NK300 Integrated CNC System

Manufacturers' Manual

20th Edition

Weihong Electronic Technology Co., Ltd.

The information contained in this manual is constantly being updated. You can login to 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 operators / manufacturers. If you use Weihong CNC system for the first time, it is suggested that you read through this manual. If not, however, you can search for the desired information via the Contents.

With ten chapters, this manual can be divided into 5 parts, as follows.

1) Part 1: the preface, introducing the precautions about transportation and storage, installation, wiring, debugging, usage and so on. You need to read them carefully before using our products to ensure safe operations.
2) Part 2: the hardware part, including chapter 1, 2 and 8. Chapter 1 and 2 introduces the configuration, hardware and wiring of NK300 integrated CNC system. Chapter 8 introduces the parameter setting and wiring diagrams of different drivers.
3) Part 3: the software operation part, including chapter 3, 4, 5, and 6. This part illustrates detailed operations and interfaces of single functionality and the related parameters, which will be an intuitional guidance to users and operators in real practice.
4) Part 4: the maintenance part, consisting of chapter 7. In this chapter, possible problems and their countermeasures are listed, aiming to help you respond instantly and take effective measures when possible failure occurs.
5) Part 5: the appendix, consisting of chapter 9 and 10, corresponding to table of parameters as well as the software license agreement.

Applicable Product Model

This manual is applicable to NK300 Integrated CNC System. Please refer to the table below for details.

Product Model	Remarks
NK300 Integrated CNC System	The system is abbreviated as NK300 and classified as NK300A series and NK300B series.

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 201400

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 the current edition.

Date	Edition	Revision
2016.02	R20	Contact information updated
2016.1	R19	Main revision is as follows. 1) Dialog box of manufacturer's access verification and confirmation is added for display of toolbox parameters, "Set ToolLib XY" in dialog box "fixed calibration", and modifications of backlash on X/Y/Z axis in interface **Screw Comp**. 2) Forbid HW reverse guide function added. 3) Program file function added. 4) Upgrade common files function added. 5) The interface and related parameters of punching and tapping software added. 6) Parameters updated.

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.

CAUTION: 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

WARNING

cabinet;

- Space between this device and other equipments 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;
- 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 misoperation 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

WARNING

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

NK300 is divided into NK300A and NK300B according to the layout of monitor and operation panel. The product pictures of the two types are as shown in Fig. 1-1 and Fig. 1-2.

Fig. 1-1 Product Picture of NK300A

NK300A integrated CNC system consists of following components:

- One NK300A host
- One WH106A2 operation panel
- One EX9A4 terminal board
- One DB9M/F cable (5m)
- One three-core power line (1.5m)
- Servo cables (brand, length and number depends on your needs)
- One NK-MPG-06 Handwheel (optional)

Fig. 1-2 Product Picture of NK300B

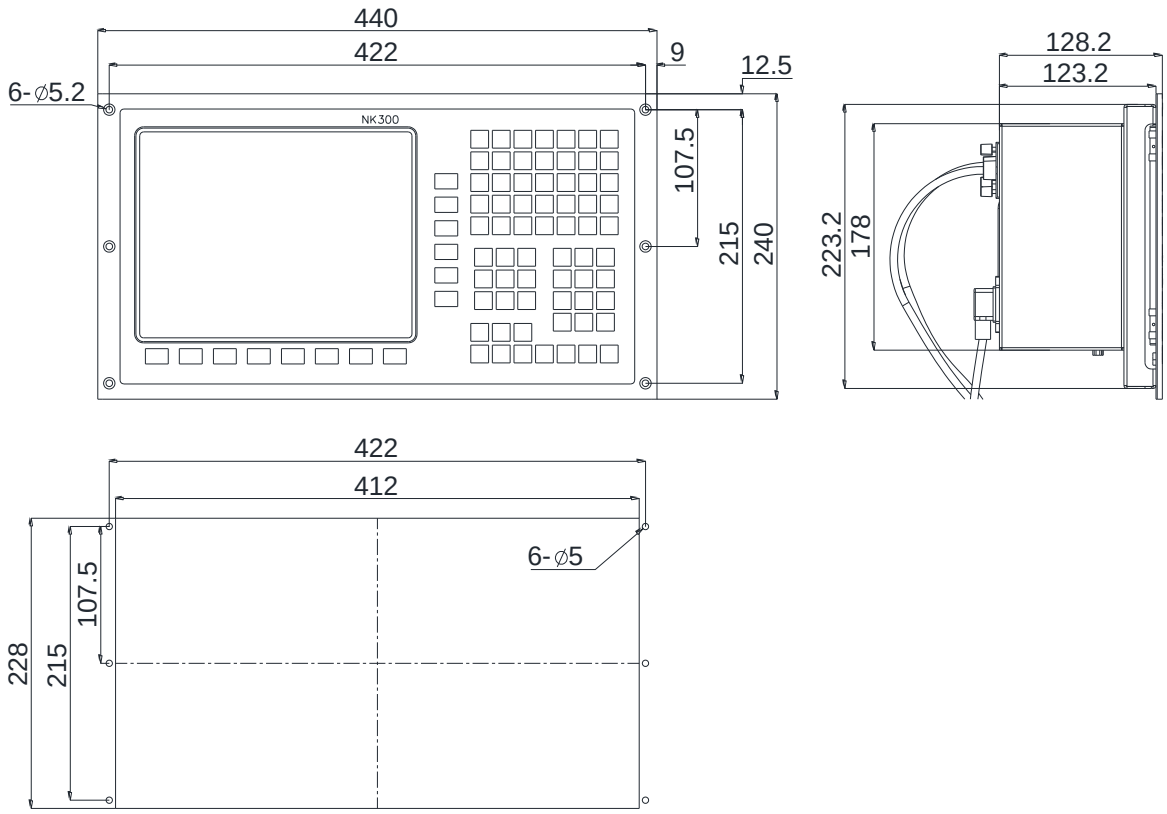
NK300B integrated CNC system consists of following components:

- One NK300B host
- One WH107A2 operation panel
- One EX9A4 terminal board
- One DB9M/F cable (5m)
- One three-core power line (1.5m)
- Servo cable (brand, length and number depends on your needs)
- One NK-MPG-06 Handwheel (optional)

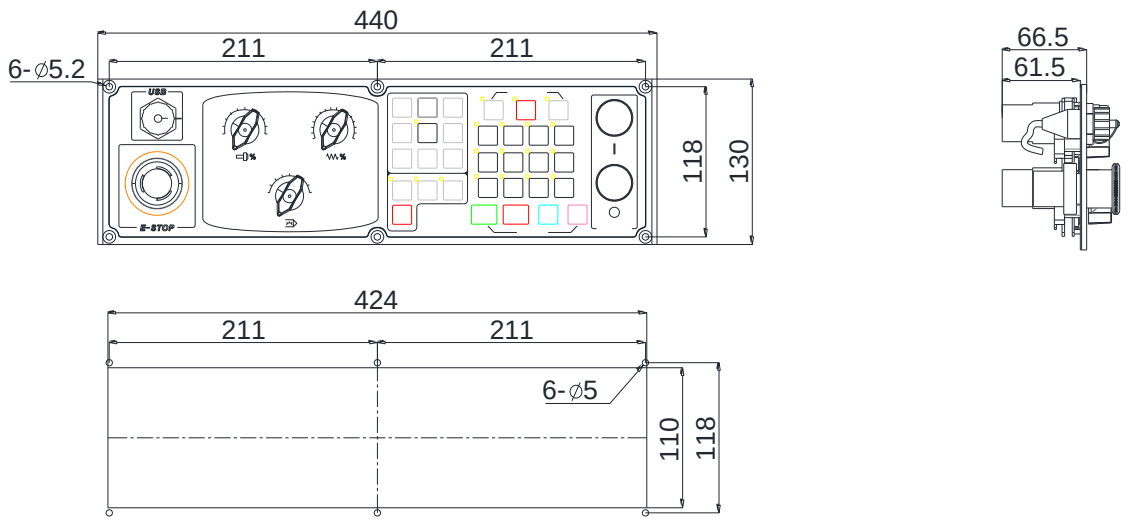
1.1.2 Mounting Dimensions

After NK300 is installed on the machine, 100 mm space should be preserved in its surrounding for wiring convenience and ensuring ventilation in the cabinet. The thickness of NK300 display panel is 128.2mm, please preserve 25mm~30mm space for the plug.

- **The mounting dimensional drawing of NK300A**



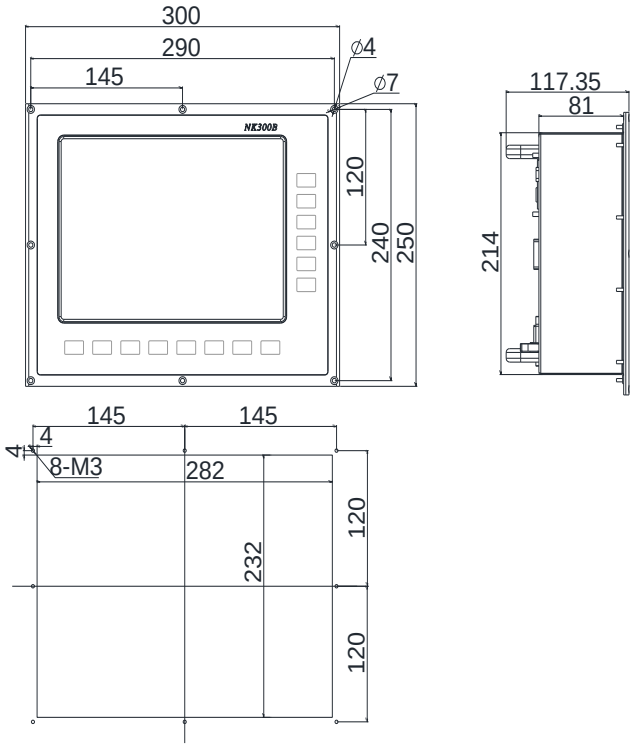
Dimension and Cutout of Host NK300A



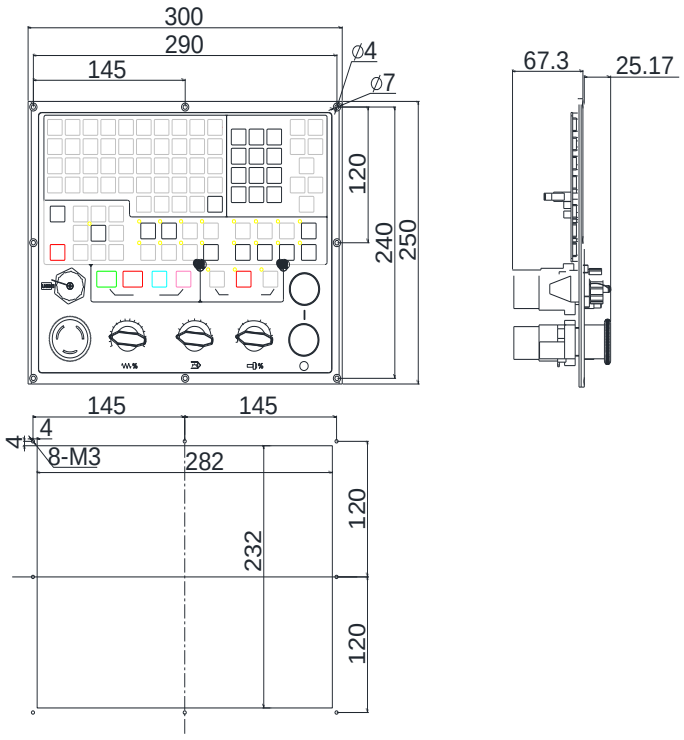
Dimension and Cutout of Operation Panel WH106A2

Fig. 1-3 Mounting Dimension Drawing of NK300A

- **The mounting dimensional drawing of NK300B**



Dimension and Cutout of NK300B



Dimension and Cutout of Operation Panel WH107A2

Fig. 1-4 Mounting Dimension Drawing of NK300B

1.1.3 Host

From appearance, NK300 series host consists of three parts, i.e. display area, keyboard area and CNC key area. Keyboard area is a collection of simulation keys, which is more convenient than USB keyboard. CNC key area is a collection of control keys for the machine tool. The main difference between NK300A and NK300B is the position of keyboard area. The keyboard area of NK300A is in the host and only CNC keys are designed in the respective operation panel WH106A2. However, only display area exists in NK300B host, and keyboard area and CNC key area exist in the respective operation panel WH107A2. Below is an introduction of NK300A. Please note the differences and similarities between NK300A and NK300B.

- **Front View of NK300A Host**

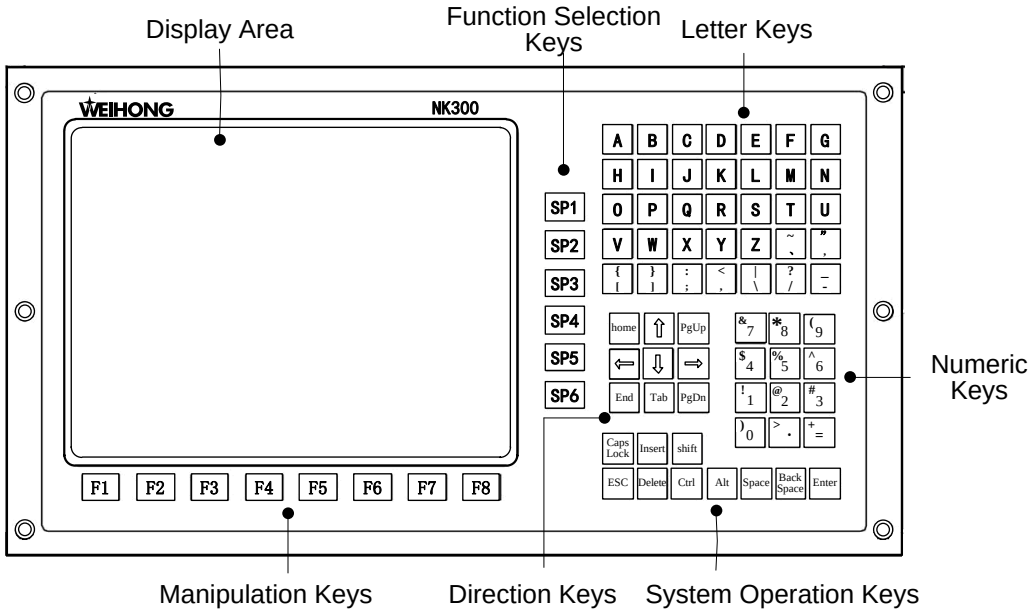


Fig. 1-5 Front View of NK300A Host

1) Display area shows user interface. It is user-friendly and convenient.
2) Function selection keys include key SP1~SP6, which respectively represents function area **General**, **Advance**, **Program**, **System**, **Parameter** and **Diagnosis**. You can access the functional areas by pressing the relevant keys.
3) Letter keys include 26 letter keys and some punctuation keys.
4) Numeric keys are used for entering numbers and symbols. By press the numeric keys directly you can enter the numbers. And by pressing shift down and then press the numeric keys, you can enter the symbols on the upper left of the keys.
5) System operation keys include Caps Lock, Backspace, Space, Enter, Delete and other keys.
6) Direction keys are for moving the cursor. The area includes direction keys, up, down, left and right, Home, End, PgUp and PgDn and Tab.
7) Manipulation keys include F1~F8, which are used to activate the functions indicated in software interfaces.

- **Rear View of NK300A Host**

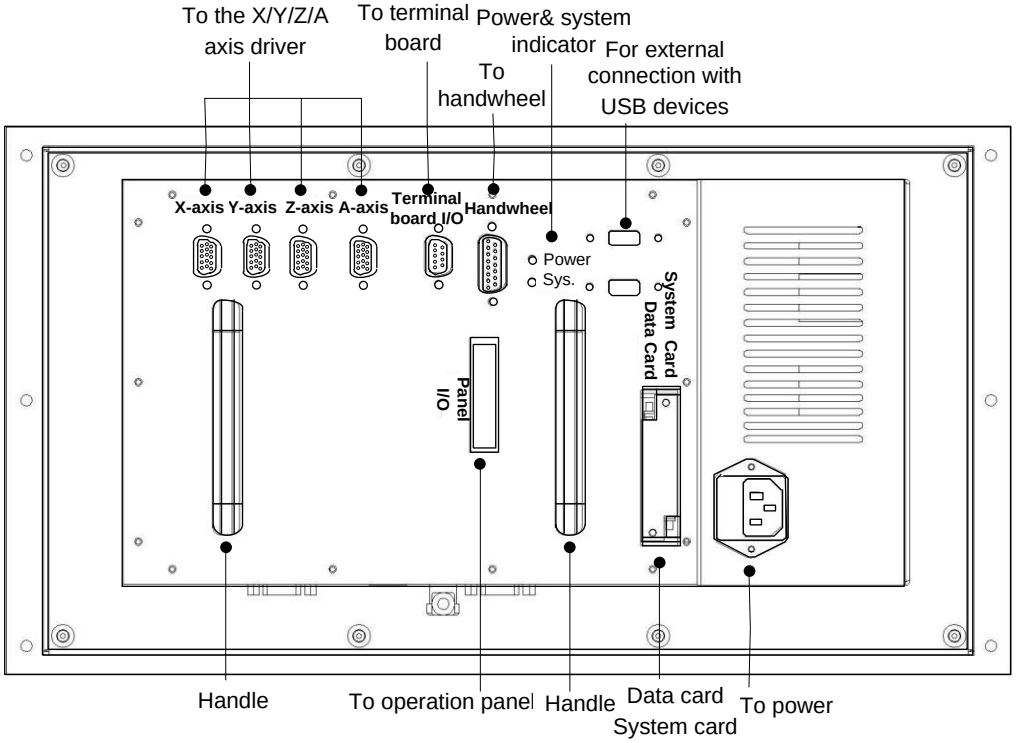


Fig. 1-6 Rear View of NK300A Host

1.1.4 Operation Panel

The operation panel for NK300A is WH106A2.

- **Front view of operation panel WH106A2**

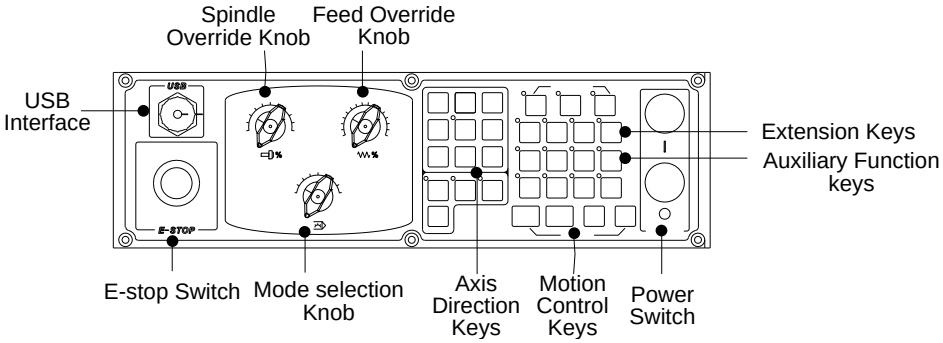


Fig. 1-7 Front View of Operation Panel WH106A2

(1) USB interface with a protection cap is used for connection with removable flash disk.

(2) E-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 3.5 for details.

(4) Feed override knob is used for feedrate override adjustment. Refer to chapter 3.10.1 for details.

(5) Mode selection knob is as shown in Fig. 1-8. You can select from auto mode, jog mode, handle mode and REF. point modes according to your actual needs.

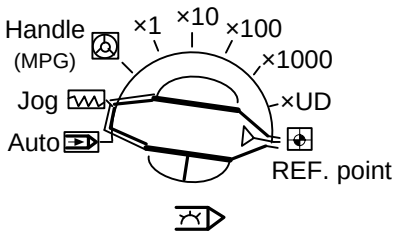


Fig. 1-8 Mode Selection Knob

(6) Extension keys include K1~K4, used for user-defined functions.

(7) Auxiliary function keys are used to control the auxiliary functions of the machine tool. Please refer to Fig. 1-9 for details.

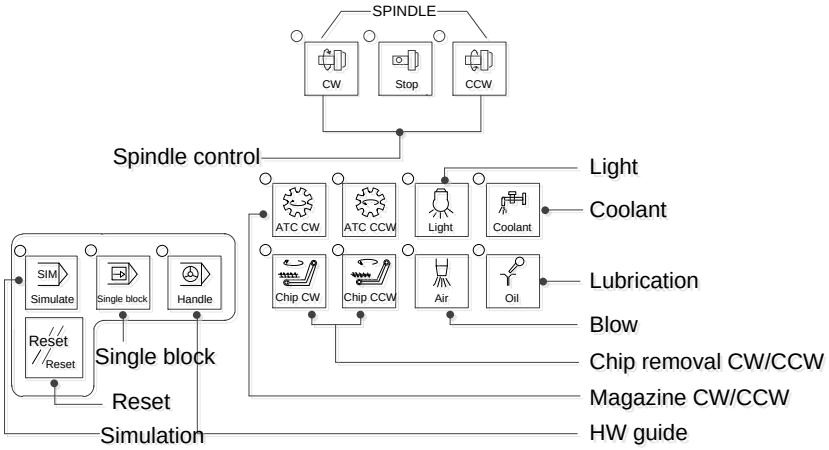


Fig. 1-9 Auxiliary Function Keys

(8) Axis direction keys are used for manual control of each axis movement in jog mode or jiggle mode. See Fig. 1-10 for concrete keys. How to use key **Rapid**: In manual mode, when any axis direction key and key **Rapid** are pressed at the same time, the axis moves at manual high speed. When any axis direction key is pressed alone, the axis moves at jog speed.

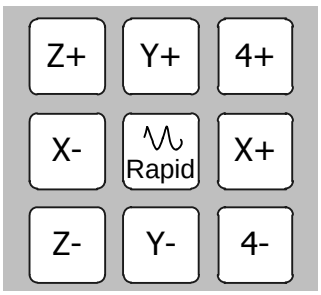


Fig. 1-10 Axis Direction Keys

(9) Motion control keys. In auto mode, you can press key **Start**, **Pause** and **Stop** to make the machine tool conduct corresponding operations. When power interruption or emergency stop occurs, you can press key **Resume** to resume machining from the interrupted point to save time on condition

that the workpiece origin is accurate for sure.

(10) Power ON/OFF switch is used to turn on/off the power supply.

- **Rear view of operation panel WH106A2**

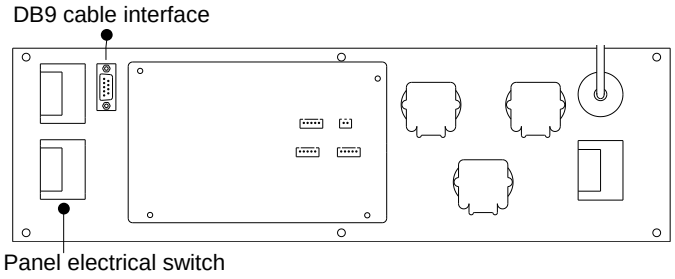


Fig. 1-11 Rear view of Operation Panel WH106A2

(1) DB9 cable interface. Connect the operation panel to the host with a DB9M/F cable (5m).

(2) Panel electrical switch is used to control the power supply of operation panel. Refer to section 2.2 for details.

