999	Immediate	3.10.1
N72030	Positive Work Coordinate Limits	(mm)	99999	Immediate	3.10.1
7.3 Pause					
N73000	F When Cycle Resume after Pause	0~100000 (mm/min)	600	Immediate	-
N73001	Lifting F on Pause	0~100000 (mm/min)	600	Immediate	-
N73002	Z-axis Lifting Pos in WCS	0; 1; 2; 3	0	Immediate	-
	0: lift to distance set in parameter; 1: lift to work coordinate set in parameter; 2: lift to Mach Coordinate set in parameter; 3: lift to fixed position set in parameter.				
N73003	Z-axis Lifting Pos in WCS	0~9999 (mm)	10	Immediate	-
N73004	Lifting Distance on Pause	0~500 (mm)	10	Immediate	-
N73005	Stop Spindle On Pause	YES: Stop; NO: Not stop	YES	Immediate	-
N73006	Z-axis Lifting Pos in MCS	-100~0 (mm)	0	Immediate	-
7.4 Return Machine Home					
N74001	Back to REF Required	YES: Required; NO: Not required	YES	Immediate	3.5.4
N74010	Machine Zero Position	0~N10030 (mm)	0	Restart	3.5.4
N74020	Coarse Positioning Dir.	1: Positive direction -1: Negative direction	X:-1 Y:-1 Z:1	Immediate	3.5.4
N74030	F in Coarse Positioning	0.001~10000 (mm/min)	1800	Immediate	3.5.4
N74040	Coarse Positioning Switch Inport Addr.	-	X: 00117 Y: 00120 Z: 00123	Immediate	3.5.4
	The input port of PLC address of coarse positioning switch of each axis.				
N74050	Fine Positioning Dir. (X/Y/Z)	1: Positive direction -1: Negative direction	X: 1 Y: 1 Z: -1	Immediate	3.5.4

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N74060	F in Fine Positioning	0.001~10000 (mm/min)	60	Immediate	3.5.4
N74070	Fine Positioning Switch Inport Addr.	-	X: 00000 Y: 00001 Z:00002	Immediate	-
	The input port PLC address of accurate positioning switch of each axis.				
N74080	Back Off Distance (X/Y/Z)	-1000~1000 (mm)	2	Immediate	3.5.4
N74090	Home Latch Count	1~100	1	Immediate	3.5.4
N74100	Lead Screw Pitch	0~100 (mm)	5	Immediate	3.5.4
N74110	Min Distance of Coarse/Fine Switches	0~screw pitch/2 (mm)	1	Immediate	3.5.4
N74120	Coarse/Fine Pos Distance Tolerance	0~100 (%)	10	Immediate	3.5.4
7.5 Tool Measurement (Parameter No. of following parameters may vary in multi-Z software and integral software, please note that.)					
N75000	Presetter Input Port Addr	00016	00016	Immediate	-
	The PLC address of the input port Tool Presetter Signal.				
N75001	F in Precise Probing	(mm/min)	60	Immediate	3.9
N75002	Precise Probing Duration	1~99999	1	Immediate	3.9
N75020	ToolMea Result Tolerance	0~10	0.1	Immediate	3.9
N75024	ToolMea Overtravel Port Addr	00124	00124	Restart	-
	The PLC address of input on I/O board, which system gets overtravel signal from the presetter.				
N75025	ToolMea Overtravel Alarm	NO: Invalid YES: Valid	YES	Immediate	3.9
N75100	Mobile Presetter Thickness	-1000~1000 (mm)	0	Immediate	-
N75203	F in Fixed Calibration	(mm/min)	300	Immediate	-
N75210	Fixed Presetter Position (X/Y/Z)	-99999~99999	0	Immediate	-
7.9 Operation others					
N79000	F Mode of Z Down Infeed	0; 1; 2	0	Immediate	-
	0: Free mode; 1: Limit the federate to “F of Z Down Infeed” when machine moves only in Z direction; 2: Limit the federate to “F of Z Down Infeed” when motion in Z direction is included.				
N79001	F of Z Down Infeed	0~100000 (mm/min)	480	Immediate	-
N79003	Safe Height	0~1000 (mm)	10	Immediate	-
N79100	Stop Mode when Cycle Completed	0: Stay where it is; 1: Tool moves to	0	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
		fixed point; 2: Tool moves to zero in WCS; 3: Return machine home.			
N79101	Run T and M3, M4, M5 Code Before Resume	YES; NO	NO	Immediate	-
	Whether system run T code and M3, M4, M5 (Spindle On/Off code) when Break-Point Resume or Advance Start.				