1.1.5 Overall Connection Diagram

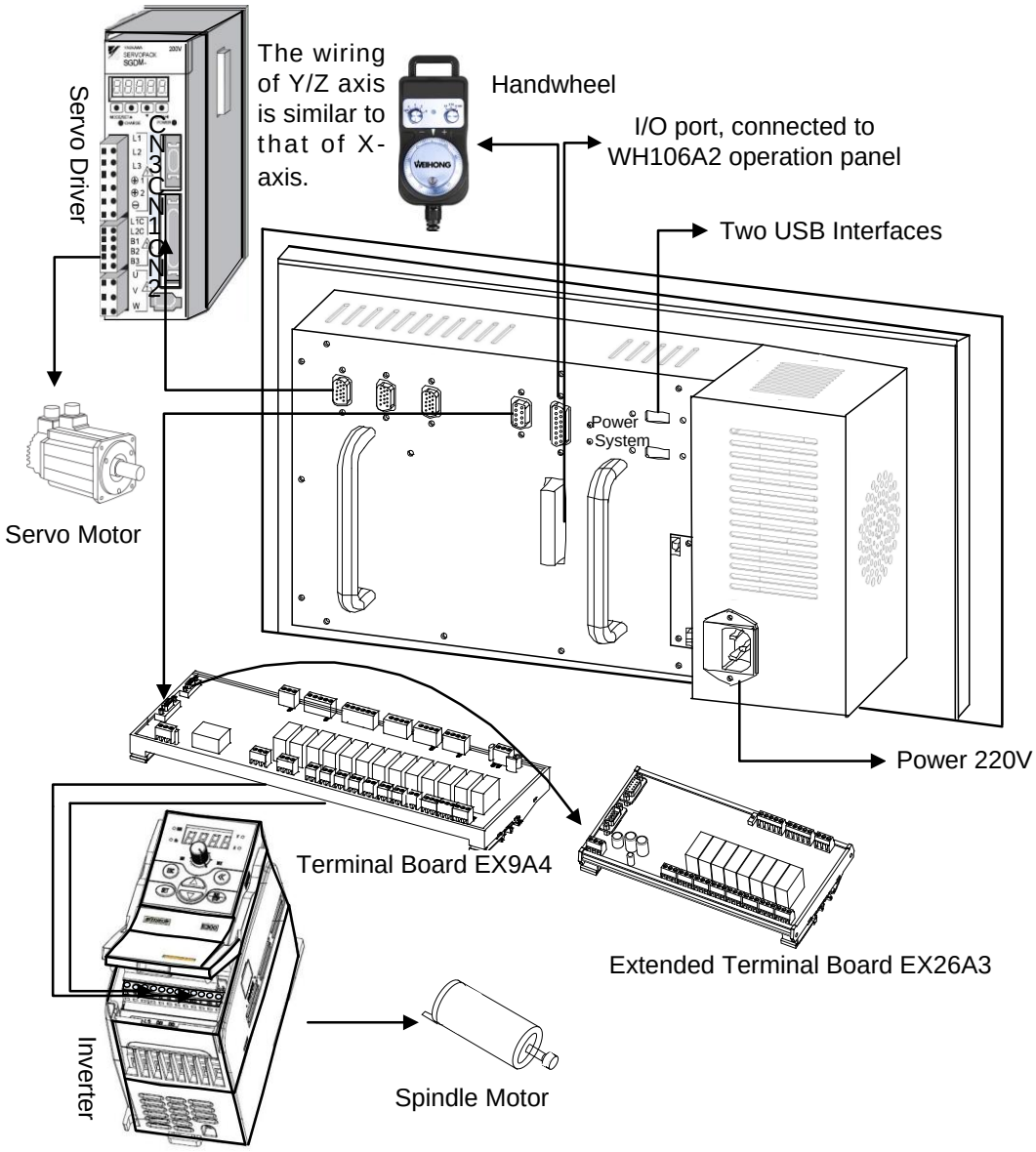


Fig. 1-12 Overall Connection Diagram of NK300A Integrated CNC System

CAUTION

In the schematic diagram above, the host machine is NK300A. For the wiring of NK300B, the I/O port is connected to operation panel WH107A2.

1.2 Software

NK300 integrated CNC system can carry software with different configuration, including three-axis configuration, four-axis configuration, double Y axis configuration and double Z axis configuration. Unless otherwise specified, this manual is based on NcStudio in three-axis standard configuration.

Software user interface is composed of 6 functional areas, which can be switched by 6 functional keys on the right side of the host. The layout of main interface in auto mode is shown as Fig. 1-13. Please refer to chapter 3 for detailed introduction to operations for each function.

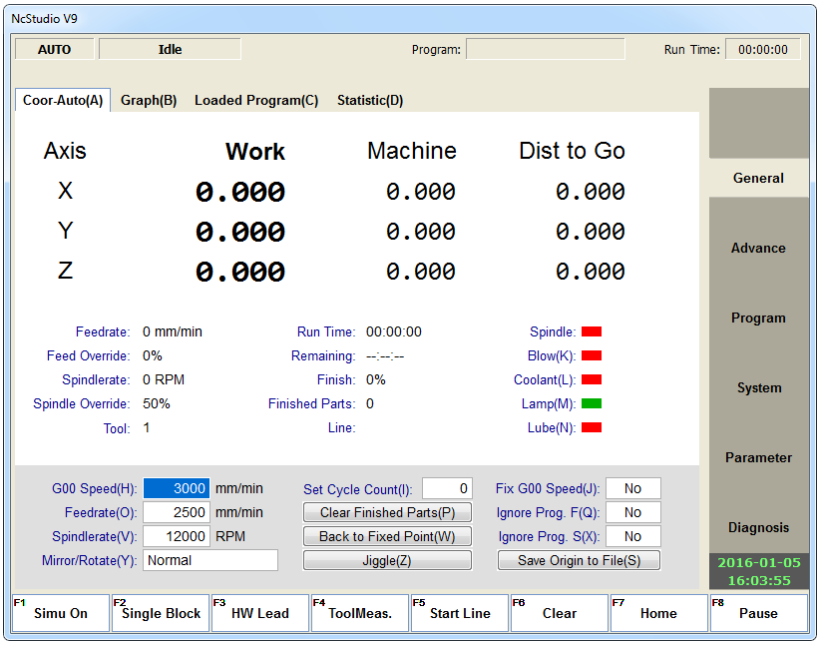


Fig. 1-13 Software Interface under Three-axis Configuration

- Functional area **General**

Interface **Coor**, **Graph**, **Loaded Program**, and **Statistic** are included in this area, where you can set common parameters, conduct common 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 information.

- Functional area **Advance**

Interface **Coor System**, **Centering**, **MDI**, **Coor Backup** and **Partcompensation** are included in this area, where you can set workpiece offset, public offset and part compensation, execute centering, edit and execute MDI, set and save coordinates.

- Functional area **Program**

Interface **Local Disk**, **Removable Disk**, **Program Wizard**, and **History Record** are included in this area, where you can manage program files, including loading, editing, deleting, arraying, unloading, new and renaming. In addition, you can load file into the system or track the history.

- Functional area **System**

Interface **System Info**, **Configuration**, **Network Info** and **Language** are included in this area, where

you can access functions related to registration, maintenance, change configurations, network setting and language setting.

- Functional area **Parameter**

Interface **General**, **Axis**, **ToolBox**, **Screw Comp** and **Auto Backup**, are included in this area.

- Functional area **Diagnosis**

Interface **Alarm List**, **Log**, **I/O Port** and **Diagnosis**, are included in this area where you can check alarms, warning, logs, ports, feedback pulses and coordinates, etc.

2 Wiring

2.1 Basic Concepts of Signal

2.1.1 Signal Types

The signal types of NK300 system can be divided into following 4 types: binary input signal, relay output signal, differential output signal and analog signal.

- **Binary Input Signal**

Binary input signal is active low and supports NO and NC input signals (through modifying input port polarity in software). Conducting to GND (i.e. grounding signal) in NO connection means signal detected, and disconnecting with GND in NC connection means signal detected.

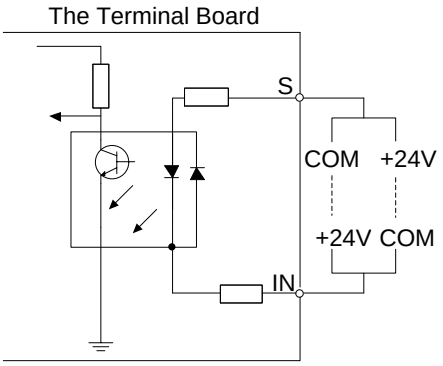


Fig. 2-1 Connection of Binary Input and Mechanical Switch

CAUTION

NK300 system also supports active high, please choose the related terminal board EX6A2 according yo your needs. Conducting to 24V in NO connection , or disconnecting with 24V in NC connection means signal detected.

- **Relay Output Signal**

The relay output contact points on the terminal board have load capacity: 10A/250VAC and 10A/30VDC. It can control 220V AC load. You can use a contractor if high power load is needed. Please see Fig. 2-2.

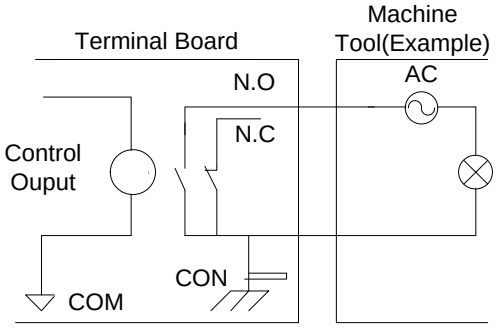


Fig. 2-2 Connection of Relay Output and Contactor

- **Differential Output Signal**

Differential signal refers to two equivalent signals with opposite phases sent by driving end, and the voltage difference of these two signals is used for deciding whether the logical status of differential signal is “0” or “1”.

Pulse command format of controlling driver motion is pulse + direction, negative logic. And this signal adopts differential signal transmission mode.

See Fig. 2-3 for pulse mode.

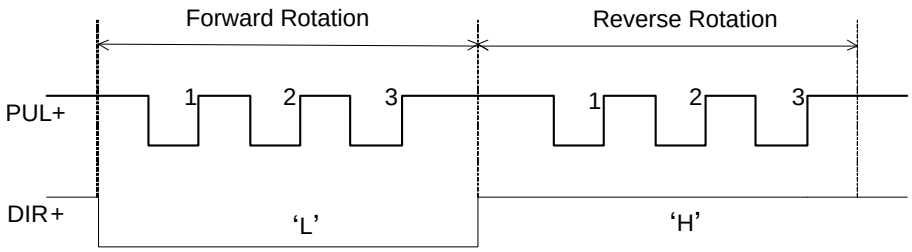


Fig. 2-3 Pulse Command Output Mode

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

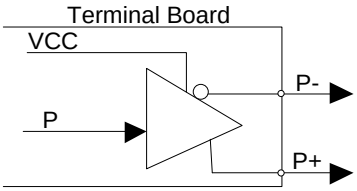


Fig. 2-4 Pulse Command Output Circuit

- **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.

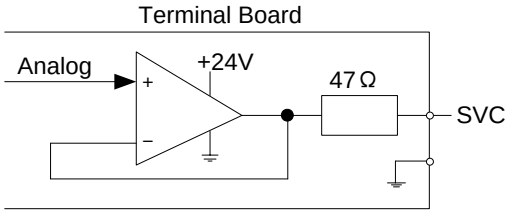


Fig. 2-5 Analog Output Signal Circuit

2.1.2 Binary Input

- **Connection of Binary Input and External Circuit**

The wiring method between binary input signal and mechanical switch is shown in Fig. 2-6:

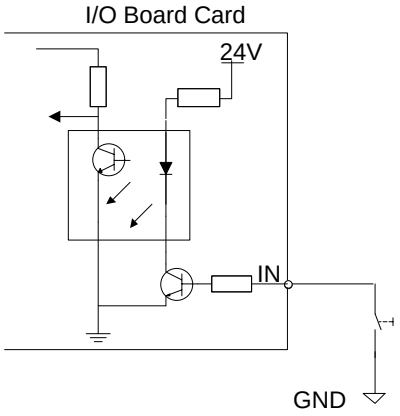


Fig. 2-6 Connection of Mechanical Switch and Binary Input

Binary input signal can be connected with photoelectric switch or proximity switch of NPN (NO or NC) type. Its joining method is as shown in Fig. 2-7. And you can use switch of PNP type by simply adopting the related terminal board EX9A4.

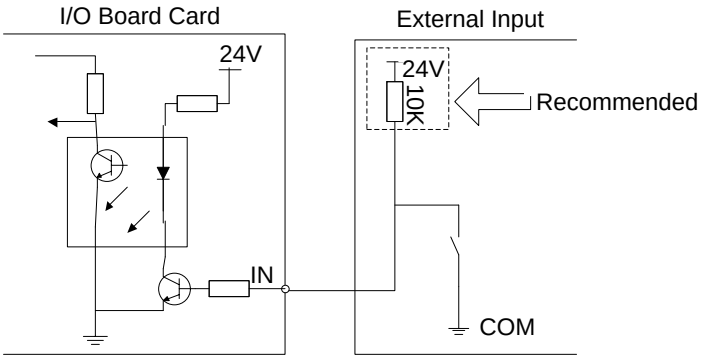


Fig. 2-7 Binary Input of NPN Type Connecting with Photoelectric Switch or Proximity Switch

- **Power Requirement**

It is recommended to adopt DC24V/4.5A switch power for the relays on the terminal board. If there are a great many DC24V relays controlled by binary output signal, users can appropriately expand the power source capacity or add extra power (forcibly sharing ground with external power supply). Z-axis brake and solenoid valve also need DC24V instead of external power to the greatest extent to reduce the interference to CNC device from solenoid valve, etc.

2.1.3 Binary Output

- **Signal Signature**

The internal equivalent circuit of binary output signal is shown in Fig. 2-8.

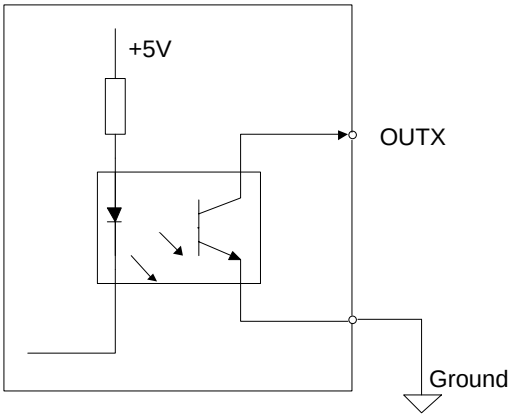


Fig. 2-8 Equivalent circuit of binary output interface

- **Technical Parameter**

1) Supply voltage: 24VDC

2) Open-collector binary output

OC (open-collector) outputs drive capability with maximum allowable operating voltage 30VDC and maximum allowable current 20mADC; so when the output terminal is active low, the maximum allowable sucked current is 20mA.

- **Connection of Binary Output and External Circuit**

The connection of solid-state relay and binary output is shown in Fig. 2-9.

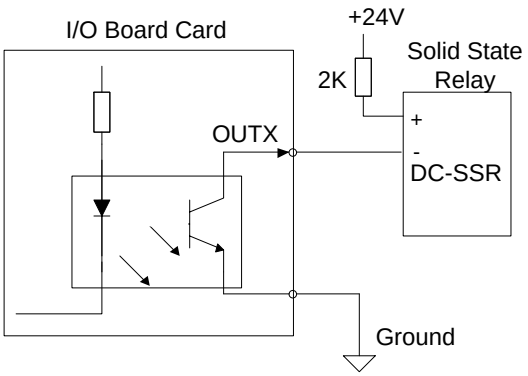


Fig. 2-9 Connection of Solid-state Relay and Binary Output

The connection of binary output and optical coupler is shown in Fig. 2-10.

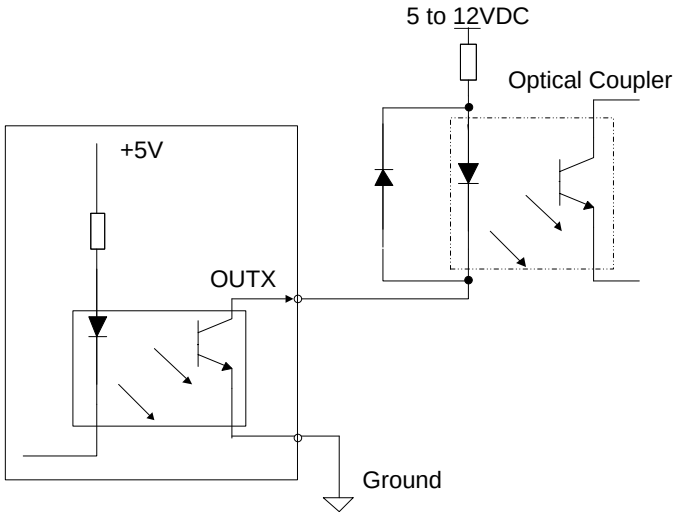


Fig. 2-10 Connection of binary output and optical coupler

CAUTION

The max. allowable voltage of optical coupling open collector output is: 30VDC, with max. allowable current 50mA.

2.1.4 Output Analog

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

2.2 Electrical Switch Wiring Diagram of Control Panel

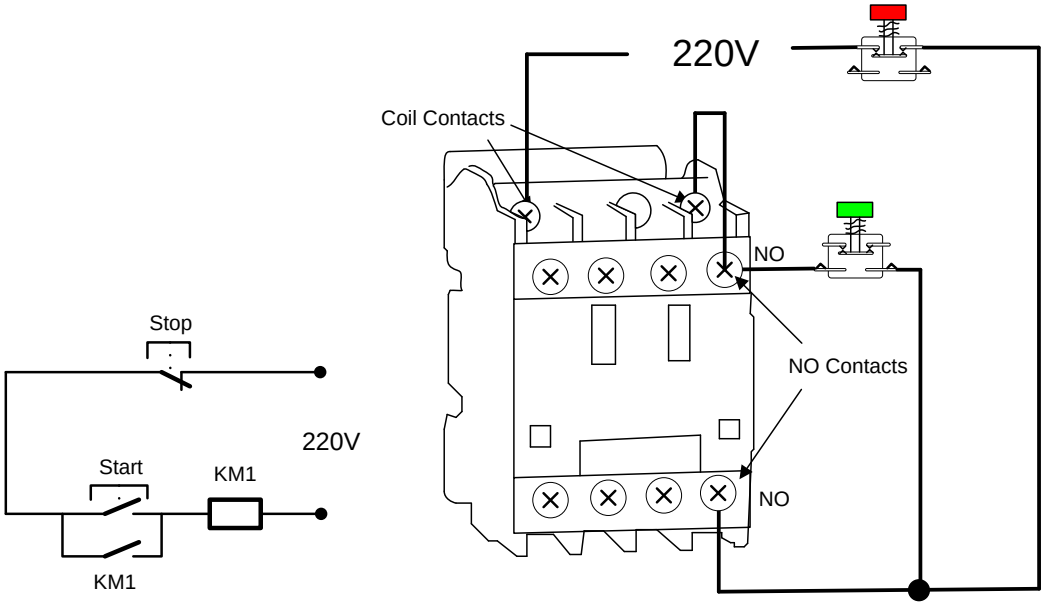


Fig. 2-11 Wiring Diagram of Electrical Switch on Control Panel

2.3 Wiring Specification of Terminal Board

NK300 integrated CNC system in three-axis, four-axis and double Y axis configuration is equipped with terminal board EX9A4, while the system in double Z configuration is equipped with terminal board EX24A1. The I/O ports of the system can be expanded with no more than 5 EX26A3 terminal boards, which can support 10 input ports and 8 output ports.

The LED indicator near each input on the terminal board EX9A4 is used to tell whether the wiring of this port is correct in machine debugging. The concrete method is: press the external switch to give the input signal, if the LED turns on, it means correct connection; if not, check whether the input is wrongly connected. To test whether the port is damaged, open the software for test. For details, please refer to section 3.6.

2.3.1 Wiring Diagrams of Terminal Board

- **Three-axis Standard Configuration**

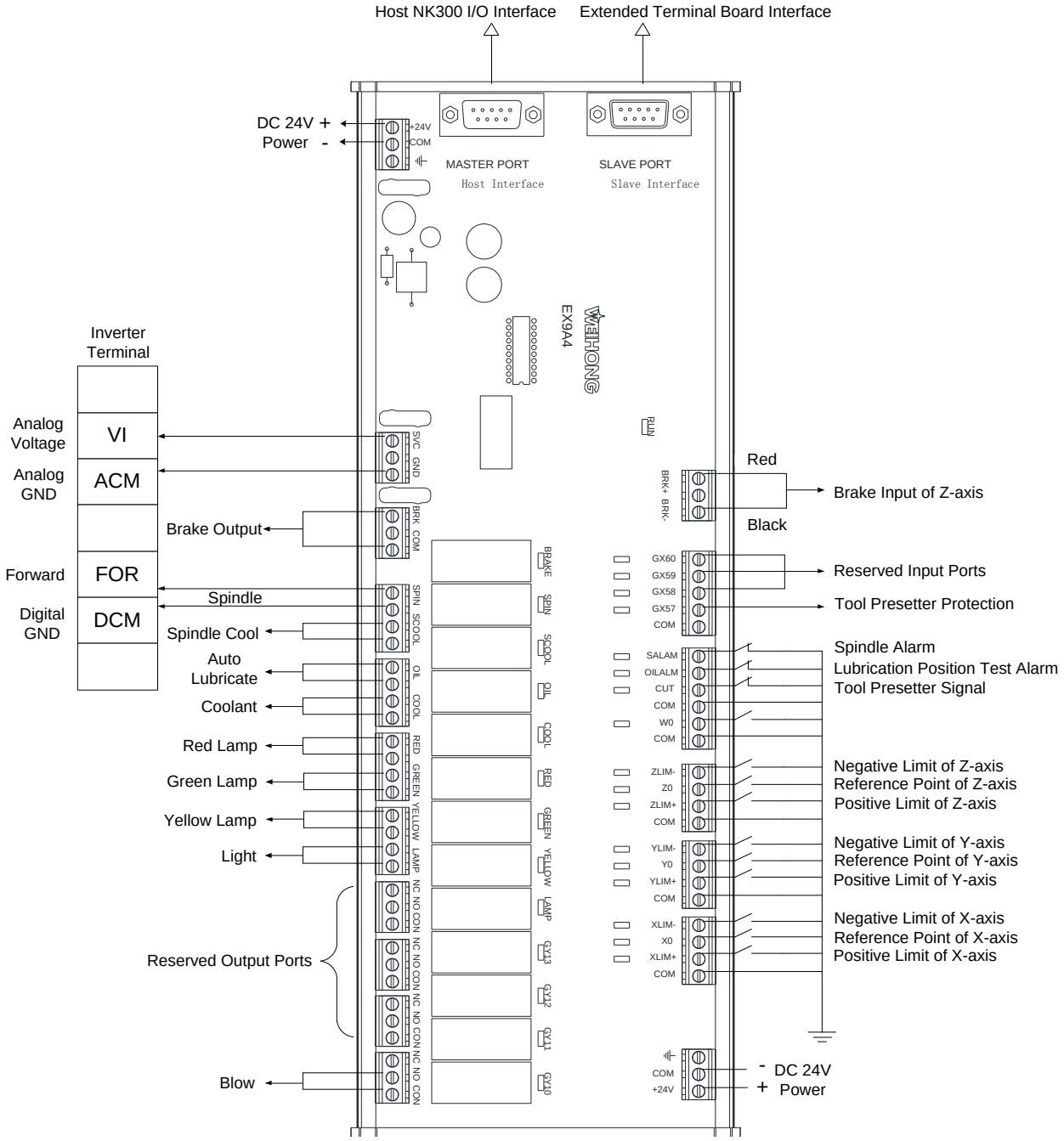


Fig. 2-12 Wiring Diagram of Terminal Board EX9A4 in Three-axis Standard Configuration

- **Four-axis Standard Configuration**

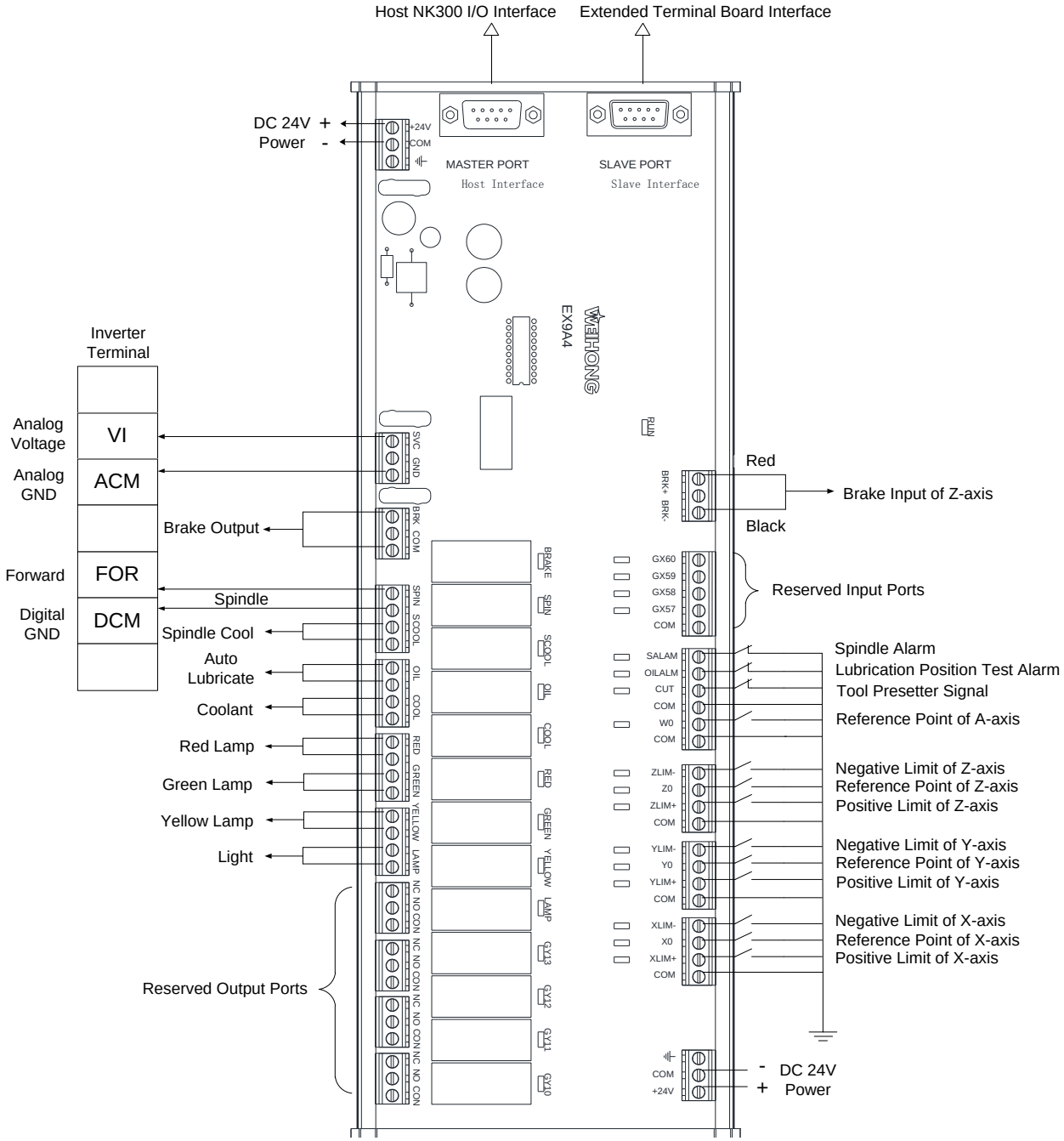


Fig. 2-13 Wiring Diagram of Terminal Board EX9A4 in Four-axis Standard Configuration

- **Double Y Axis Configuration**

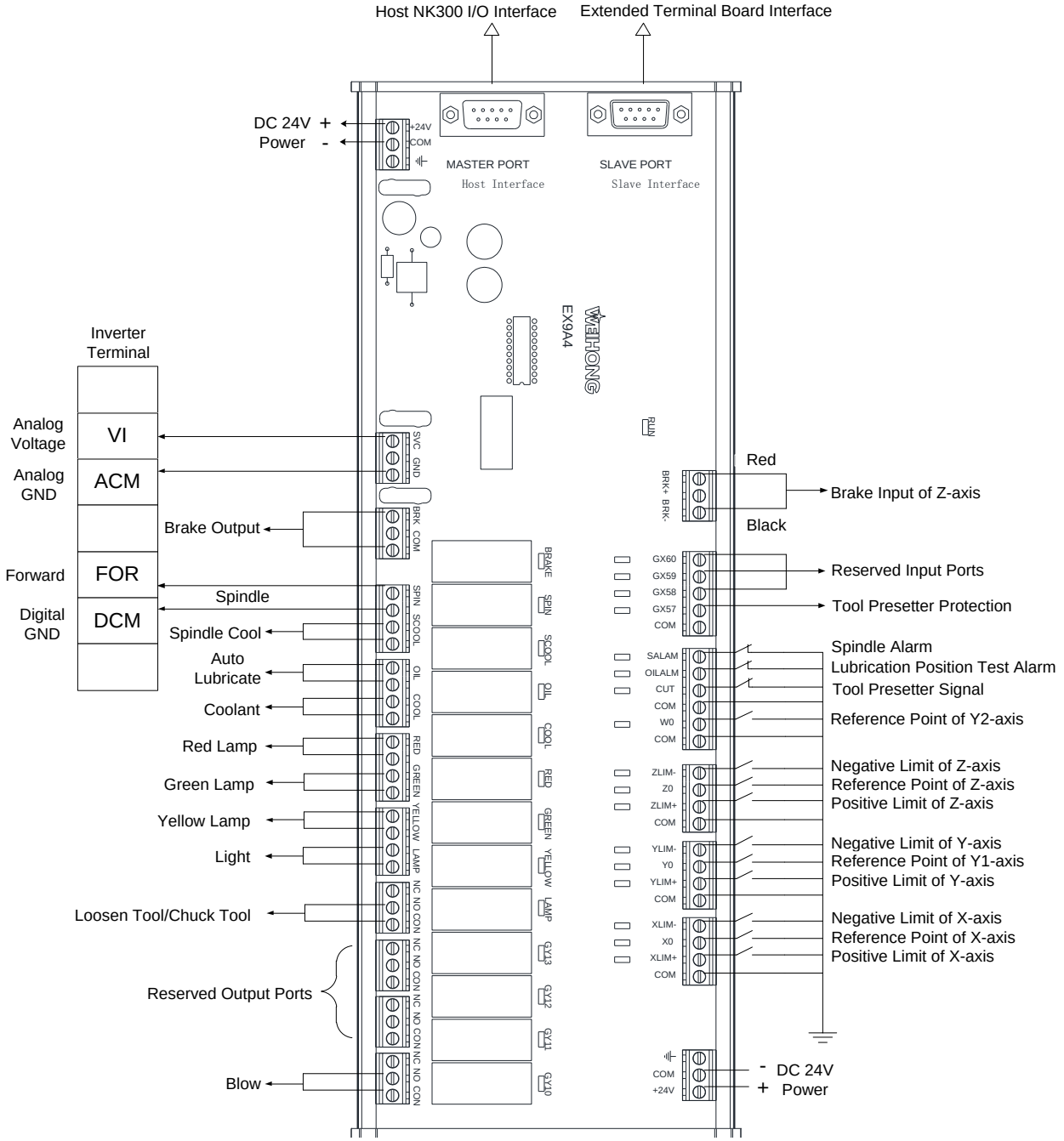


Fig. 2-14 Wiring Diagram of Terminal Board EX9A4 in Double Y Axis Configuration

Double Z Axis Configuration

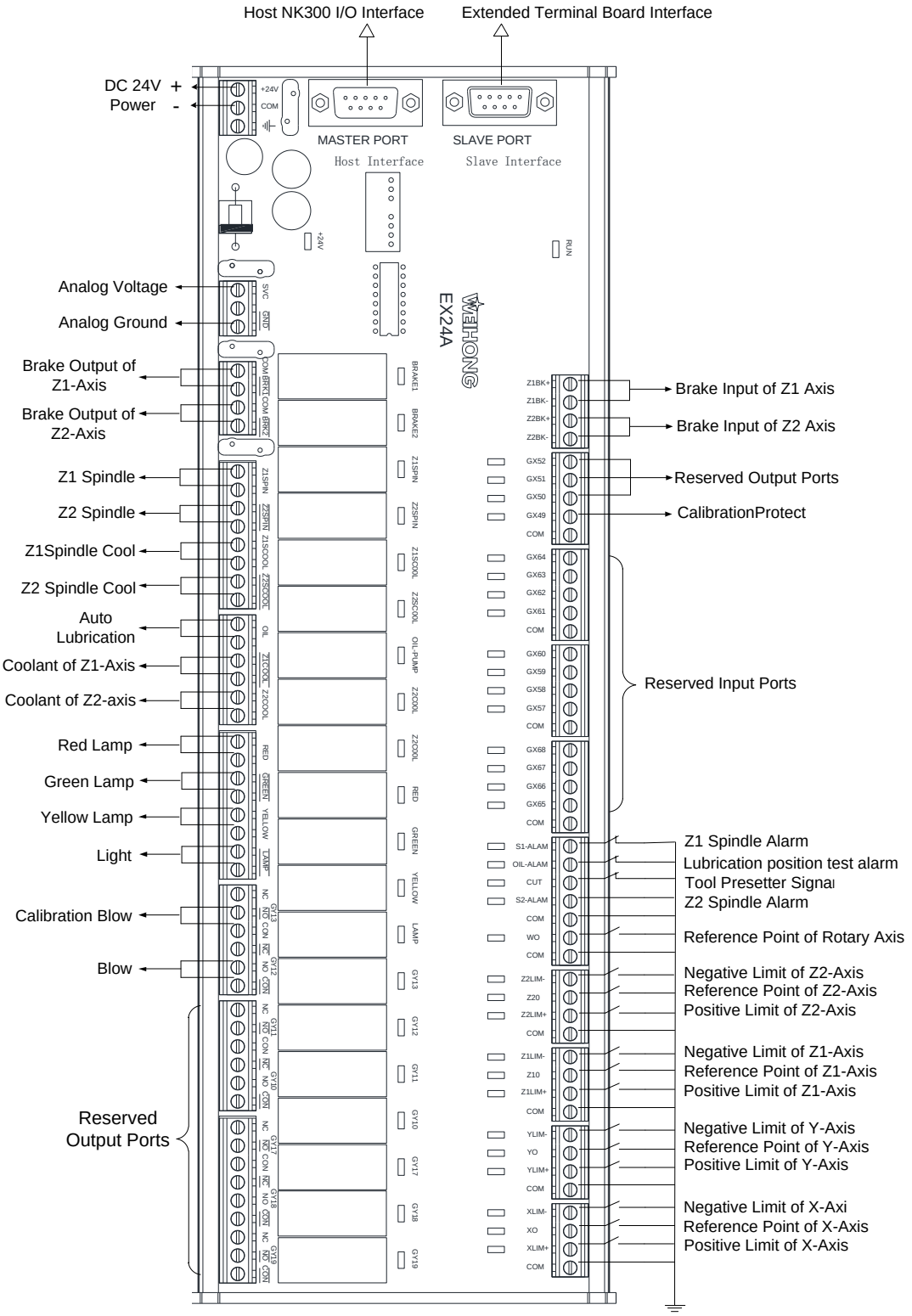


Fig. 2-15 Wiring Diagram of Terminal Board EX24A in Double Z Axis Configuration

- **Punching and Tapping Configuration**

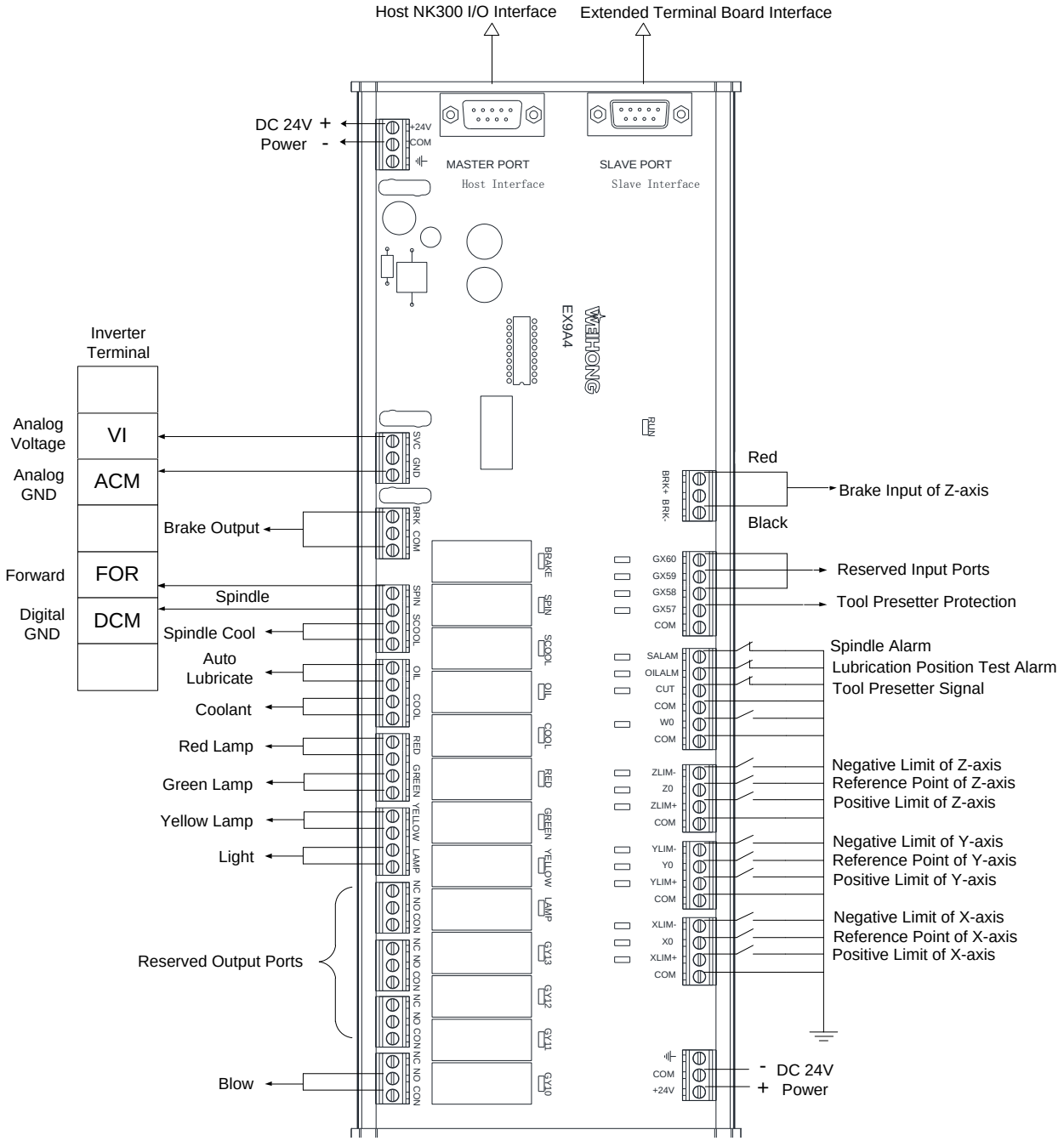


Fig. 2-16 Wiring Diagram of Terminal Board EX9A4 in Punching and Tapping Configuration

- **Wiring Diagram of Extended Terminal Board EX26A3**

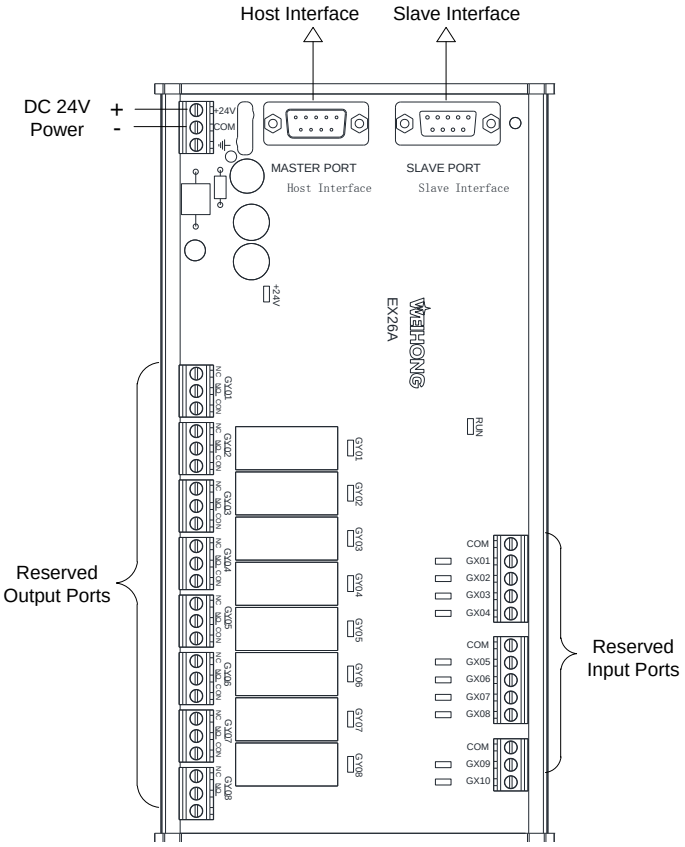


Fig. 2-17 Wiring Diagram of Extended Terminal Board EX26A3

2.3.2 Port Specification of Terminal Board

- **Three-axis Standard Configuration**

Category	Pin No.	Description	Specification
External Power	+24V	DC 24V power	Terminal board should be powered by external power supply.
	COM		
Origin Signal	X0/Y0/Z0	Reference point of X/Y/Z axis	Binary input, active low; connected to reference point switch of X/Y/Z axis.
	COM	Common port	Signal common port.
Limit Signal	XLIM+/YLIM+/ ZLIM+	Positive limit of X/Y/Z axis	Binary input, active low; connected to positive limit switch of X/Y/Z axis.
	XLIM-/YLIM-/ ZLIM-	Negative limit of X/Y/Z axis	Binary input, active low; connected to negative limit switch of X/Y/Z axis.
	COM	Common port	Common port of digital binary signal.
Common Input Ports	SALAM	Spindle alarm	Binary input signal, connected to spindle alarm switch.
	OILAM	Lubrication position test alarm	Binary input signal, connected to sense switch of lubrication position.

Category	Pin No.	Description	Specification
	GX32(CUT)	Tool Presetter Signal	Binary input signal, connected to tool presetter signal.
	GX57	Tool Presetter Protection	Binary input signal, connected to tool presetter protection.
	COM	Common port	Common port of digital binary signal.
Spindle Control	SVC	Signal output of analog voltage (from 0 to 10V)	Controls the spindle motor speed by controlling inverter frequency due to voltage change. Externally connects to the inverter analog voltage frequency command input port (i.e. AVI/VI)
	GND	Analog voltage ground	Connects to Inverter analog ground (generally known as ACM)
	SPIN	Spindle start/ stop	Relay output, its two terminals separately connected to the inverter digital ground (i.e. DCM) and the inverter forward rotation input port (i.e. FOR).
Z-axis Brake	BRK	Brake control	Relay output signal. Powered by 24V of terminal board, “Brake” port is directly connected to brake coil, forming a brake circuit. The two terminals of “Brake” will be conductive after the servo of machine tool is normally on, and Z brake will be valid.
	BRK+; BRK-	Brake input of Z-axi	Two cables of Z-axis are included, red or blue one for brake signal of servo output (open-collector output), black for grounding. And red or blue line should be connected to “BK+”, while black line to “BK-”.
Signal Lamp	GY04(RED)	Red lamp	Red light on when machining ends or during E-stop
	GY03(GREEN)	Green lamp	Green light on during normal working state of machine
	GY02(YELLOW)	Yellow Lamp	Yellow light on during idle state after machining ends or during waiting state
Common Output Ports	GY06(OIL)	Auto Lubricate	The port controls auto lubrication. Relay contact output. LED on during lubrication and off when lubrication stops
	GY05(COOL)	Coolant	Relay contact output. Two terminals equaling to a switch, connected to workpiece cooling switch.
	GY07(SCOOL)	Spindle cool	Relay contact output. Two terminals equals to a switch, connecting to spindle cooling switch.
	GY10	Blow	Relay contact output. Two terminals equals to

Category	Pin No.	Description	Specification
			a switch, connecting to blow switch.
	GY14	Light	Relay contact output. Two terminals equaling to a switch, connected to light switch.
Reserved Input Ports	GX58~GX60	Reserved input ports	Available for customized input.
Reserved Output Ports	GY11~GY13	Reserved output ports	Available for customized output.

- **Four-axis Standard Configuration**

Category	Pin No.	Description	Specification
External Power	+24V	DC 24V power	Terminal board should be powered by external power supply.
	COM		
Origin Signal	X0/Y0/Z0/W0	Reference point of X/Y/Z axis	Binary input, active low; connected to reference point switch of X/Y/Z/A axis.
	COM	Common port	Signal common port.
Limit Signal	XLIM+/YLIM+/ ZLIM+	Positive limit of X/Y/Z axis	Binary input, active low; connected to positive limit switch of X/Y/Z axis.
	XLIM-/YLIM-/ ZLIM-	Negative limit of X/Y/Z axis	Binary input, active low; connected to negative limit switch of X/Y/Z axis.
	COM	Common port	Common port of digital binary signal.
Common Input Ports	SALAM	Spindle alarm	Binary input signal, connected to spindle alarm switch.
	OILAM	Lubrication position test alarm	Binary input signal, connected to sense switch of lubrication position.
	GX32(CUT)	Tool Presetter Signal	Binary input signal, connected to tool presetter signal.
	COM	Common port	Common port of digital binary signal.
Spindle Control	SVC	Signal output of analog voltage (from 0 to 10V)	Controls the spindle motor speed by controlling inverter frequency due to voltage change. Externally connects to the inverter analog voltage frequency command input port (i.e. AVI/VI)
	GND	Analog voltage ground	Connects to Inverter analog ground (generally known as ACM)
	SPIN	Spindle start/ stop	Relay output, its two terminals separately connected to the inverter digital ground (i.e. DCM) and the inverter forward rotation input port (i.e. FOR).
Z-Axis	BRK	Brake control	Relay output signal. Powered by 24V of

Category	Pin No.	Description	Specification
Brake			terminal board, “Brake” port is directly connected to brake coil, forming a brake circuit. The two terminals of “Brake” will be conductive after the servo of machine tool is normally on, and Z brake will be valid.
	BRK+; BRK-	Brake input of Z-axis	Two cables of Z-axis are included, red or blue one for brake signal of servo output (open-collector output), black for grounding. And red or blue line should be connected to “BK+”, while black line to “BK-”.
Signal Lamp	GY04(RED)	Red lamp	Red light on when machining ends or during E-stop
	GY03(GREEN)	Green lamp	Green light on during normal working state of machine
	GY02(YELLOW)	Yellow Lamp	Yellow light on during idle state after machining ends or during waiting state
Common Output Ports	GY06(OIL)	Auto Lubricate	The port controls auto lubrication. Relay contact output. LED on during lubrication and off when lubrication stops
	GY05(COOL)	Coolant	Relay contact output. Two terminals equaling to a switch, connected to workpiece cooling switch.
	GY07(SCOOL)	Spindle cool	Relay contact output. Two terminals equals to a switch, connecting to spindle cooling switch.
Reserved Input Ports	GX57~GX60	Reserved input ports	Available for customized input.
Reserved Output Ports	GY10~GY13	Reserved output ports	Available for customized output.