N79110	Fixed Point Position	-99999~99999 (mm)	0	Immediate	-
8.0 User interface					
N80002	Support Part Compensation	YES: Support; NO: Not support	NO	Restart	-
N80005	Calibration Type	0: Mobile calibration; 1: Fixed calibration; 2: First/Exchanged calibration	0	Immediate	3.9.1
N80010	Support Part Compensation	YES: Support; NO: Not support	NO	Restart	3.10.2
N80018	Show Remaining Time	YES; NO	YES	Immediate	-
	Whether to show the remaining time in Controller Information Page.				
N80020	Popup Right Ribbon	YES; NO	YES	Restart	-
	Whether to pop up the right ribbon when switching page group.				
N80021	Port Arrangement	0: Arranged by the PLC address; 1: Ports of the terminal board arranged first.	1	Restart	-
N80050	Print Info	YES; NO	YES	Immediate	-
	It will show debug information about the process of the returning to the REF (only used for machine with encoder at present).				
N80090	Use New Frp Algorithm	YES: Use; NO: not use	NO	Immediate	-
N80111	Auto Restart after Register	YES; NO	YES	Immediate	-
8.1 Position view					
N81000	Auto Load Graph	YES: Load automatically; NO: Not load automatically	NO	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N81001	Max File Size	(KB)	1000	Immediate	-
N81010	Gradient Fill	YES, NO	YES	Immediate	-
N81011	Draw Workbench	YES, NO	NO	Immediate	-
N81012	Draw Grid	YES, NO	NO	Immediate	-
N81015	Clear On Loading	YES, NO	YES	Immediate	-
N81016	Draw WC Origin	YES, NO	NO	Immediate	-
N81017	Draw MC Origin	YES, NO	NO	Immediate	-
N81018	Bkground Color 1	Select a color	0×00000000	Immediate	-
N81019	Bkground Color 2	Select a color	0×00000000	Immediate	-
N81020	G00 Color (running)	Select a color	0×0000FFFF	Immediate	-
N81021	G01 Color (running)	Select a color	0×00FFFF00	Immediate	-
N81022	G02 Color (running)	Select a color	0×00FFFF00	Immediate	-
N81023	G03 Color (running)	Select a color	0×00FFFF00	Immediate	-
N81032	G00 Color (loading)	Select a color	0×04000000	Immediate	-
N81033	G01 Color (loading)	Select a color	0×00600000	Immediate	-
N81034	G02 Color (loading)	Select a color	0×00600000	Immediate	-
N81035	G03 Color (loading)	Select a color	0×00600000	Immediate	-
N81045	Grid Color	Select a color	0×00800080	Immediate	-
N81046	Coordinate Color	Select a color	0×0000FF00	Immediate	-
N81049	WC Origin Color	Select a color	0×0000FFFF	Immediate	-
N81050	MC Origin Color	Select a color	0×0000FFFF	Immediate	-
The parameters above are shared in integral software and multi-Z axes software. Please note that the NO. and default values of some parameters are not consistent in the two version software.					
N11001	Encoder type	0: Incremental encoder; 1: Absolute encoder	0	Restart	-
	Note: when absolute type encoder is chosen, LD5E controller should be together used; otherwise, it cannot work normally.				
N11190	Adjust position at E-stop	Yes: Adjust No: Not to adjust	Yes	Restart	-
	When encoder feedback function is enabled, whether to adjust position after E-stop is canceled. Note that this function is only effective in situation where incremental type encoder is adopted. When absolute encoder is adopted, position adjustment is enabled anytime.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N11200	Motor rotate type (X/Y/Z)	1;-1	-1	Restart	-
	1: taking CW as the positive direction; -1: taking CCW as the positive direction. Note that the parameter is in need of setting only when absolute encoder is adopted.				
N11306	Double Y Static Tolerance	0.001~tolerance limit	5	Restart	-
N11307	Double Y Dynamic Tolerance	0.001~tolerance limit	5	Restart	-
N11308	Double Y Adjust Range	0.001~100	10	Immediate	
N14004	Rotary axis acceleration	0.001~1e+011	500	Restart	
N15030	Effective Radius of Rotary axis	1.0~9999.0	57.296	Restart	
N63001	Look Ahead Distance for Velocity	0~0.05	0	Immediate	-
N63003	Max Look Ahead Path Segments in LEP	1~1000	100	Immediate	-
N63007	Path Pretreatment Options	0: Null; 1: Tolerance; 2: Smoothing	0	Immediate	-
N63008	Path Pretreatment Precision	0~0.1	0	Immediate	-
N63009	Path Pretreatment Max Angle	0~180(degree)	180	Immediate	-
N63020	Rotate Axis by Sign	YES: Enable; NO: Disable	NO	Immediate	-
	In absolute dimension, rotatory axis direction is decided by sign “+”"-" in code. Please confirm the current file if you change this parameter.				