- **Double Y axis Configuration**

Category	Pin No.	Description	Specification
External Power	+24V	DC 24V power	Terminal board should be powered by external power supply.
	COM		
Origin Signal	X0/Y0/Z0/W0	Reference point of X/Y1/Z/Y2 axis	Binary input, active low; connected to reference point switch of X/Y1/Z/Y2 axis.
	COM	Common port	Signal common port.
Limit Signal	XLIM+/YLIM+/ ZLIM+	Positive limit of X/Y/Z axis	Binary input, active low; connected to positive limit switch of X/Y/Z axis.
	XLIM-/YLIM-/	Negative limit of	Binary input, active low; connected to

Category	Pin No.	Description	Specification
	ZLIM-	X/Y/Z axis	negative limit switch of X/Y/Z axis.
	COM	Common port	Common port of digital binary signal.
Common Input Ports	GX64(SALAM)	Spindle alarm	Binary input signal, connected to spindle alarm switch.
	GX63(OILALM)	Lubrication position test alarm	Binary input signal, connected to sense switch of lubrication position.
	GX32(CUT)	Tool Presetter Signal	Binary input signal, connected to tool presetter signal.
	GX57	Tool Presetter Protection	Binary input signal, connected to tool presetter protection.
	COM	Common port	Common port of digital binary signal.
Spindle Control	SVC	Signal output of analog voltage (from 0 to 10V)	Controls the spindle motor speed by controlling inverter frequency due to voltage change. Externally connects to the inverter analog voltage frequency command input port (i.e. AVI/VI)
	GND	Analog voltage ground	Connects to Inverter analog ground (generally known as ACM)
	SPIN	Spindle start/ stop	Relay output, its two terminals separately connected to the inverter digital ground (i.e. DCM) and the inverter forward rotation input port (i.e. FOR).
Z-Axis Brake	BRK	Brake control	Relay output signal. Powered by 24V of terminal board, “Brake” port is directly connected to brake coil, forming a brake circuit. The two terminals of “Brake” will be conductive after the servo of machine tool is normally on, and Z brake will be valid.
	BRK+; BRK-	Brake input of Z-axis	Two cables of Z-axis are included, red or blue one for brake signal of servo output (open-collector output), black for grounding. And red or blue line should be connected to “BK+”, while black line to “BK-”.
Signal Lamp	GY04(RED)	Red lamp	Red light on when machining ends or during E-stop
	GY03(GREEN)	Green lamp	Green light on during normal working state of machine
	GY02(YELLOW)	Yellow Lamp	Yellow light on during idle state after machining ends or during waiting state
	GY13	Loosen tool/Chuck tool	The light is on when the spindle loosens or chucks tool.
Common	GY06(OIL)	Auto Lubricate	The port controls auto lubrication. Relay

Category	Pin No.	Description	Specification
Output Ports			contact output. LED on during lubrication and off when lubrication stops
	GY05(COOL)	Coolant	Relay contact output. Two terminals equaling to a switch, connected to workpiece cooling switch.
	GY07(SCOOL)	Spindle cool	Relay contact output. Two terminals equals to a switch, connecting to spindle cooling switch.
	GY10	Blow	Relay contact output. Two terminals equals to a switch, connecting to blow switch.
Reserved Input Ports	GX58~GX60	Reserved input ports	Available for customized input.
Reserved Output Ports	GY11~GY12	Reserved output ports	Available for customized output.

- **Double Z axis Configuration**

Category	Pin No.	Description	Specification
External Power	+24V	DC 24V power	Terminal board should be powered by external power supply.
	COM		
Origin Signal	X0/Y0/Z10/Z20 /W0	Reference point of X/Y/Z1/Z2/rotary axis	Binary input, active low; connected to reference point switch of X/Y/Z1/Z2/rotary axis.
	COM	Common port	Signal common port.
Limit Signal	XLIM+/YLIM+/ Z1LIM+/ Z2LIM+	Positive limit of X/Y/Z1/Z2 axis	Binary input, active low; connected to positive limit switch of X/Y/Z1/Z2 axis.
	XLIM-/YLIM-/ Z1LIM-/ Z2LIM+	Negative limit of X/Y/Z1/Z2 axis	Binary input, active low; connected to negative limit switch of X/Y/Z1/Z2 axis.
	COM	Common port	Common port of digital binary signal.
Common Input Ports	S1-ALAM	Z1 spindle alarm	Binary input signal, connected to Z1 spindle alarm switch.
	S2-ALAM	Z2 spindle alarm	Binary input signal, connected to Z2 spindle alarm switch.
	OILAM	Lubrication position test alarm	Binary input signal, connected to sense switch of lubrication position.
	CUT	Tool Presetter Signal	Binary input signal, connected to tool presetter signal.
	GX49	CalibrationProtect	Binary input signal, connected to tool presetter protection.
	COM	Common port	Common port of digital binary signal.

Category	Pin No.	Description	Specification
Spindle Control	SVC	Signal output of analog voltage (from 0 to 10V)	Controls the spindle motor speed by controlling inverter frequency due to voltage change. Externally connects to the inverter analog voltage frequency command input port (i.e. AVI/VI)
	GND	Analog voltage ground	Connects to Inverter analog ground (generally known as ACM)
	Z1SPIN; Z2SPIN	Spindle start/ stop	Relay output, its two terminals separately connected to the inverter digital ground (i.e. DCM) and the inverter forward rotation input port (i.e. FOR).
Z-Axis Brake	BRK1; BRK2	Brake control	Relay output signal. Powered by 24V of terminal board, “Brake” port is directly connected to brake coil, forming a brake circuit. The two terminals of “Brake” will be conductive after the servo of machine tool is normally on, and Z brake will be valid.
	Z1BRK+; Z1BRK-	Brake input of Z1-axis	Two cables of Z1 axis are included, red or blue one for brake signal of servo output (open-collector output), black for grounding. And red or blue line should be connected to “Z1 BK+”, while black line to “Z1 BK-”.
	Z2BRK+; Z2BRK-	Brake input of Z2-axis	Two cables of Z2 axis are included, red or blue one for brake signal of servo output (open-collector output), black for grounding. And red or blue line should be connected to “Z2 BK+”, while black line to “Z2 BK-”.
Signal Lamp	RED	Red lamp	Red light on when machining ends or during E-stop
	GREEN	Green lamp	Green light on during normal working state of machine
	YELLOW	Yellow Lamp	Yellow light on during idle state after machining ends or during waiting state
	GY13	Calibration blow	The light is on during calibration blow.
Common Output Ports	OIL	Auto Lubricate	The port controls auto lubrication. Relay contact output. LED on during lubrication and off when lubrication stops
	Z1COOL; Z2COOL	Coolant	Relay contact output. Two terminals equaling to a switch, connected to workpiece cooling switch.
	Z1SCOOL; Z2SCOO	Spindle cool	Relay contact output. Two terminals equals to a switch, connecting to spindle cooling switch.

Category	Pin No.	Description	Specification
	GY12	Blow	Relay contact output. Two terminals equals to a switch, connecting to blow switch.
Reserved Input Ports	GX57~GX68	Reserved input ports	Available for customized input.
Reserved Output Ports	GY10; GY11; GY17	Reserved output ports	Available for customized output.

- **Punching and Tapping Configuration**

Category	Pin No.	Description	Specification
External Power	+24V	DC 24V power	Terminal board should be powered by external power supply.
	COM		
Origin Signal	X0/Y0/Z0	Reference point of X/Y/Z axis	Binary input, active low; connected to reference point switch of X/Y/Z axis.
	COM	Common port	Signal common port.
Limit Signal	XLIM+/YLIM+/ ZLIM+	Positive limit of X/Y/Z axis	Binary input, active low; connected to positive limit switch of X/Y/Z axis.
	XLIM-/YLIM-/ ZLIM-	Negative limit of X/Y/Z axis	Binary input, active low; connected to negative limit switch of X/Y/Z axis.
	COM	Common port	Common port of digital binary signal.
Common Input Ports	SALAM	Spindle alarm	Binary input signal, connected to spindle alarm switch.
	OILAM	Lubrication position test alarm	Binary input signal, connected to sense switch of lubrication position.
	GX32(CUT)	Tool Presetter Signal	Binary input signal, connected to tool presetter signal.
	GX57	Tool Presetter Protection	Binary input signal, connected to tool presetter protection.
	COM	Common port	Common port of digital binary signal.
Spindle Control	SVC	Signal output of analog voltage (from 0 to 10V)	Controls the spindle motor speed by controlling inverter frequency due to voltage change. Externally connects to the inverter analog voltage frequency command input port (i.e. AVI/VI)
	GND	Analog voltage ground	Connects to Inverter analog ground (generally known as ACM)
	SPIN	Spindle start/ stop	Relay output, its two terminals separately connected to the inverter digital ground (i.e. DCM) and the inverter forward rotation input port (i.e. FOR).

Category	Pin No.	Description	Specification
Z-Axis Brake	BRK	Brake control	Relay output signal. Powered by 24V of terminal board, “Brake” port is directly connected to brake coil, forming a brake circuit. The two terminals of “Brake” will be conductive after the servo of machine tool is normally on, and Z brake will be valid.
	BRK+; BRK-	Brake input of Z-axis	Two cables of Z-axis are included, red or blue one for brake signal of servo output (open-collector output), black for grounding. And red or blue line should be connected to “BK+”, while black line to “BK-”.
Signal Lamp	GY04(RED)	Red lamp	Red light on when machining ends or during E-stop
	GY03(GREEN)	Green lamp	Green light on during normal working state of machine
	GY02(YELLOW)	Yellow Lamp	Yellow light on during idle state after machining ends or during waiting state
Common Output Ports	GY06(OIL)	Auto Lubricate	The port controls auto lubrication. Relay contact output. LED on during lubrication and off when lubrication stops
	GY05(COOL)	Coolant	Relay contact output. Two terminals equaling to a switch, connected to workpiece cooling switch.
	GY07(SCOOL)	Spindle cool	Relay contact output. Two terminals equals to a switch, connecting to spindle cooling switch.
	GY10	Blow	Relay contact output. Two terminals equals to a switch, connecting to blow switch.
	GY14(LAMP)	Light	Relay contact output. Two terminals equaling to a switch, connected to light switch.
Reserved Input Ports	GX58~GX60	Reserved input ports	Available for customized input.
Reserved Output Ports	GY11~GY13	Reserved output ports	Available for customized output.

2.4 Port Definition and Wiring Specification

2.4.1 Driver Interface Definition

NK300 system provides 4 pulse feed driver interfaces, i.e. for X, Y, Z and A respectively. The type of the

interface is 15-pin D-type socket (DB15 pins). The pins definition is shown as Fig. 2-18.

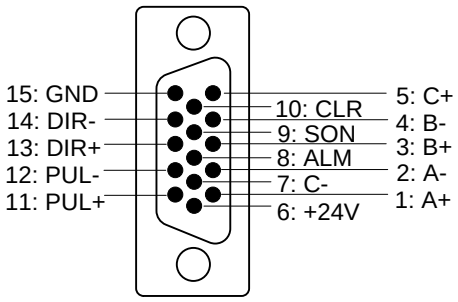


Fig. 2-18 Driver Interface Definition

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, this 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	The alarm state will be cleared when this signal keeps closing with COM- for above 120ms.
PUL+; PUL-	Pulse output	Output, differential signal transmission mode	-
DIR+; DIR-	Direction output	Output, differential signal transmission mode	-
+24V; GND	DC 24V power	Output	Connected to driver

CAUTION

SON signal will be effective at 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.

2.4.2 Handwheel Interface Definition

NK300 is available of an external manual pulse generator, with DB15 core dual-in-line holes joint. Its pin definition is as shown in Fig. 2-19.

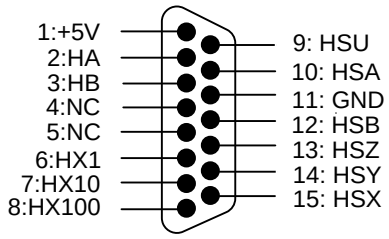


Fig. 2-19 Handwheel Interface Definition

Pin No.	Definition	Description
1	+5V	Power on handwheel encoder
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	NC	Selection of the 5th axis
11	GND	Digital ground
12	NC	Selection of the 6th axis
13	HSZ	Selection of Z-axis
14	HSY	Selection of Y-axis
15	HSX	Selection of X-axis

2.4.3 USB Interface Definition

There are two USB interfaces on the back of NK300 chassis, another one on operation panel for external connection of USB device (E.g. USB disk).

2.4.4 I/O Pin Definition on Panel

I/O interfaces on panel are connected to operation panel of NK300, and the pin definition is shown in Fig. 2-20:

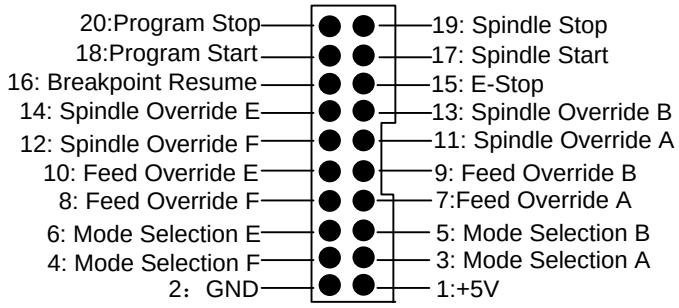


Fig. 2-20 I/O Pin Definition on Panel

2.4.5 Slot Definition of CF Card

Two CF cards are equipped in NK300 system, i.e. system disk and data disk separately.

3 Operation

3.1 Debugging Steps

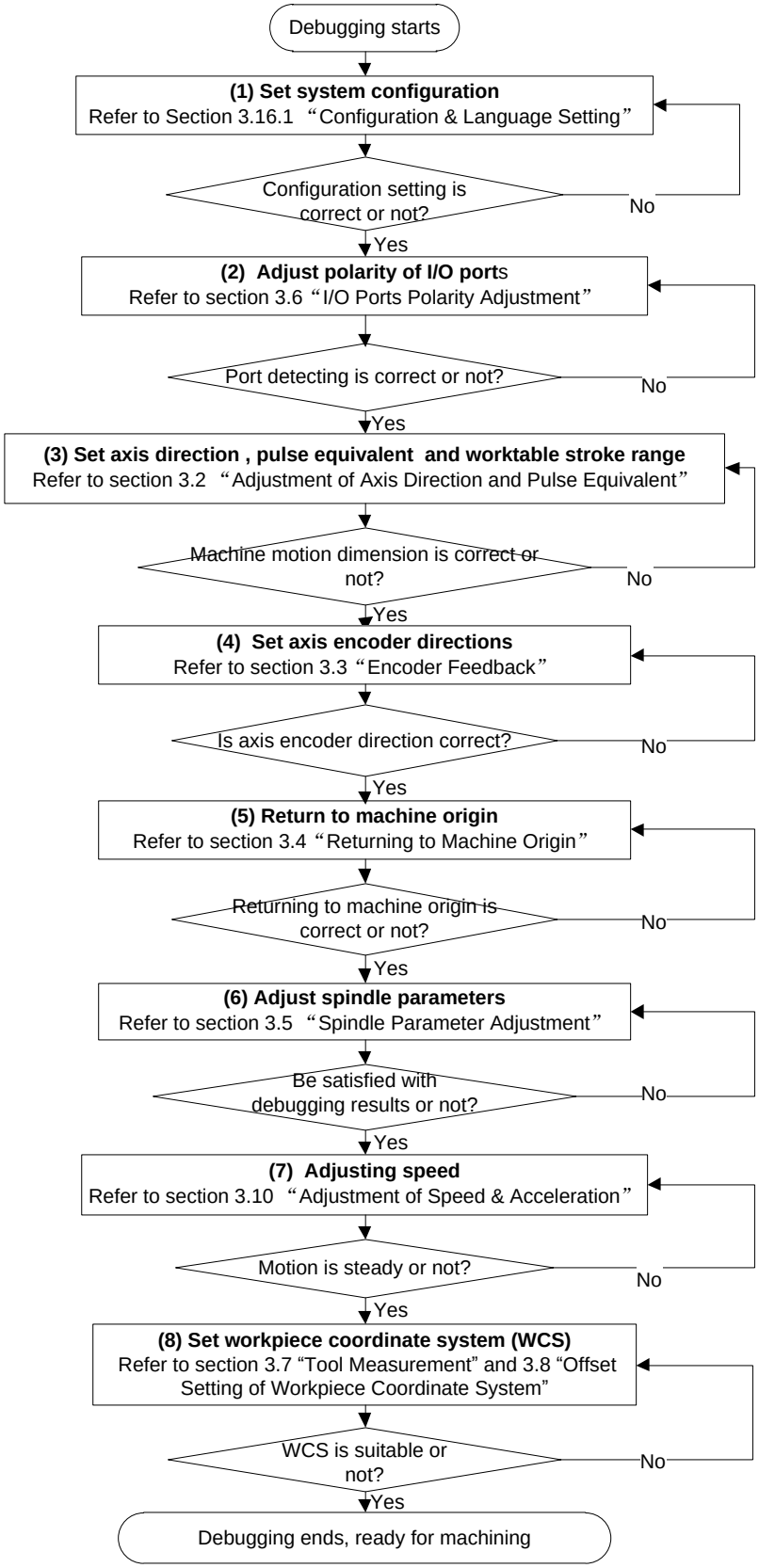


Fig. 3-1 Debugging Steps

3.2 Adjustment of Axis Direction and Pulse Equivalent

3.2.1 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. 3-2.

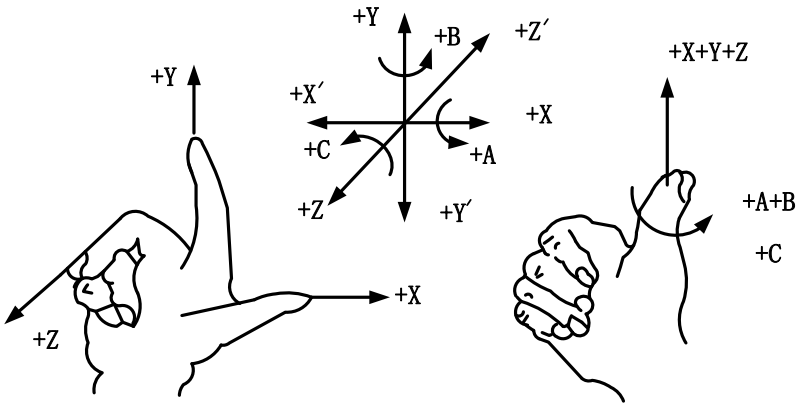


Fig. 3-2 Standard Coordinate System of Right Hand Principle

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. Take 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”).			

3.2.2 Pulse Equivalent Adjustment

Pulse equivalent (p): the moving distance of workbench or rotation degree of rotary axis per pulse sent by the CNC system, 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 NK300 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.

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

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 5 mm, 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:

$$\text{Pulse Equivalent} = \frac{5\text{mm}}{\frac{360}{1.8}\times 10\times 1} = 0.0025\text{mm/p}$$

- **Servo Motor**

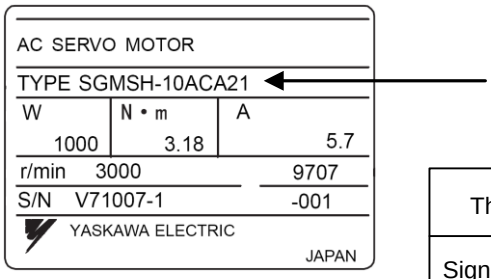
In general, set the default value of pulse equivalent (p) as 0.001mm/p and calculate electronic gear ratio (B/A). Their relationship can be shown as:

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

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

Electronic gear ratio (B/A): the parameter of servo driver (take YASKAWA driver as an example, B is PN202 while A is PN203). This ratio represents servo scales up or down the pulse frequency sent by CNC system. When B/A > 1, it means scaling up, and vice versa. For example, provided the pulse frequency sent by CNC system is 100HZ, if the numerator of electronic gear ratio (B) is set as 1 while the denominator 2, the actual running speed of servo is 50HZ. On the contrary, if the numerator is set as 2 while denominator 1, the actual running speed turns to 200HZ

Encoder Resolution (F): needed pulse number for one circle of servo motor. Please see the servo motor label plate and then refer to the corresponding manual to confirm its encoder resolution. A label plate of YASKAWA SGMSH type motor is as shown in Fig. 3-3, and the 4th character in motor type is the serial encoder specification, so the resolution of this motor is 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. 3-3 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.

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

- **Rotary Axis**

The pulse equivalent of rotary axis refers to the rotation degree of the axis clamping the workpiece corresponding to each pulse. The difference of rotary axis movement from linear axis movement lies in that the screw pitch of rotary axis is 360 degrees. Therefore, in calculating rotary axis pulse equivalent,

you just need to replace screw pitch with 360.

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.	-0.0000009~999

CAUTION

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

3.2.3 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 specifies 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 specifies 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 specifies whether to check the stroke range of worktable.	YES: enabled; NO: disabled

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.

3.3 Encoder Feedback

3.3.1 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.

The operation is the same 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, set X-axis direction as -1, and pulse as “Pulse + Direction, Negative Logic”. The value of parameter N11110 will be 1= (-1)×(-1).

Please refer to section 3.2.1 for axis direction setting, and refer to section 3.2.2 for pulse direction setting.

- **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.

3.3.2 Encoder Feedback Parameter Specification

- **Related Parameters**

Parameter		Details	Setting Range
N11110	Axis Encoder Dir	It specifies the direction of encoder.	1: positive -1: negative
N11160	Frequency Division Pulses	It specifies the encoder feedback pulse number via frequency division of servo per	1~999999

Parameter		Details	Setting Range
	of PG (*4)	revolution of motor.	
N11304	Encoder Feedback	Whether to enable encoder feedback function or not.	Yes: enabled; No: disabled
N80004	Print Info	Whether to show debugging info or not.	Yes: show; No: not show

Encoder feedback function is used to measure and give feedback to angular displacement and linear displacement of a screw servo motor. It can be enabled through modifying parameter N11304 “Encoder Feedback”. If the parameter is set as “No”, please see section 3.4.1 for the principle and process of the returning. Otherwise, please see section 3.4.2.

Only when parameter N11304 is set as “Yes”, parameter N80004 is enabled.

- **Encoder Error Related Parameters**

Parameter		Details	Setting Range
N11130	Check Encoder Error	Whether to check the encoder error between feedback value and output value or not.	Yes: check; No: do 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 P
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 P

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). Default is 500.

3.4 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 starting CNC system, it is necessary to return to machine origin (home all axes) manually or automatically.

CAUTION

Functions which will not be activated until backing to machine origin completed are Soft Limit, Setting Fixed Point and Tool Change.

- **The Process of Returning to Machine Origin**

The processes of returning to machine origin for X, Y, Z-axis are the same, as shown in Fig. 3-4 (an example of X-axis).

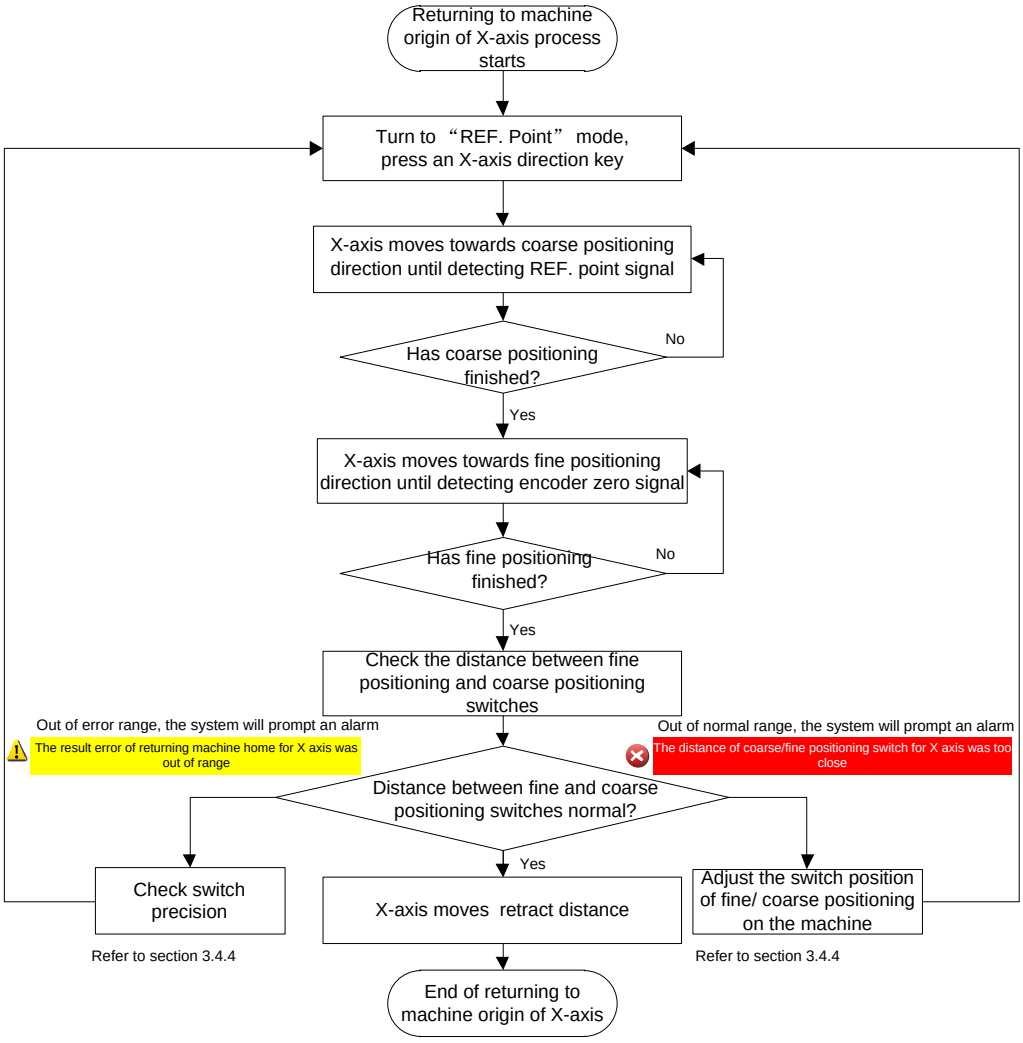


Fig. 3-4 The Process of Returning to Machine Origin (X-axis)

3.4.1 Principle of Returning to Machine Origin without Encoder Feedback

The encoder feedback function is involved in the system, specified by parameter N11304. The sketch map of returning to machine origin without encoder feedback with servo motor is as below:

- **Coarse Positioning Stage**

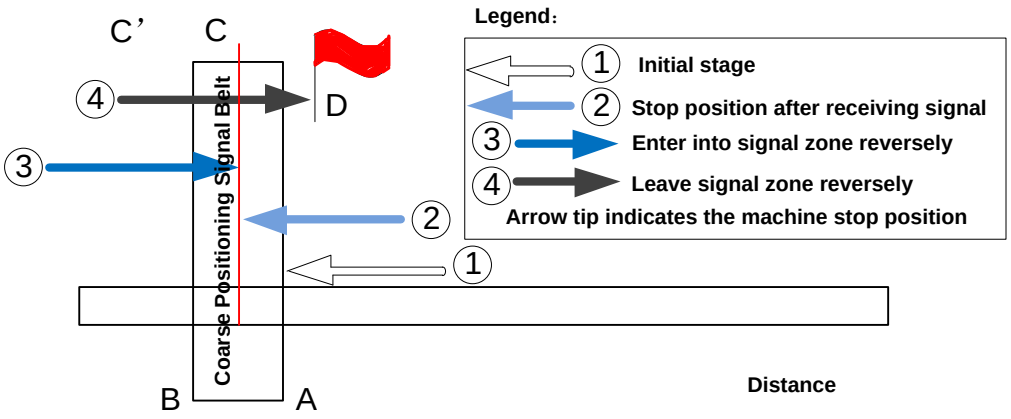


Fig. 3-5 Sketch Map of Coarse Positioning (Stopping within the Signal Belt after Receiving Coarse Positioning Signal)

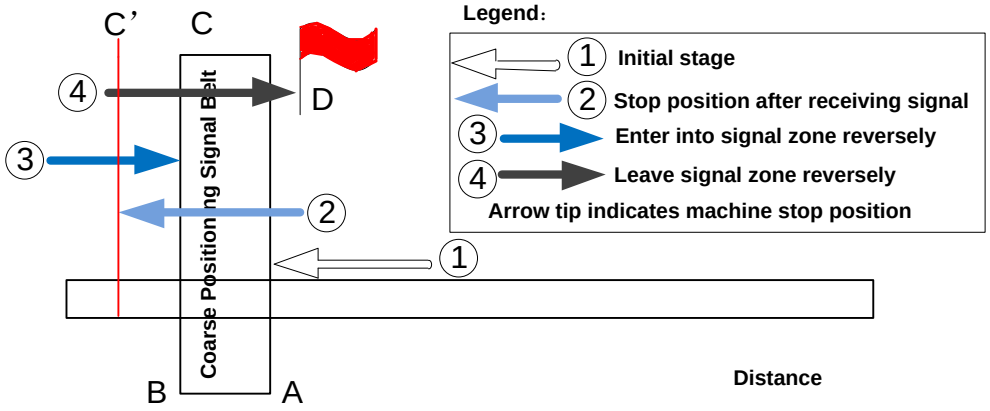


Fig. 3-6 Sketch Map of Coarse Positioning (Stopping Out of the Signal Belt after Receiving Coarse Positioning Signal)

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.

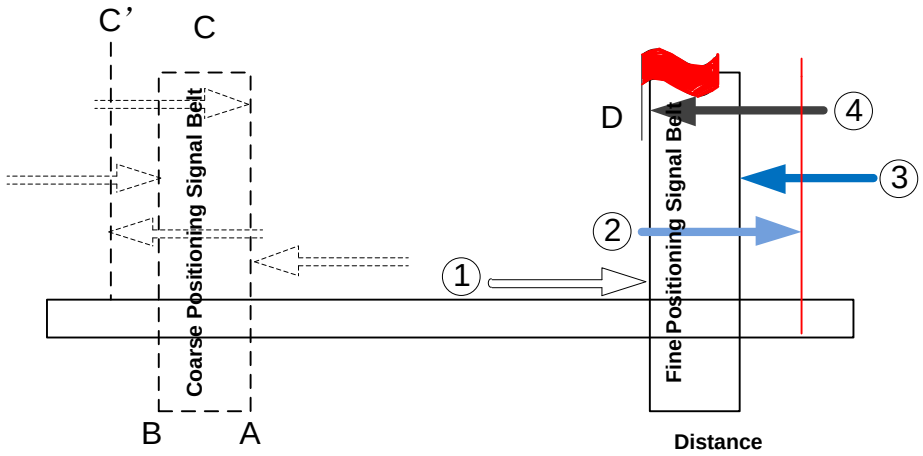


Fig. 3-7 The Process of Fine Positioning

- **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. 3-8.

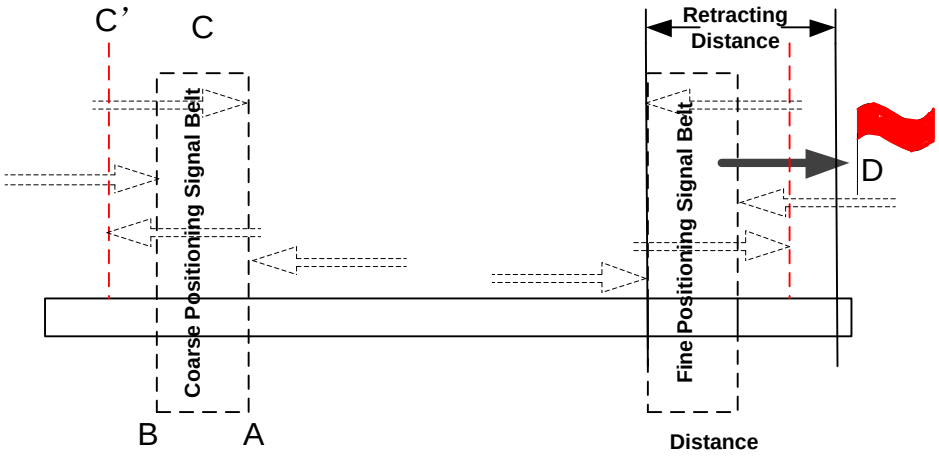


Fig. 3-8 Retracting Stage

3.4.2 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. 3-9:

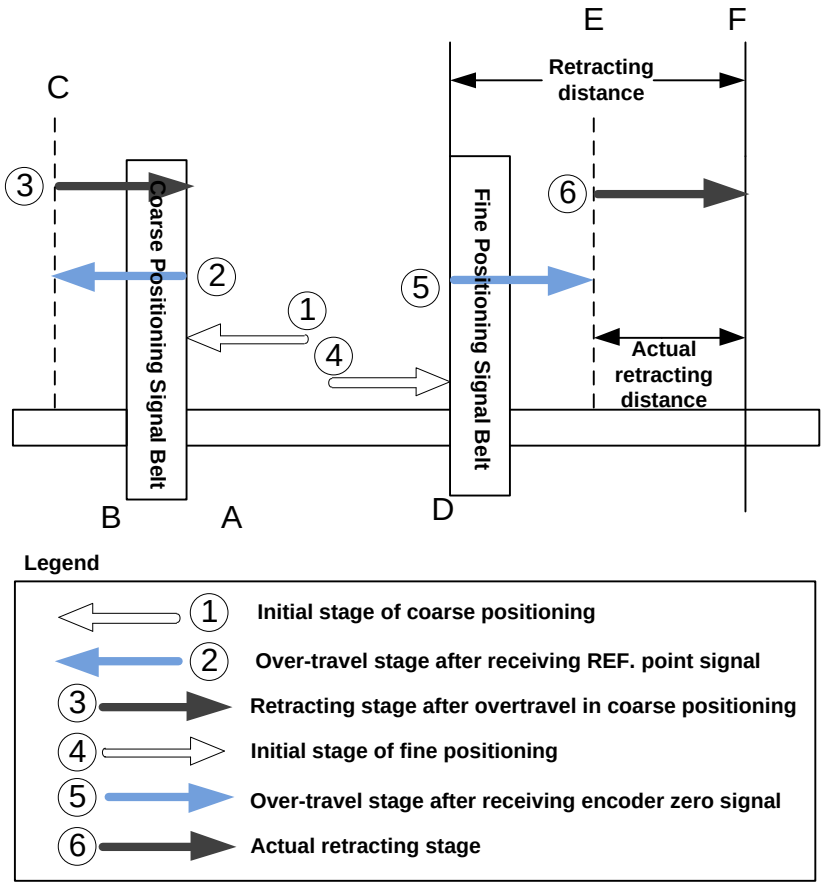


Fig. 3-9 The Process of Returning to Machine Origin

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.

3.4.3 Parameter Specifications

- **Safe Operations Related Parameters**

Parameter		Details	Setting Range
N74000	Cancel REF Sign when Reset	Whether cancel REF sign when reset.	YES: cancel; NO: do not Cancel

Parameter		Details	Setting Range
N74001	Back to REF Required	Whether backing to REF first before starting cycle is required or not.	YES: required; NO: not required
N74002	Cancel REF Sign when Estop	Whether cancel REF sign when E-stop.	YES: cancel; NO: do not Cancel
Returning to machine origin before machining can avoid machining offset to ensure precision of positioning. It is suggested to set N74002 and N74000 as "YES", and the system will remind you to return to machine origin by clearing machine origin mark "◒" when reset and e-stop occurs. And it is suggested to set N74001 as "YES", if there is no backing to machine origin mark "◒" 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. When you have enabled encoder feedback function, the system will correct the position of the machine tool automatically after e-stop to keep it same as the data in the system. In this case, it is suggested to set N74002 as "NO" and N74000 as "YES"			

- **Related Parameters in the Process of Backing to Machine Origin**

Parameter		Details	Setting Range
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
N74050	Fine Positioning Dir. (X/Y/Z)	The moving direction of machine at any point towards encoder zero	1: positive direction; -1: negative direction
N74060	F in Fine Positioning	Moving speed of machine towards encoder zero (fine positioning speed)	0.001~10000
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
N74090	Home Latch Count	Times of fine positioning in returning to machine origin, with default setting of "1"	-
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.			

Parameter	Details	Setting Range
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 3.4.4 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**

Parameter		Details	Setting Range
N74100	Leadscrew Pitch	For analysis of switch distance of fine and coarse positioning in backing to machine origin	0~100
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

Too close distance between home switch and encoder zero switch may lead to deviation of one 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 “. Press letter key N “ Clear History Records(N) ” to clear the measured history record after changing the home switch.

Related to the specific machine tool, N74100 should be set after measured in actual operation.

3.4.4 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. 3-10.

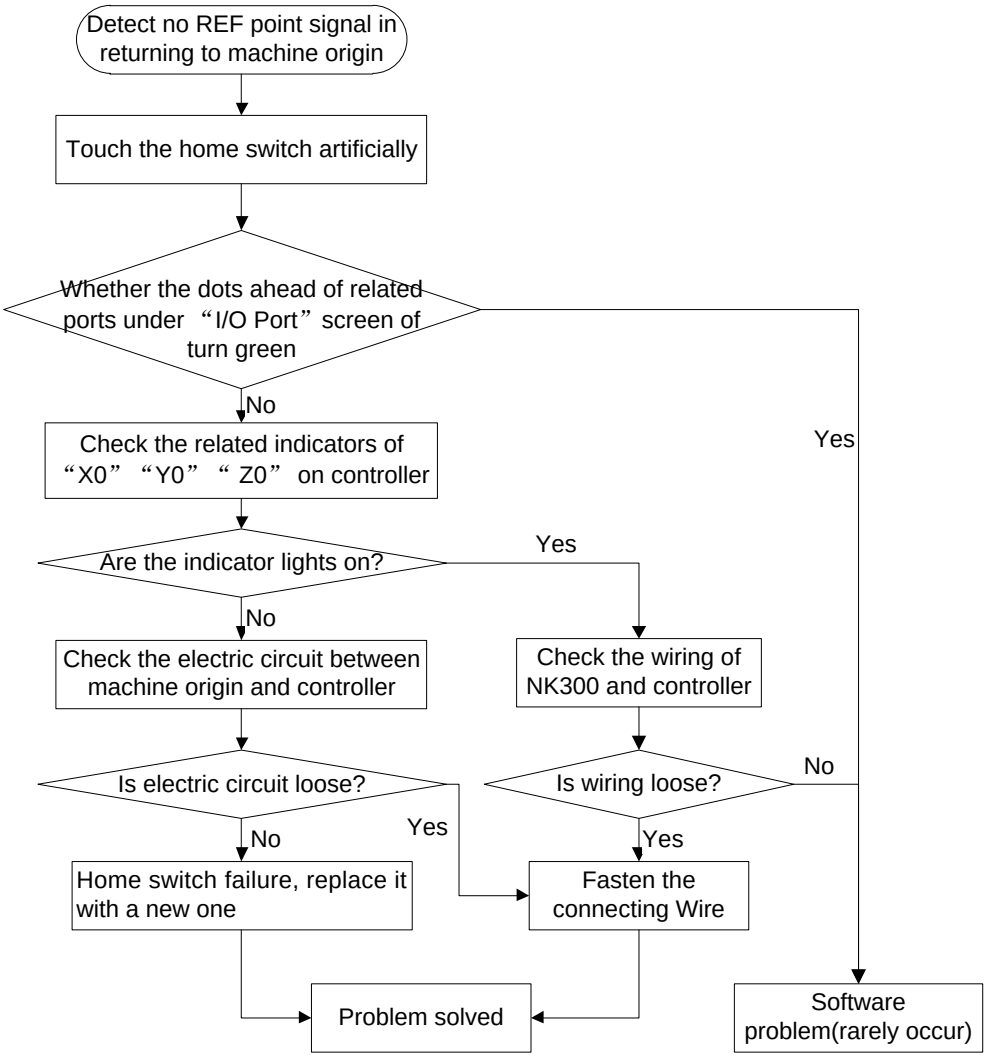


Fig. 3-10 Debugging Steps

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.

3.5 Spindle Parameter Adjustment

You can set the spindle rate directly in the software.

In auto mode, press key SF1 to enter the function area **General** and then press letter key A to enter interface Coor-Auto, as shown in Fig. 3-11.

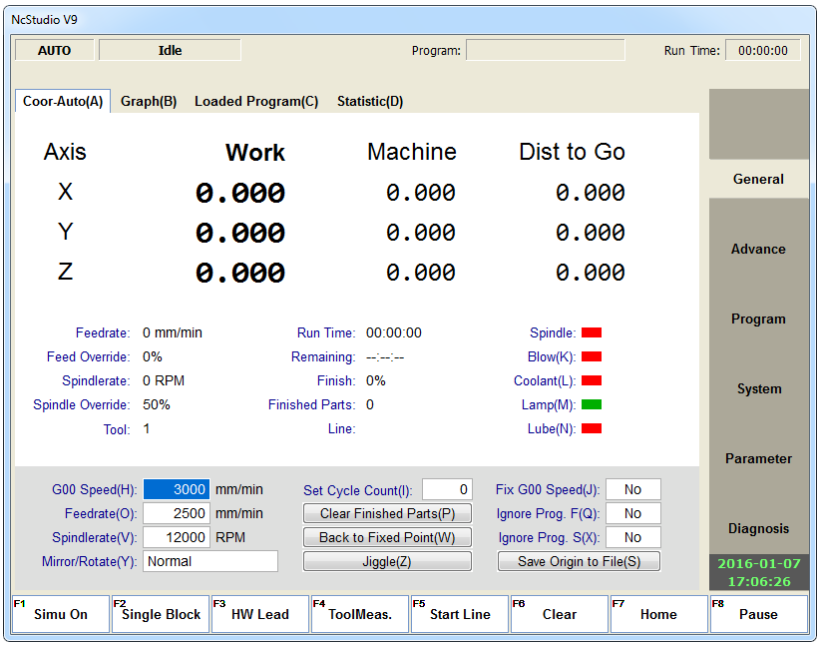


Fig. 3-11 Interface Coor-Auto

Spindle speed can be directly set in the parameter setting area above the manipulation button bar. As shown in Fig. 3-12, when "Ignore Prog. S(X)" is set to "Yes", spindle speed in auto machining will be the system setting value, i.e. the value of "Sindlerate(V)". When "Ignore Prog. S(X)" is set to "No", spindle speed in auto machining is the specified spindle speed in the machining file.

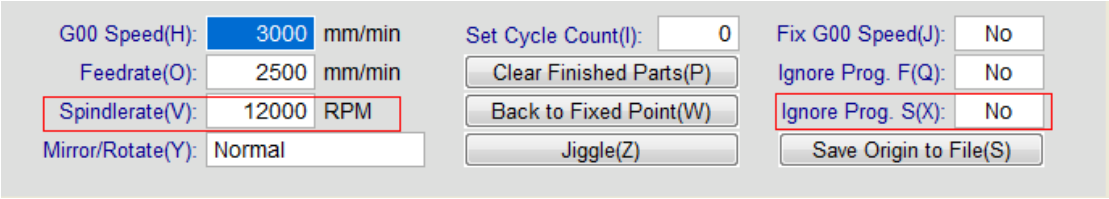


Fig. 3-12 Parameter Setting Area-Spindle Speed Setting

There are two ways to select and set the parameters:

1) Press " ↑ " and " ↓ " to move the cursor onto the desired parameter, and then press Enter to open the input box.

2) Press the corresponding shortcut key to open the corresponding parameter setting input box. For instance, for "Spindlerate(V)", press letter key "V" on the operation panel to open the input box for setting spindle rate.

Spindle rate 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. 3-13.

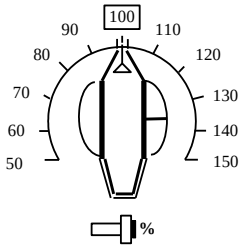


Fig. 3-13 Spindle Override Knob

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
The value of "Sindlerate(V)" must be less than that of N20001; the max. setting value of N20001 corresponds 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. $\text{Real-time voltage of analog SVC} = \frac{\text{Current spindle speed}}{\text{N20001}} \times 10\text{V} \times \text{Spindle override}$			

- **Related Parameters**

Parameter		Details	Setting Range
N20005	SpindleCool Off Delay Time	Delay time of closing spindle cooling pump after spindle stop	0~600

Parameter		Details	Setting Range
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
Parameter N20003 sets 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 Cycle Stop	Whether spindle will automatically stop when machining stops regularly	YES: stop; NO: do not stop
N72008	Spindle On when Cycle Start	Whether spindle will automatically rotate when machining begins	YES: stop; NO: do not stop
N73005	Stop Spindle on Pause	Whether spindle will automatically stop when machining pauses	YES: stop; NO: do not stop
This group of parameters sets the spindle action when commands of machining stop/ start/ pause are executed.			

3.6 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 SF6 to access functional area **Diagnosis**, then press key "C" to enter interface **I/O Port**. Select the target I/O port for modification by pressing the key " ↑ " and " ↓ ", and press F5 to modify the polarity of the port. Restart to validate the modification.

Interface **I/O Port** is as shown in Fig. 3-14, and some function screens need password before operation, such as Test On, Test Off, Cancel All], Invert Polarity and Change Property.

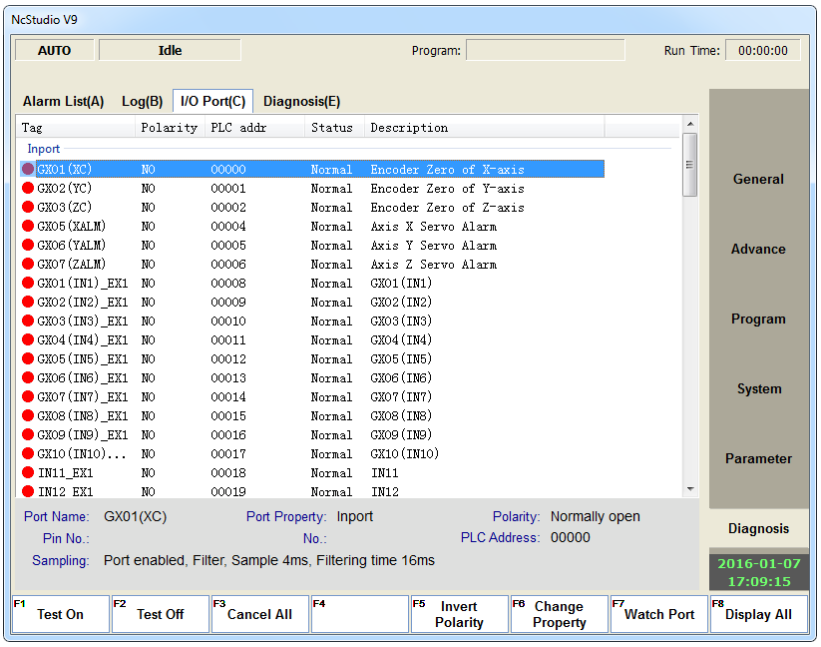


Fig. 3-14 Interface I/O Port

- **Test On/Off**

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**

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

- **Invert Polarity**

Press F5 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”.

- **Change Property**

Press F6, a new manipulation dialogue box will pop out, as shown in Fig. 3-15.