N64103	Speed Up Acceleration	0.001~100000 (mm/s^2)	800	Immediate	-
N64104	Speed Down Deceleration	0.001~100000 (mm/s^2)	800	Immediate	-
N64241	Decelerate at Max Connect Angle	YES: Enable; NO: Disable	YES	Immediate	-
N64242	Enable non-linear interpolation compensation	Yes: Enable; No: Disable	NO	Immediate	-
N64243	Mechanical structure of machine with 4 axes	0; 1	0	Immediate	-
	It refers to four axes machine mechanical structure, which is used for non-linear interpolation compensation. 0: rotary axis is parallel to X-axis; 1: rotary axis is parallel to Y-axis.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N64246	Smooth Speed for Short Lines	YES: Enable; NO: Disable	NO	Immediate	3.12.4
N64247	Reference Length of Short Lines	0.001~10	1	Immediate	-
N64248	Enable Length of Short Lines	YES: Enable; NO: Disable	NO	Immediate	-
N66000	Prompt For Tool Change	YES: Enable; NO: Disable	NO	Immediate	-
N66002	Pause in Tool Change for Same Active and Target Tool No.	YES: Pause; NO: Not pause.	NO	Immediate	-
N66033	Check ToolNo	YES; NO	YES	Immediate	-
	YES: Limit the target tool number within (0, 255); NO: No limit and keep tool number unchanged.				
N73007	Return to Fixed Point On Pause	-99999~99999	0	Immediate	-
7.8 Preheat And Wear					
N78000	Warm-up and trial-run switch	YES: Open NO: Close	NO	Immediate	3.7
N78001	Warm-up switch	YES: Open NO: Close	YES	Immediate	3.7
N78002	Trial-run switch	YES: Open NO: Close	YES	Immediate	3.7
N78100	Coolant On during warming up	YES: Turn on NO: Turn off	YES	Immediate	3.7
N78110	Warm-up startup speed	0~the maximal warm-up startup speed	0	Immediate	3.7
N78111	Warm-up max.speed	0~24000	0	Immediate	3.7
N78112	Spindle speed increment	0~24000	0	Immediate	3.7
N78113	Spindle speed increase interval.	1~60	1	Immediate	3.7
N78200	Lubrication On during trial run.	YES: Turn on NO: Turn off	NO	Immediate	3.7
N78210 ~ N72812	Trial-run end (X/Y/Z)	-99999~100000	X/Y: 100 Z: -20	Immediate	3.7
N78220	Trial-run times	1~1000	1	Immediate	3.7
N78221	Trial-run speed	0~100000	1500	Immediate	
Please note that the parameters above are unique in integral software.					

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N41006	Lube pump detection start time	-	0	Immediate	-
N41007	Lube pump detection end time	-	-	Immediate	-
N41010	Auto Draining	YES; NO	NO	Immediate	-
N41011	Auto Draining Interval	3.6~3600000	1800	Immediate	-
N41012	Auto Draining Duration	1~100	10	Immediate	-
N62090	Exact Stop Tolerance	0~99	0.001	Immediate	-
N63001	Look Ahead Distance for Velocity	0~0.1	0	Immediate	-
N64100	Axial Acceleration	0.001~100000	800	Immediate	-
N64244	Optimize Performance	YES; NO	YES	Immediate	-
N67003	Change tool overtravel limit-Negative Z2	(mm)	-10000	Restart	-
N67013	Change tool overtravel limit-Positive Z2	(mm)	10000	Restart	-
N74000	Cancel REF. at reset	Yes: Cancel No: Not cancel	Yes	Immediate	-
	Once reset operation is enabled during machining, sign of returned machine origin will be cleared.				
N74002	Cancel REF. sign at E-stop	Yes: Cancel No: Not cancel	Yes	Immediate	-
	With encoder feedback function is enabled, sign of returned machine origin will not be cleared at emergency stop occurrence if the parameter is set to “No”.				
N75026	Calibration Procedure	0: With multiple presetters, all Z axes are calibrated at the same time; 1: With only one presetter for all Z axes, calibrate in turn.	1	Restart	-
N75400	Auto Leveling Z Axes	YES; NO	YES	Immediate	-
N79401	Z1 Pos when Change Spindle	-100~0mm	0	Immediate	-
N79402	Z2 Pos when Change Spindle	-100~0mm	0	Immediate	-
N79403	Switch to Z1-axis when task ends	-100~0mm	NO	Immediate	-
N79404	Z1Z2 Spacing Offset X	-9999~9999mm	0	Restart	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N79405	Z1Z2 Spacing Offset Y	-9999~9999mm	0	Restart	-
Please note that the parameters above are unique in multi-Z axes software.					

8 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;
3. 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.