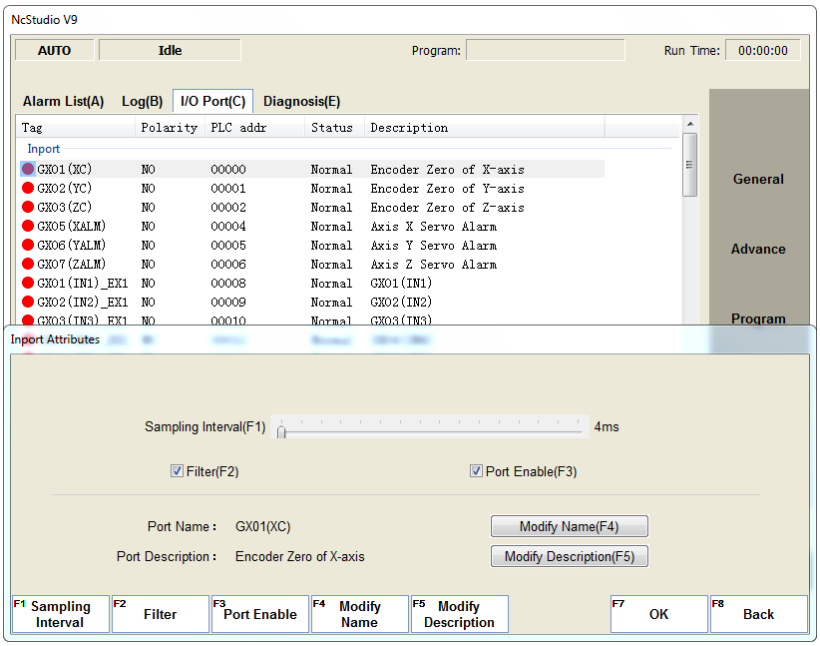


Fig. 3-15 Dialog Box Input Port Attribute

In this attribute dialogue, you can set sampling interval (Press F1 and up or down simultaneously), select filter and port enable function, and modify port name and description.

- **Watch Port**

Locate cursor to target port by pressing direction keys, and press F7 to monitor the port. Refer to section 3.13.2 for detail.

- **Display All**

Press F7 to display all I/O ports, including ports hidden by default. Press F8 again to hide all I/O ports not in use.

3.7 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).

With the help of a tool presetter, tool measurement is realized. As shown in Fig. 3-16, 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, fixed calibration, and first/exchanged calibration.

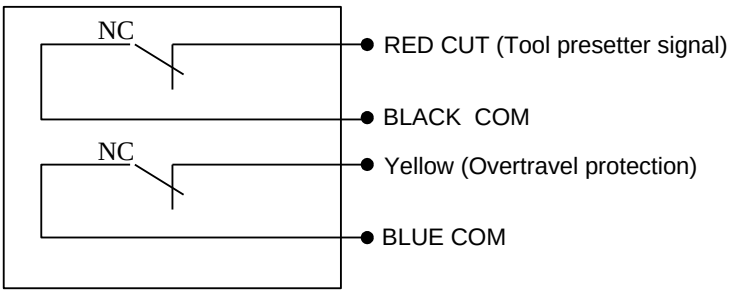


Fig. 3-16 Electrical Wiring Diagram of A WEIHONG Tool Presetter

Fig. 3-17 is the sketch map of tool calibration with a tool presetter.

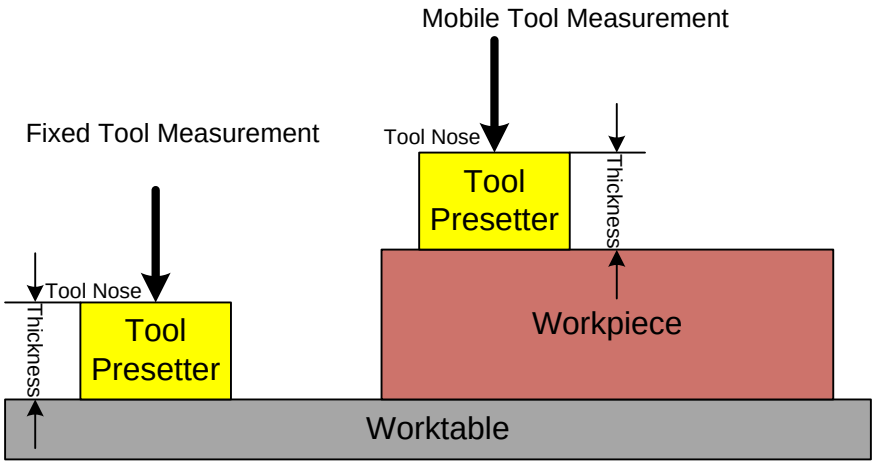


Fig. 3-17 Sketch Map with A Tool Presetter

3.7.1 Software Interface

Before tool proceeding tool measurement, select a calibration type you need by setting the value of parameter N80032. The default for calibration type is fixed calibration. Then press SP1 to access functional area **General** and then press F4 to open the corresponding calibration type dialog box, as shown in Fig. 3-18.

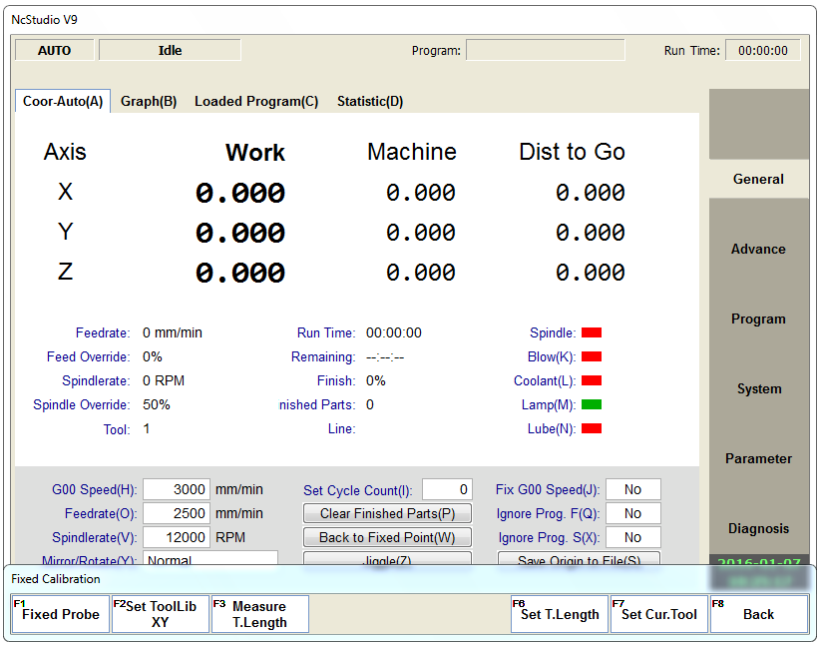


Fig. 3-18 Dialog Box “Fixed Calibration”

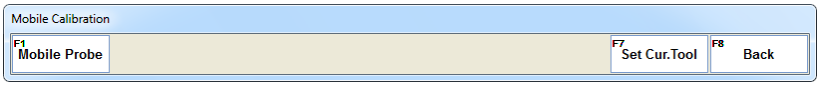


Fig. 3-19 Dialog Box “Mob Calibration”

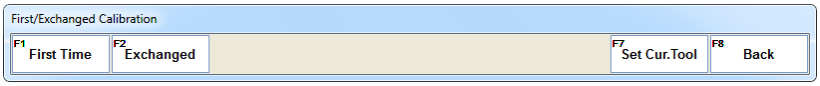


Fig. 3-20 Dialog Box “First/Exchanged Calibration”

- **Fixed Probe, Mobile Probe First Time, and Exchanged**

Press F1 for different calibration types to execute tool measurement. In dialog box “First/Exchanged Calibration”, **F2 Exchanged** can only be pressed after executing **F1 First Time** operation.

- **Set ToolLib XY**

In dialog box “Fixed Calibration”, press F2 to set the current X and Y position in tool lib as the machine coordinate of X/Y axis. After you press F2 and verify manufacturer’s access, another dialog box asking “Is sure to load x and y position in tool lib?” will pop out. Select “Yes” to confirm or “No” to cancel.

- **Measure T.Length**

In dialog box “Fixed Calibration”, press F3 to measure the length of tool. The measurement result will be set to the tool offset of current toll No. Then

- **Set T.Length**

In dialog box “Fixed Calibration”, press F6 to set the machine coordinates of current Z axis as the current tool length. A dialog box will pop out asking “Whether to set current mc to tool length?” Select “Yes” to confirm or “No” to cancel.

- **Set Cut.Tool**

Press F7, an input box will pop out. You can input tool No. and then press F1 to confirm.

- **Back**

Press F8 to close the current calibration dialog box.

3.7.2 Mobile Calibration

Mobile calibration is used to set workpiece origin of Z-axis by executing calibration at the current position. The thickness of the mobile presetter is decided by parameter N75100. After mobile calibration, the system will calculate according to “Calibration result” and set it into the part offset.

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. 3-21 and Fig. 3-22 for the sketch map of the process of mobile tool calibration.

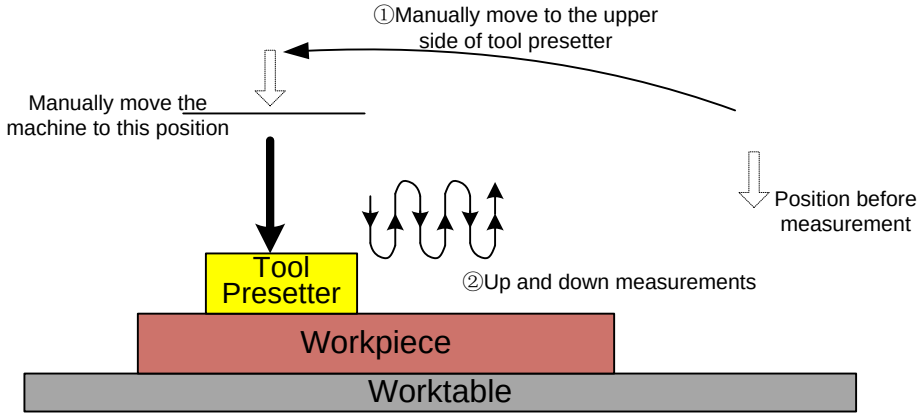


Fig. 3-21 The Process of Mobile Calibration without Encoder Feedback Function

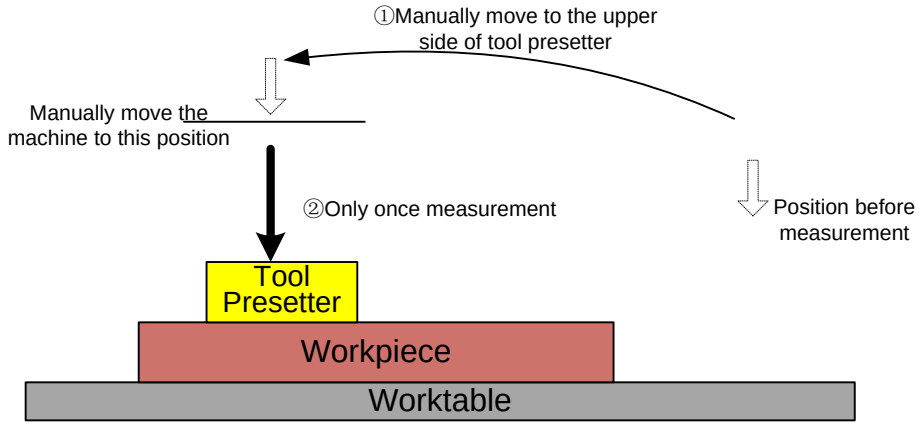


Fig. 3-22 The Process of Mobile Calibration with Encoder Feedback Function

- **Related Parameters**

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

Parameter		Details	Setting Range
	Thickness		
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	ToolMea Fine Speed	Tool speed when approaching the presetter surface in tool measurement	-
N75002	ToolMea Fine Time	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
N10050	Positive ToolMeas. Travel Limits	Machine coordinate of upper limit of worktable range in tool measurement	-99999~99999
N10055	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, otherwise measurement fails.			

3.7.3 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. 3-23 and Fig. 3-24 for the sketch map of fixed calibration.

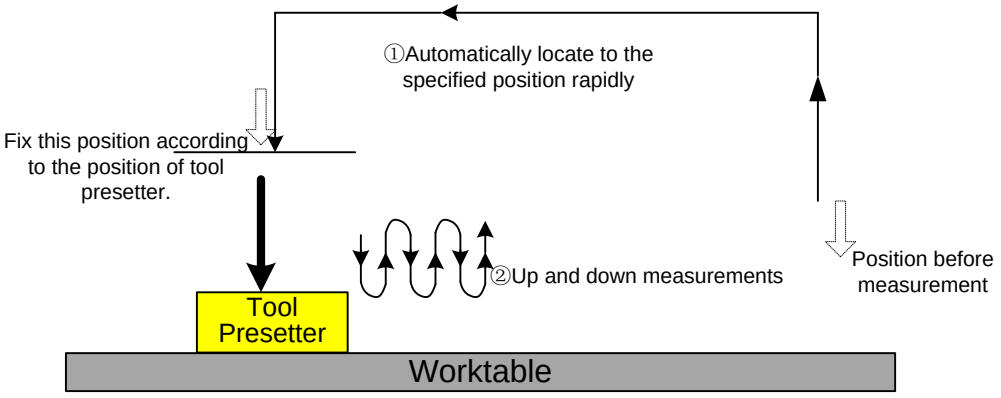


Fig. 3-23 The Process of Fixed Calibration without Encoder Feedback Function

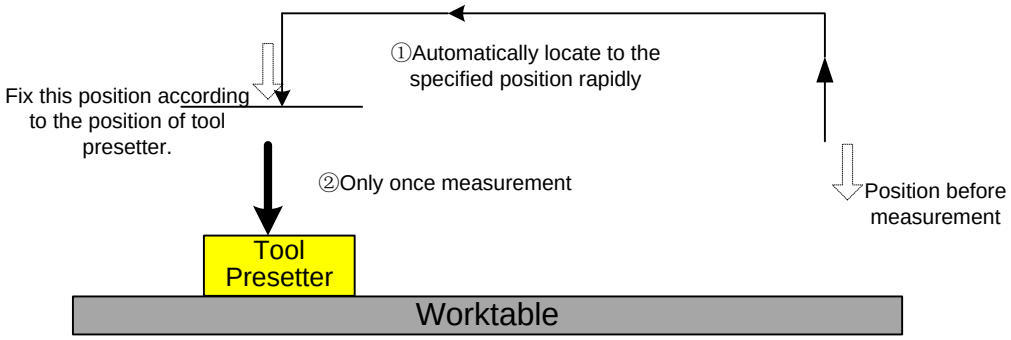


Fig. 3-24 The Process of Fixed Measurement with Encoder Feedback Function

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

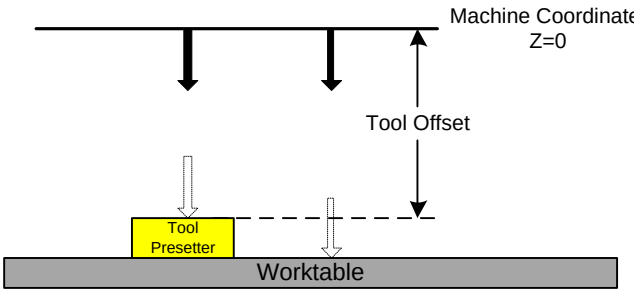


Fig. 3-25 The Sketch Map of Tool Offset

The steps of fixed calibration are as follows.

1) Select a tool according to tool No.;
2) Execute fixed calibration to the selected tool and record the tool offset.
3) Execute step 1 and 2 to each tool;
4) Select any tool to move to workpiece surface for clearing.

- **Related Parameters**

Parameter		Details	Setting Range
N75201	Fixed Probe Surface to WCS Z0	The distance from fixed probe surface to WCS Z0. It can be used for system calculating WCS Z0 automatically by cutting touching probe.	0 ~ 99999
N75203	Fixed Preset Speed	The speed that the tool moves from the highest point to the calibration-start point in fixed calibration.	0~99999
N75210	Fixed Tool Sensor Position	The machine coordinate of the fixed tool presetter	-99999~99999
N75025	Tool Mea Overtravel Alarm	Whether to execute the over-travel protection of tool calibration	Yes: enabled; No: disabled
The measurement method for parameter N75201 is as below: 1) Manually move the Z-axis to the certain point on workpiece surface→ shift down its tool nose until reaching the surface of workpiece→ record the current coordinate Z1 of Z-axis. 2) Uplift Z-axis→ put the fixed tool presetter on workpiece surface→ shift down Z-axis slowly until reaching the presetter and getting the calibration signal→ record the current coordinate Z2 3) Z2 - Z1, and the result equals to the thickness of fixed tool presetter. Manually enter this result into parameter N75201.			

For other related parameters about fixed calibration, such as N75001, N75002, N75020, N10050 and N10055, refer to section 3.7.2.

3.7.4 First/Exchanged Calibration

The operation steps are as below:

1) Firstly, manually move Z axis to workpiece surface, and then confirm the workpiece origin by floating presetting or manual clear. The method for manual clear: press F6 **Clear**, and then press F3 **Z Clear** in the new pop-up manipulation button bar.

2) Secondly, press F4 **ToolMea.**, and then press F1 **First Time** in the new pop-up manipulation button bar to execute first calibration. The system will record the current workpiece coordinate value of Z axis automatically, as shown in Fig. 3-26.The system will then complete the process automatically.

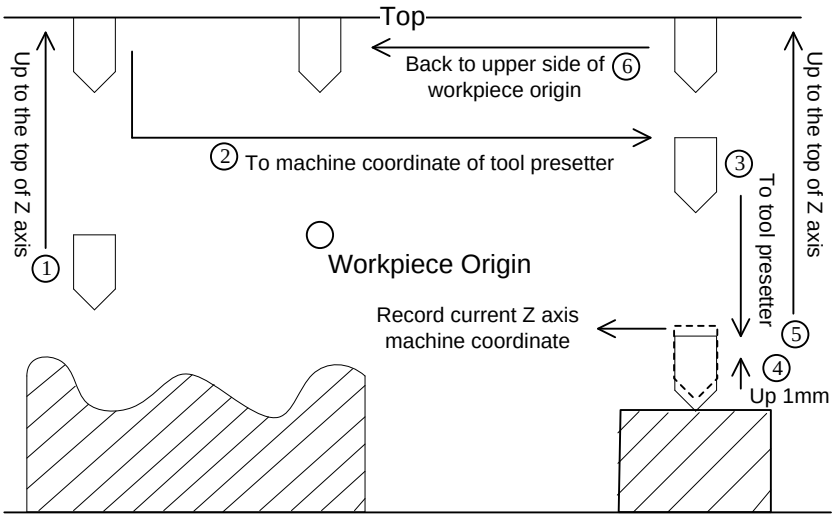


Fig. 3-26 First-time Calibration

3) Start machining after first-time calibration completed.

4) After tool change or tool break, press F4 **ToolMea.**, and then press F2 **Exchanged** in the new pop-up manipulation button bar to restore the workpiece coordinate value of Z axis of current point . The process is automatically completed by the system, as shown in Fig. 3-27.

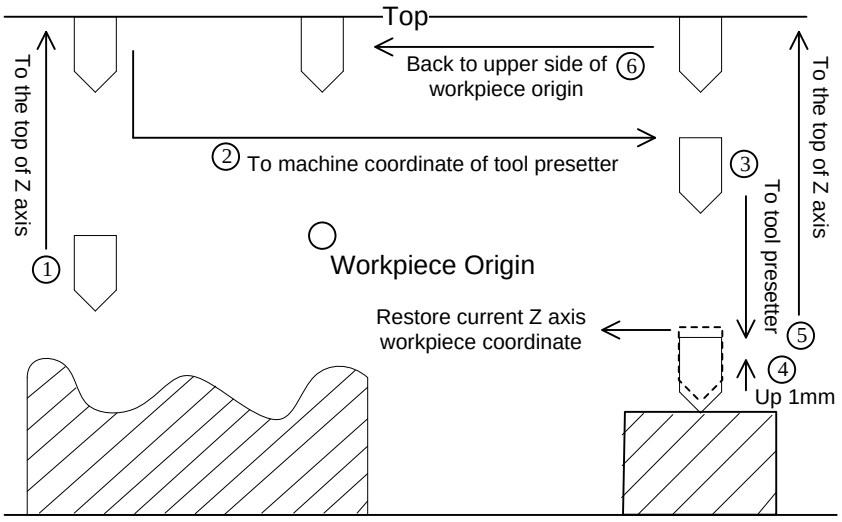


Fig. 3-27 Exchanged Calibration

5) Start machining after exchanged calibration process is completed.

3.8 Offset Setting of Workpiece Coordinate System

3.8.1 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. 3-28.

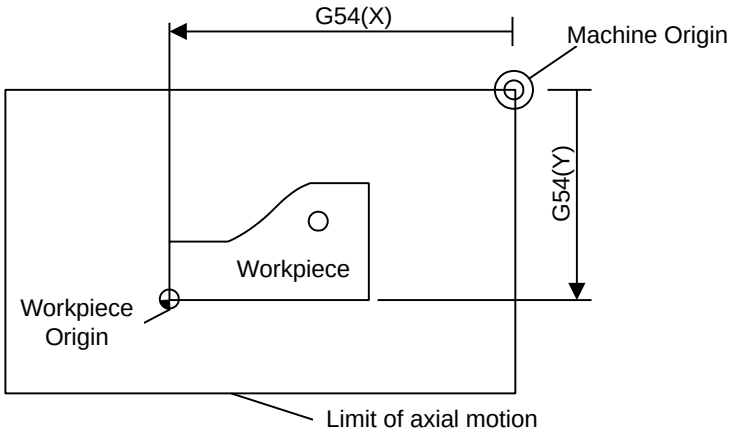


Fig. 3-28 The Relationship of Work Offset and Machine Coordinate System

One, two or multi-work offset can be used in machining program. As shown in Fig. 3-29, 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.

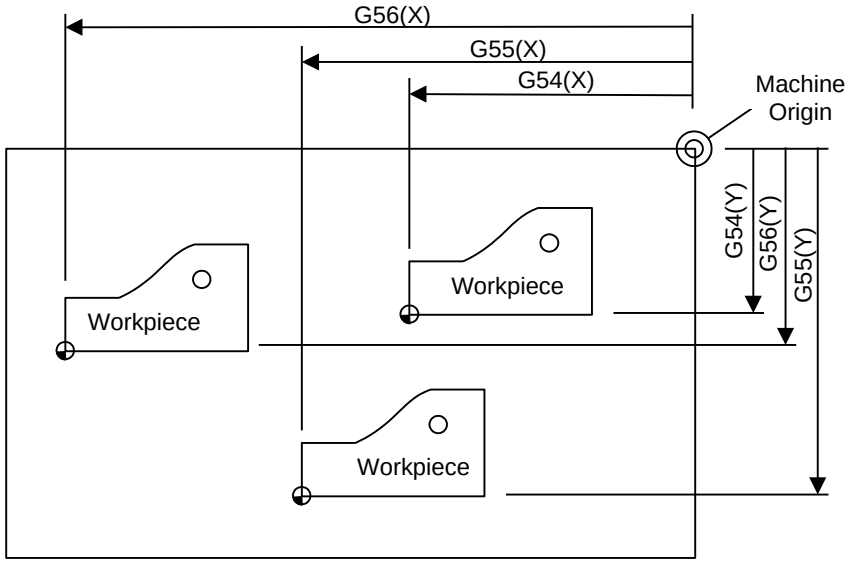


Fig. 3-29 Sketch Map

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

- **Related Parameters**

Parameter		Details	Setting Range
N72010	Enable Work Coordinate Limits(WCS)	Whether to check the bounds of the work-area in workpiece coordinate.	Yes: enabled; No: disabled
N72020	Negative Work Coordinate Limits	The lower limit of the work-area in workpiece coordinate.	-99999~ table travel upper limit (machine coordinate)
N72030	Positive Work Coordinate Limits	The upper limit of the work-area in workpiece coordinate.	Table travel lower limit (machine coordinate) ~ 99999
The default setting of N72010 is “No”, i.e. there is no need of checking workpiece coordinate limits. If users need to set N72010 to be “Yes” due to special needs, setting workpiece coordinates limits in N72020 and N72030 is needed.			

3.8.2 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 NK300 and 126 work offsets can be programmed.

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

3.8.3 Software Interface

Press letter key A to enter sub-screen of [coordinate system], shown in Fig. 3-30. You can see the currently edited workpiece coordinate and the related workpiece offset & public offset in this interface. The interface is shown in Fig. 3-31 after parameter N80002 is set as “Yes”.

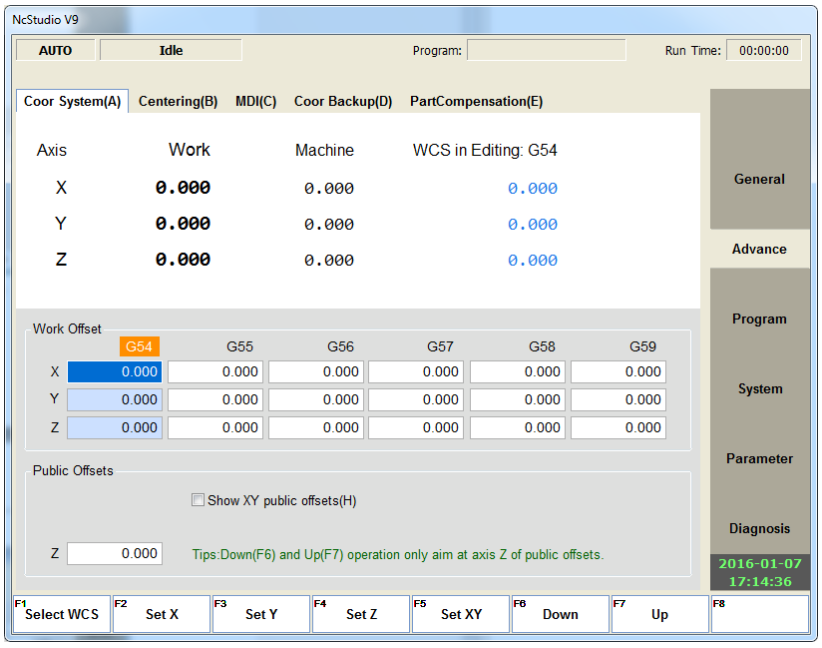


Fig. 3-30 Coordinate System Management Interface

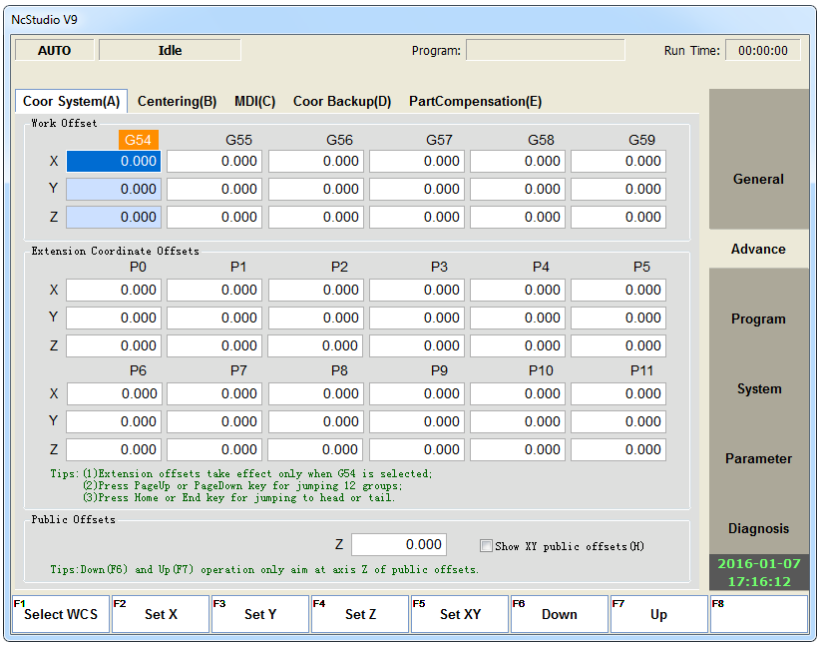


Fig. 3-31 Extended Coordinate System Management Interface

Press key " ↑ " and " ↓ "to move the cursor to the workpiece offset, extended offset or public offset, and then press Enter to modify the workpiece offset or public offset. And you can also press F6 or F7 to modify Z-axis public offset. When the option "show XY public offsets" is not selected, only the public offset of Z-axis can be modified under **Public Offsets** area.

Currently, the supported workpiece coordinate systems in NK300 are G54, G55, G56, G57, G58, and G59. The extended coordinate systems are based on G54, from G54P0 to G54P119. Press key "PgUp" and "PgDn" to turn pages, and "Home" or "End" to the page heading or footing.

The manipulation buttons under this function screen will be introduced as follows.

- **Select WCS**

You can press key "←" and "→" in interface **Coor System** to select the specified WCS, and then press F1 to set currently edited coordinate system as current WCS.

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

Press key 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 remain the same.

- **Set XY**

Press F5 to set workpiece offset of X and Y workpiece coordinates in the current WCS as the value of current machine coordinate, while the related machine coordinate of Z will remain the same.

- **Down**

Press F6 and an input box will pop up. You can input the adjustment of Z-axis in the box and press F1 to confirm. The workpiece origin on Z-axis will move down a specified distance. Press F6 successively, and the moving distance will be the accumulated value of several adjustments.

- **Up**

Press F7 and an input box will pop up. You can inpu the adjustment of Z-axis in the box and press F1 to confirm. The workpiece origin on Z-axis will move up specified distance.

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

3.8.4 Related Parameters

Parameter		Details	Setting Range
N80002	Support Extension Workpiece Offset	Show coordinate system page which contain 120 groups extra-workpiece offsets under the advanced function.	Yes: support; No: do not support
The default parameter setting is “No”. When clamping above 6 workpieces on workbench is needed, please set this parameter as “Yes” to support the extended coordinate systems of workpiece offset, so as to save multi-group of workpiece offsets, which is more user-friendly.。			

3.9 Centering

The system supports manual and auto centering. In auto centering, the spindle does not rotate. Manual centering is divided into two-point centering and circle centering. In manual centering, whether to enable spindle speed in manual centering is decided by two parameters.

When the parameter N81004 “Allow Spindle-ON when centering” is set to “YES”, press F5 **Center Start** before centering and the button will turn orange, meaning the spindle ON, before executing manual centering. Spindle speed is decided by the parameter N20006 “CenterSpindlerev”, whose value is 500 by default and should not be set small.

When the parameter N81004 “Allow Spindle-ON when centering” is set to “NO”, function **Center Start** is disabled. To make spindle ON, press “Spindle CW” or “Spindle CCW” at spindle speed set in the software.

3.9.1 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.

Press SF2 to enter functional area **Advance**, and press letter key B to enter interface **Centering**. The default centering type is **Centering**, as shown in Fig. 3-32.

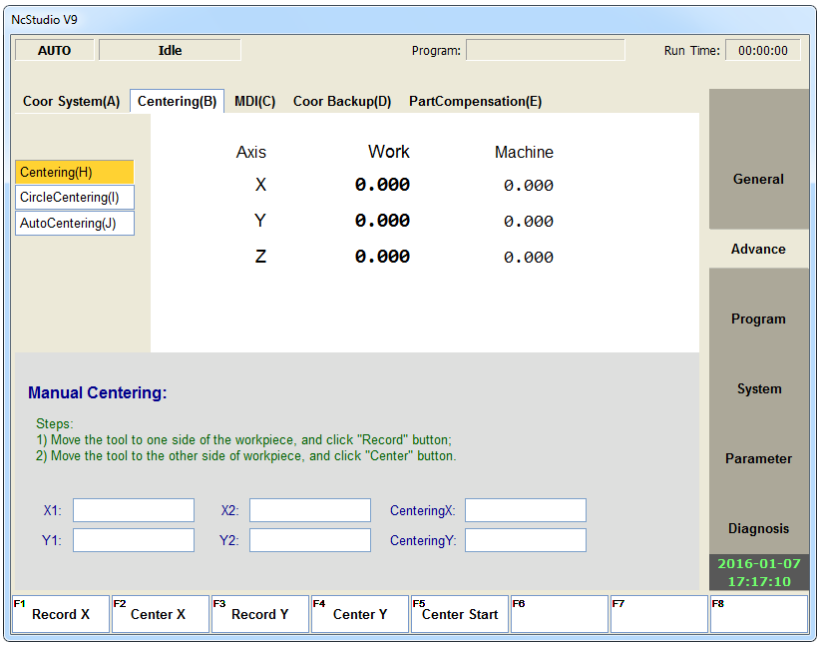


Fig. 3-32 Dialog Box "Line Centering"

The operation steps of line centering are as follows (An example of X-axis).

1) In HW mode, manually move the cutter to one side of workpiece, and then press F1 **Record X** to record the machine coordinate of current point.
2) Move the cutter to the other side of workpiece, and then press F2 **Center X** to calculate the midpoint coordinate based on the coordinate of current position and last recorded value and set it as workpiece origin.

CAUTION

In the process of centering of a certain axis, the other coordinate axis should keep unchanged.

3.9.2 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.

Press SF2 to enter functional area **Advance**, and press letter key B to enter interface **Centering**. Then press letter key I or directly click **CircleCentering (I)** to enter sub interface circle centering, as shown in Fig. 3-33.

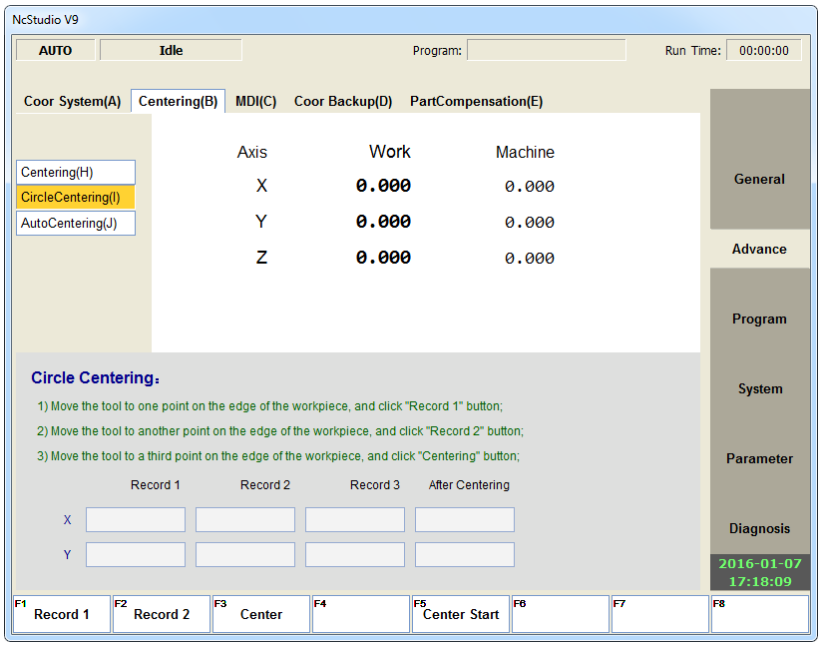


Fig. 3-33 Dialog Box “Circle Centering”

The steps of circle centering are as follows.

1) Manually move the cutter to one point on the circumference of a circular blank, and then press F1 **Record** 1 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 2** 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 **Center** 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.

3.9.3 Auto Centering

Auto centering is used for moving cutter to workpiece center. And inner center is for locating cavity shaped workpiece center.

Press SF2 to enter functional area **Advance**, and press letter key B to enter interface **Centering**. Then press letter key J or directly click **AutoCentering (J)** to enter sub interface auto centering, as shown in Fig. 3-34.

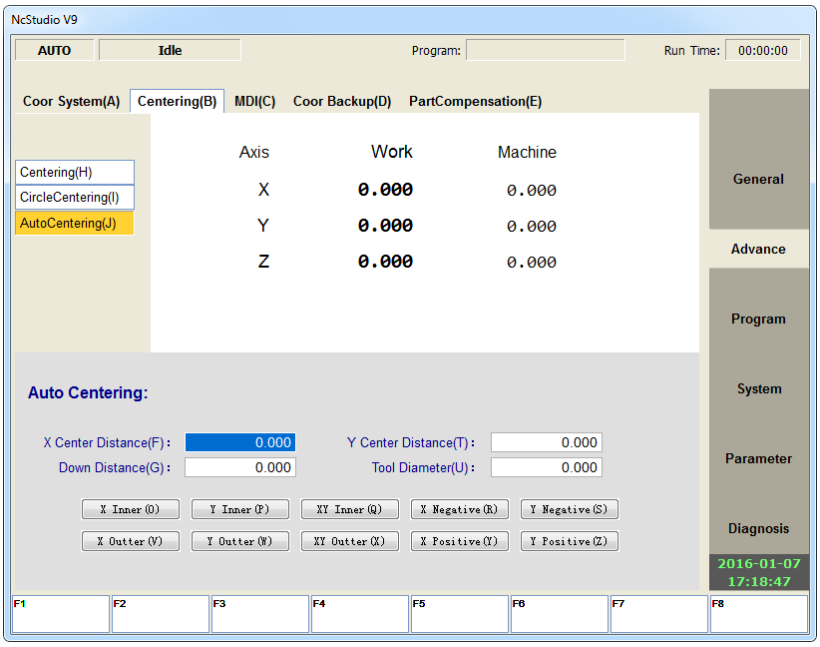


Fig. 3-34 Auto Centering

X Center Distance: the pre-estimated distance from midpoint of workpiece to X boundary must be a little larger than its actual value in outer center, while a litter smaller in inner center.

Y Center Distance: the pre-estimated distance from midpoint of workpiece to Y boundary must be a little larger than its actual value in outer center, while a little smaller in inner center.

Down Distance: tool descending / lifting distance in tool presetting. Down distance in inner center must be smaller than the distance from tool nose to workpiece surface, and Down Distance in outer center should be larger than the distance from tool nose to workpiece surface.

Tool Diameter: actual diameter of the tool.

- **The Process of X Outer Center (the same as those of Y outer center and XY outer center)**

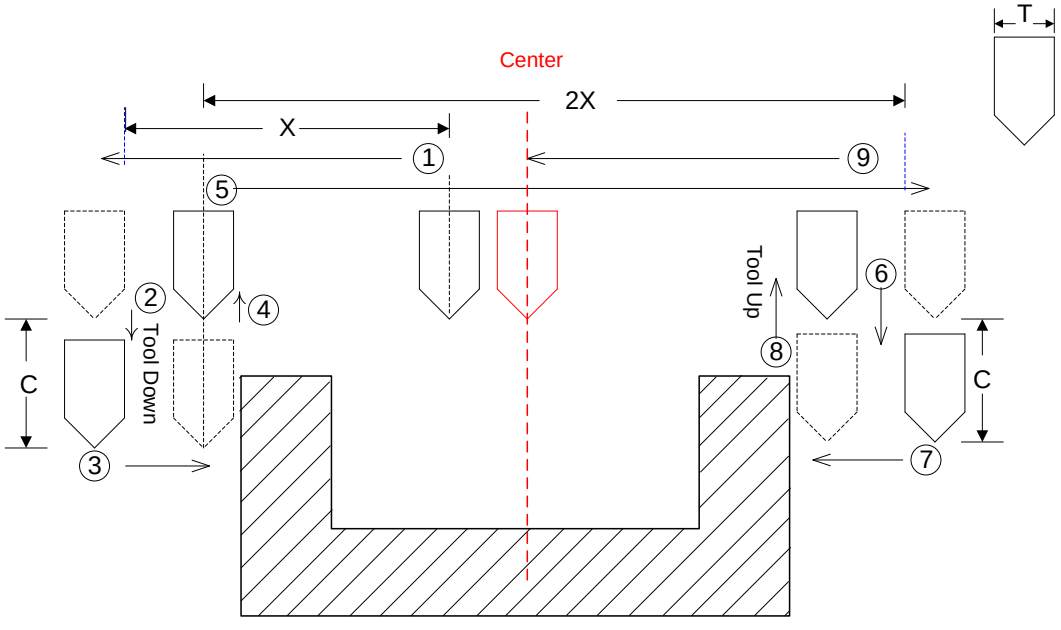


Fig. 3-35 The process of X outer center

Place the conducting workpiece (copper, iron, aluminum) on the insulated workbench, and connect it to the port CUT on terminal board, while cutter to COM port on terminal board.

Put the cutter over the predicated center point position before performing automatic centering, and then press letter key V to execute “X Outer” to make cutter move “X Center Distance”, shift down “Down Distance” and translate towards workpiece a short distance until reaching the conducting workpiece so as to conduct the circuit and transfer the signal, and system will automatically record the current axial coordinate X1 (Y1).

Then cutter will raise “Down Distance”, then move two “Center Distance”, then move down “Down Distance”, and then translate towards workpiece a short distance until reaching the conducting workpiece so as to conduct the circuit and transfer the signal, and system will automatically record the current axial coordinate X2 (Y2) to calculate the coordinate of center point of workpiece and then move the cutter to this center point.

- **The Process of X Inner Center (the same as those of Y inner center and XY inner center)**

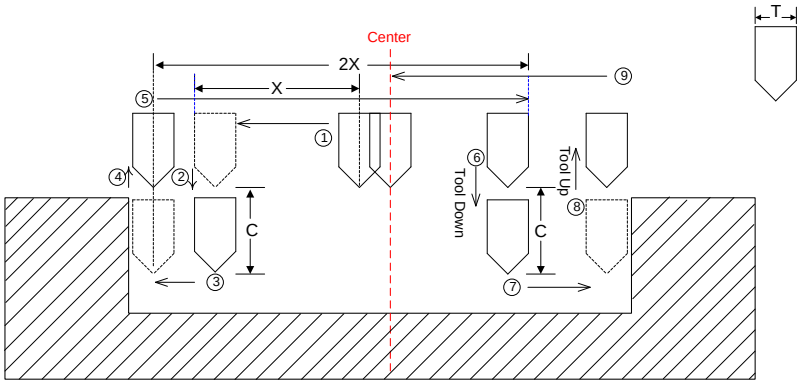


Fig. 3-36 Sketch Map of X Inner Center

CAUTION

1) Before automatic centering, the tool nose must be over the predicted center position, and the value of center distance, down distance and tool diameter must be specified in advance.

2) The inner center distance must be smaller than workpiece radius, while the outer center distance must be larger than the workpiece radius.

- **Boundary Presetting**

Boundary presetting refers to setting the boundary point as workpiece origin, involving positive/ negative boundary presetting. The process is as shown in Fig. 3-37.

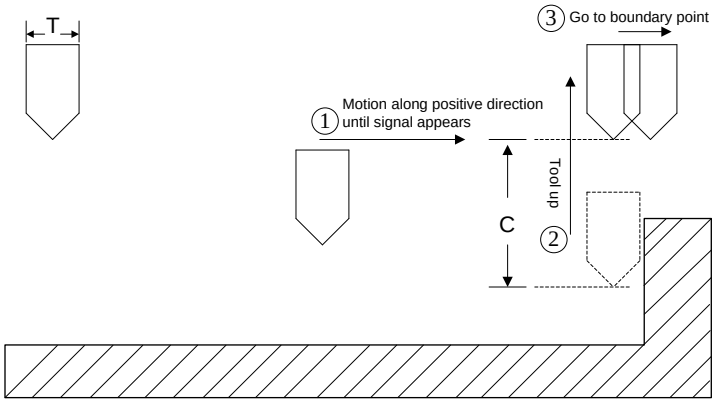


Fig. 3-37 Process of Boundary Presetting

3.10 Adjustment of Speed & Acceleration

3.10.1 Feedrate Setting

Feedrate can be set directly in the software.

In the interface **Coor-Auto** of functional area **General**, feedrate can be directly specified on the parameter setting section above the manipulation button bar, as shown in Fig. 3-38: when “Ignore Prog. F” is set as “Yes”, the feedrate will adopt the system setting value, i.e. the value of “Feedrate”; when it is set as “No”, the feed rate will adopt the value specified in machining files.

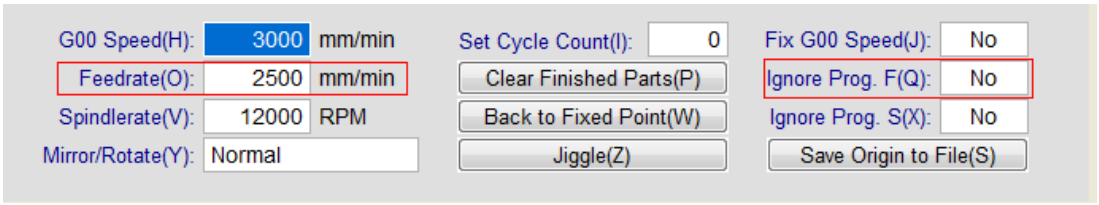


Fig. 3-38 Parameters Setting Zone-feedrate Setting

There are two methods to set the parameter:

1) Press “↑” or “↓” to move cursor to the corresponding parameter, and then press Enter to open the input box.
2) Press the corresponding shortcut key behind the desired parameter to open the input box. Take “Feedrate(O)” as an example, by pressing letter key O, you can open the input box for setting feedrate.

Feedrate is also related to current feedrate override, so it can be controlled by adjusting the current feedrate override, and the formula is as follows.

$$\text{Current feedrate} = \text{Setting feedrate} \times \text{Current feedrate override}$$

Feedrate override knob is on the operation panel, as shown in Fig. 3-39.

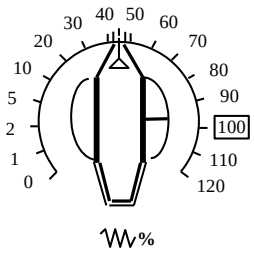


Fig. 3-39 Feedrate Override Knob

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

3.10.2 G00 Speed Setting

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

Similar to feed speed, G00 speed can also be set directly in the software, as shown in Fig. 3-40. When “Fix G00 Speed” is set “Yes”, the running speed of machine tool under G00 code is fixed by the value set in “G00 Speed”; when “Fix G00 Speed” is set “No”, the running speed of machine tool under G00 code is not fixed. And if G00 speed is set in the processing file, this value will be adopted by the system for running; if not, the value of “G00 Speed” will be adopted.

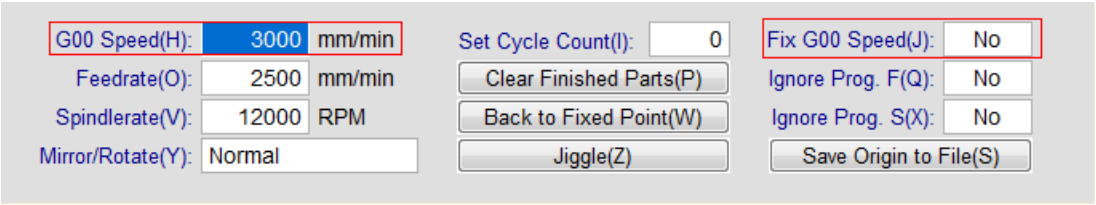


Fig. 3-40 Parameters Setting Section –G00 Speed Setting

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

3.10.3 Parameter Specification

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

1) Speed

- **Related Parameters**

Parameter		Details	Setting Range
N64000	Startup Speed	The initial velocity motors start running, normally for stepper motors avoiding low speed.	0~100000
N13000	Max Axis Velocity	Maximum velocity reachable for each axis.	0.01~100000
N71000	Slow Jog Speed	There are two kinds of speed for option under manual mode: jog speed (Slow Jog	0 ~ Rapid Jog Speed

Parameter		Details	Setting Range
N71001	Rapid Jog Speed	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.	Slow Jog Speed~ Max Feedrate of each axis
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. 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”. 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.			

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
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)	0.001~100000
Axial acceleration is used to describe the acceleration/ deceleration capability of each feed axis, with unit “mm/s^2”, depending on the physical feature of machine, such as quality of motion part, torque, cutting load and resistance of the feed motor. The larger the value of parameter is, the less time the machine will spend in acceleration/ deceleration during motion process, the higher the efficiency is. Generally, for servo motor system, it should be within “600 ~ 3500”. Set a smaller value at first, and repeatedly execute typical motion for a period of time. If there is no abnormal situation, gradually increase the value. If abnormal condition occurs, reduce the value, with “50% ~ 100%” insurance allowance. Angular for corners refers to the max. acceleration of feed motion on adjacent axes, and “1 ~4” times of the “Axis Acceleration” is recommended, generally within “1200 ~ 5000”. For higher speed requirement, “2 ~ 4” times of the “Axis Acceleration” is recommended. “Axial Jerk” refers to growth rate of acceleration, i.e. the increment of acceleration in unit time, with			

Parameter	Details	Setting Range
unit "mm/s^3". It is available for S_type and LEP_type of acceleration & deceleration, used to mitigate the bad effect caused by abrupt acceleration & deceleration of machine.		

3) Reference Circle and Circular Speed Limit

- **Related Parameters**

Parameter		Details	Setting Range
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; 3: Acceleration trapezoid algorithm
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			

Parameter	Details	Setting Range
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.		

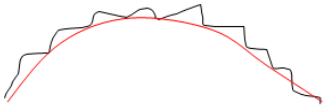
5) Smooth Setting

- **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
N64240	Smoothing Time Factor	Ratio of Smoothing speed interval to a control cycle.	0.01~10
N64241	ConnectSpeed decreased at MaxConnectAngle	Whether the machine connect speed decreased at MaxConnectAngle.	YES: enabled; NO: disabled
N64245	Prepared number of path for optimizing performance	Segments for performance optimization, having no effect on the result of velocity planning.	1~2000
N64246	Slide speed for small lines	Eliminate speed wave for small lines.	YES: enabled; NO: disabled
N64247	Reference length of slide speed for small lines	Adjust the speed wave for the line whose length is shorted than reference length.	0.001~10
N64248	Velocity slide optimization	Optimizing the path smoothing and the Acc or Dcc handling after interpolation, improving efficiency.	YES: enabled; NO: disabled

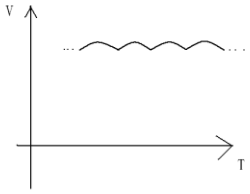
Parameter	Details	Setting Range

"Smoothing Time" works as post-acceleration/ deceleration. The larger the value is, the more ambiguous the details of workpiece are, i.e. the workpiece is more smooth. But it will lead to reduction of arc radius in machining arc. And it will also dwarf wave peak in machining workpiece resembling waves, as following. The range within 0.05s is recommended.

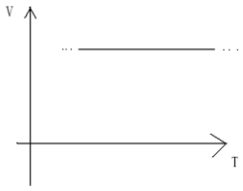


In machining an arc (or other curves) composed of short segments, velocity fluctuation, like frequent acceleration and deceleration obvious in our S-type algorithm, will occur at places where curvature is relatively large, as follows:

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:



In this way, frequent acceleration and deceleration is avoided, oscillation of a machine tool is eliminated, and machining quality is improved.

3.11 Simulation & Track

3.11.1 Simulation

The function of simulating provides a fast but lifelike simulated processing environment for users.

Running under the mode of simulating, the system will not drive the machine tool to do the relative actions but only show the processing trace of the cutter in high speed in the trace window. By simulating, you can see moving form of the machine tool in advance, avoiding machine tool damage due to programming mistakes in processing procedure. And they can also know other additional information.

The steps of simulation are as below:

1) Press SF3 to enter functional area **Program**, and then press F1 **Load** after selecting a machining file in interface **Local Disk**;

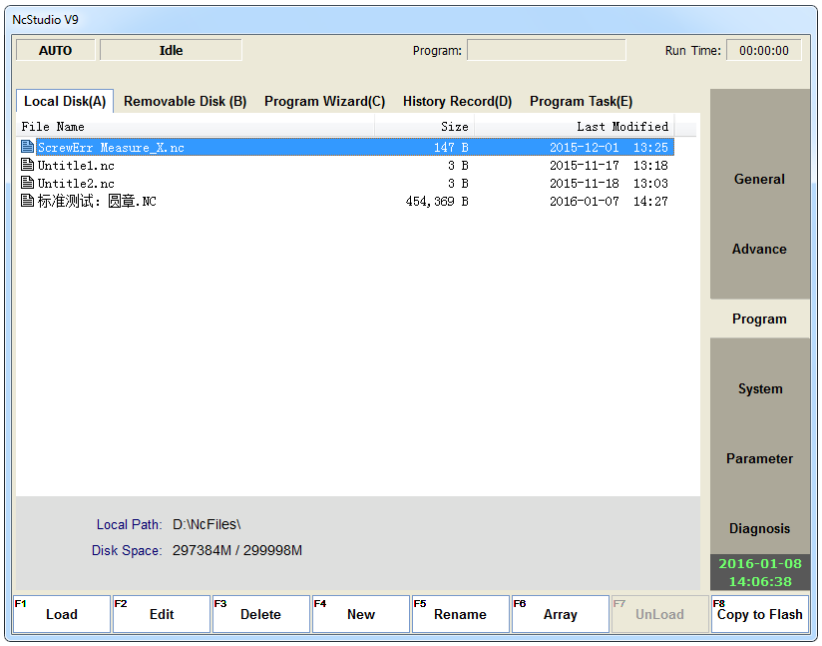


Fig. 3-41 Interface Local Disk

2) Press SF1 to enter functional area **General** and then F1 **Simu On** to begin simulation, and then press letter key B to enter interface **Graph** to view the simulated track, as shown in Fig. 3-42.

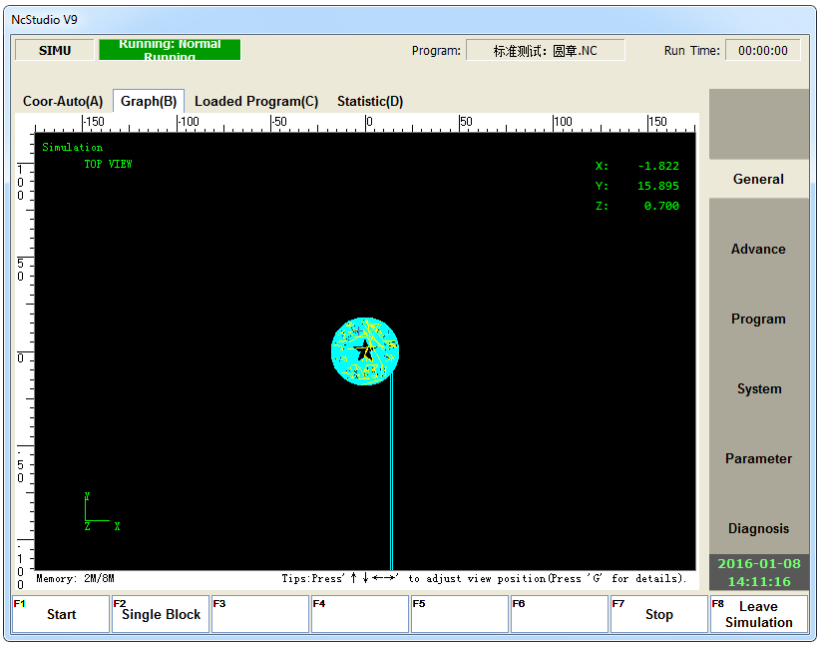


Fig. 3-42 Interface Graph in Simulation Mode

3) System begins simulation, and you can view the overall machining path of cutter in interface **Graph**. And additional information such as “Bounds X/ Y/ Z”, “machining time” can be read after you press F7 **Stop** in interface **Loaded Program**, as shown in Fig. 3-43.

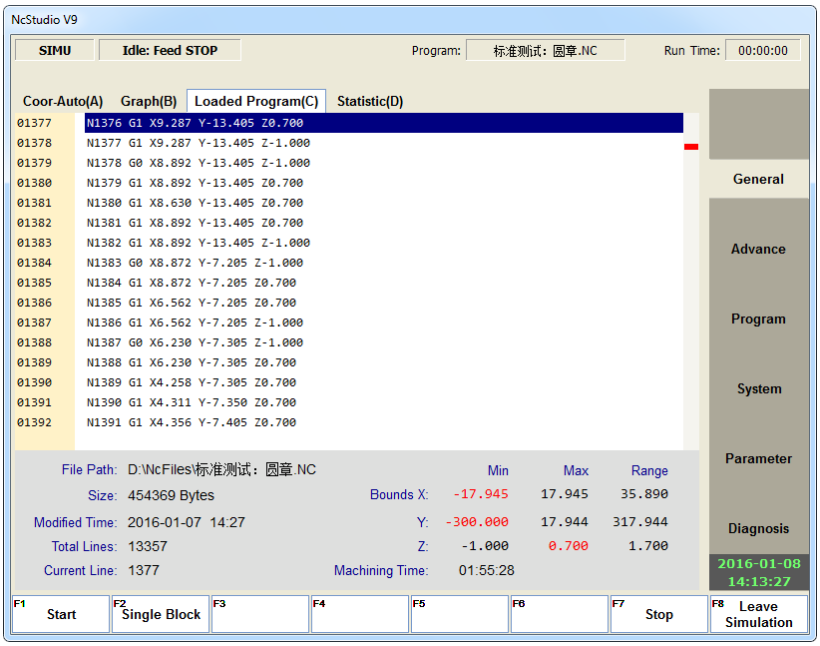


Fig. 3-43 Interface Loaded Program in Simulation Mode

The default setting of **Simu On** is **Start**, and **Single Block** is for option, i.e. you can set the single block mode for the machining task to be executed, facilitating error diagnosis and failure recovery.

You can press F7 **Stop** to stop simulation or F8 **Leave Simulation** to exit.

3.11.2 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.

Press letter key G in interface Graph, a view adjustment box will appear in the center of the screen, as shown in Fig. 3-44, in which you can choose the appropriate angle of view to see the processing track. As a result, you can check the tool path processed more conveniently and detect the processing state.

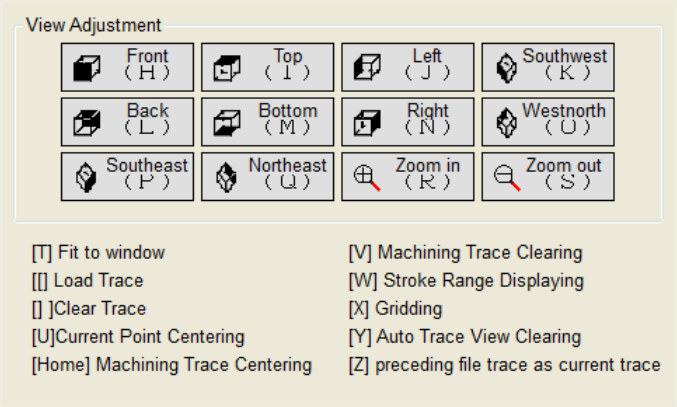


Fig. 3-44 View Adjustment

3.11.3 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.	YES: analyze; NO: do not 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	YES: use; NO: do not use
N81011	Draw Workbench	Setting whether to draw the boarder of the worktable in the track window	YES: draw; NO: do not draw
N81012	Draw grid	Setting whether to draw grid in graph interface	YES: draw; NO: do not draw
N81013	2D Mode	Setting whether to use 2D mode to view the track in the track window	YES: use; NO: do not use
N81015	Clear on Loading	Setting whether to clear the contents of the	YES: clear;

Parameter		Details	Setting Range
		current view when a new file is loaded	NO: do not clear
N81016	Draw WC Origin	Setting whether to display workpiece origin in the track window	YES: display; NO: do not display
N81017	Draw MC Origin	Setting whether to display machine origin in the track window	YES: display; NO: do not 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

The parameters above are related to modifying the property of interface **Graph**.

3.12 Compensation

3.12.1 Screw Error Compensation

- **Related Parameters**

Parameter		Details	Setting Range
N12000	Screw Error Comp	It sets whether to enable screw error compensation and decides compensation type.	0: no compensation; 1: single compensation; 2: double compensation
N12001	Enable Backlash Compensation	It sets whether to enable backlash compensation.	Yes: enabled; No: disabled
1. The setting range for parameter N12000 is: 0 (no compensation); 1 (single compensation); 2			

(double compensation).

2. The setting range for parameter N12001 is: Yes (enable backlash compensation; No(disable backlash compensation).

To enable only backlash compensation, set N12000 as “0” while N12001 “Yes”.

To enable single compensation

1) Use “Err Pos.” data and backlash value in **Screw Comp** interface to compensate, and set N12000 as “1” and N12001 as “Yes”.

2) If backlash compensation does not exist, use only “Err Pos.” data in **Screw Comp** interface to compensate, and set N12000 as “1” and N12001 as “No”.

To enable doublel compensation, i.e. to compensate by reading “Err Pos.” (forward error) and “Err nEG.” (backward error) data in “Leadscrew Comp” interface, set N12000 as “2”.

To disable compensation, set N12000 as “0”, and N12001 as “No”.

3.12.1.1 Software Interface and Operation

Press SP5 to enter functional area **Parameter** and then press letter key E enter interface **Screw Comp**. When parameter N12000 is set as “0” or “1”, the interface is as shown in Fig. 3-45; when set as “2”, the interface is as shown in Fig. 3-46.

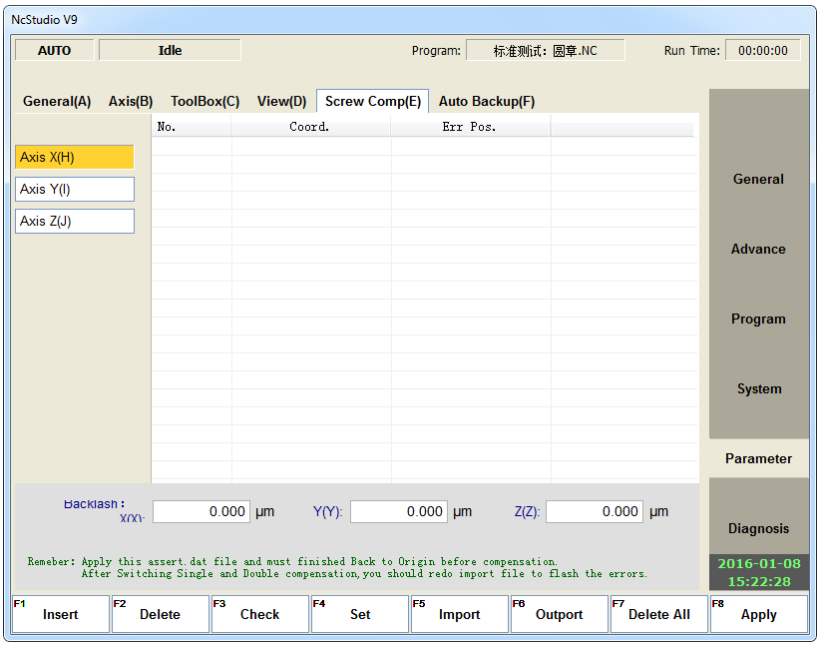


Fig. 3-45 Single Screw Compensation

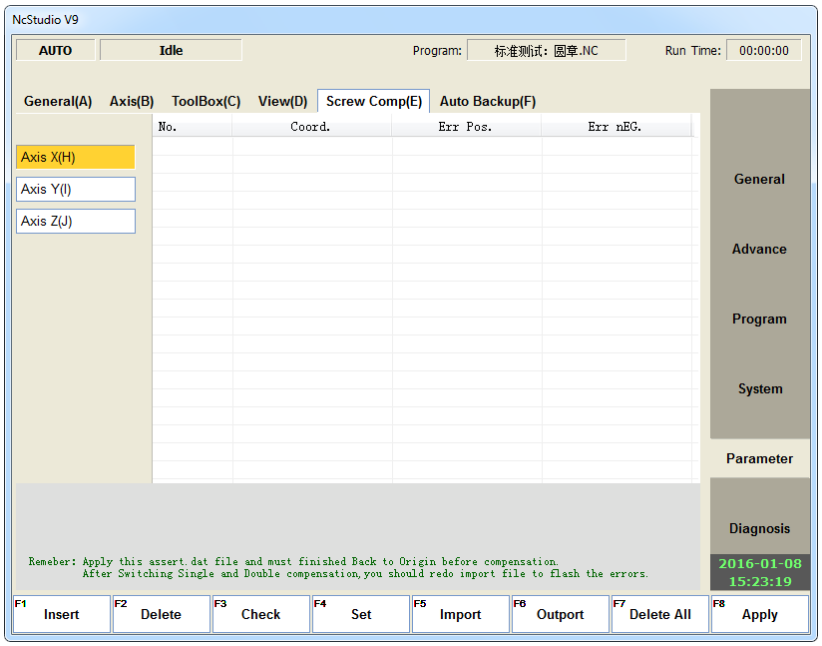


Fig. 3-46 Double Screw Compensation

Single 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.

Double 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**

Coor. (Position coordinate): mm

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

- **Import**

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

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**

Press F7 to delete all data.

- **Apply**

After you press F8, 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.

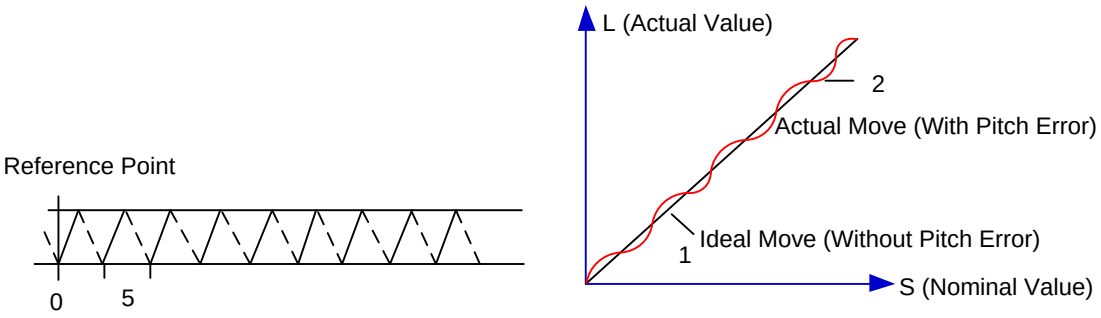
3.12.1.2 Causes of Screw Error and Compensation Method

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. 3-47(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. 3-47(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

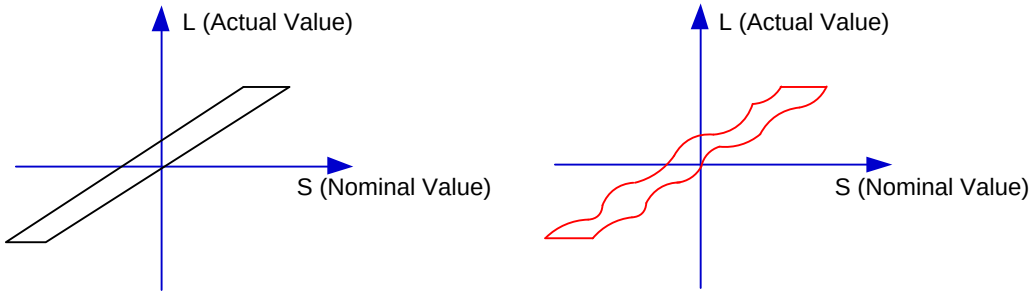
Fig. 3-47 Analysis of Pitch Error

- **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.

- **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. 3-48(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. 3-48(B).



(A) Hysteresis Feature (B) Actual Moving Curve

Fig. 3-48 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).

3.12.1.3 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. 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 3-1 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 3-1 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, or the compensation is invalid.

Item	Specification
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.

1. 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 >

2. 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 3-2 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.

3.12.2 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 interface **ToolBox**, if tool nose radius is altered after tool wear, tool sharpening or tool change, avoiding the trouble to modify the programmed processing procedure. The interface is as shown in Fig. 3-49.

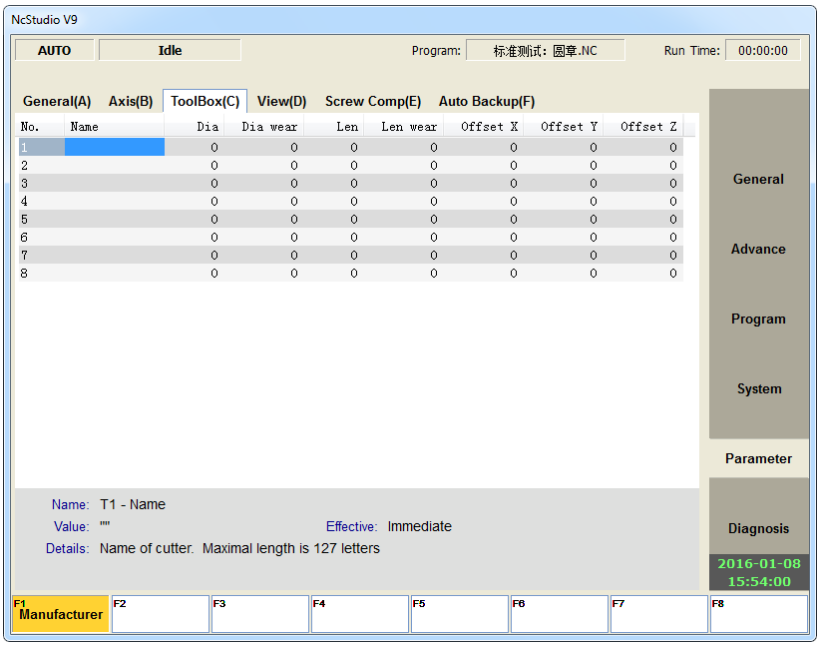


Fig. 3-49 The Interface of ToolBox

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/G01 to make tool compensation.

- **Related Parameters**

Parameter		Details	Setting Range
N62410	Enable Cutter Compensation	Setting whether to perform tool compensation	YES: enabled; NO: disabled
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; 2; 3
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	YES: force; NO: do not force

Parameter		Details	Setting Range
		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.	
Interference refers to error phenomena, such as over-cut and wrong machining direction, caused by too large tool radius. The parameter N62413 decides interference detection among how many 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.			

3.12.2.1 Tool Radius Compensation

Tool radius compensation code can make the tool moved by the offset value, as shown in Fig. 3-50.

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.

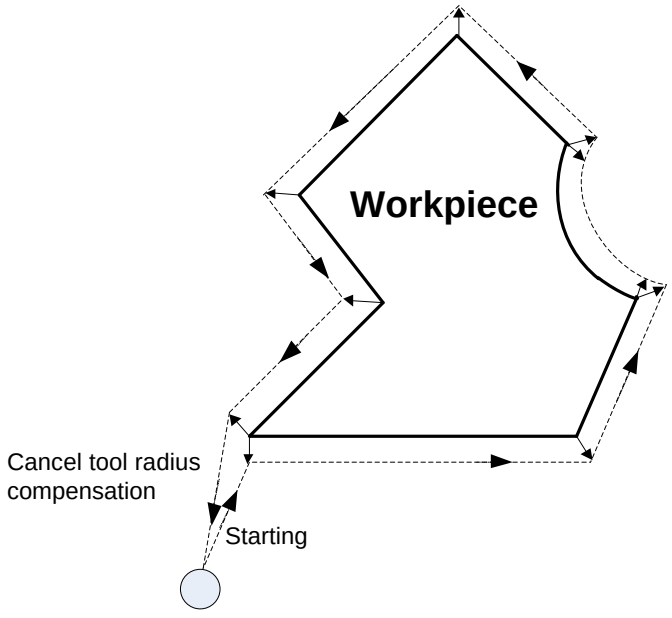


Fig. 3-50 Schematic Diagram for Tool Radius Compensation

3.12.2.2 Types of Establishing Tool Compensation

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, establishment of tool compensation includes two establishing segments, as segment 1 and 2 in Fig. 3-51.This software offers 3 types of establishing 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.

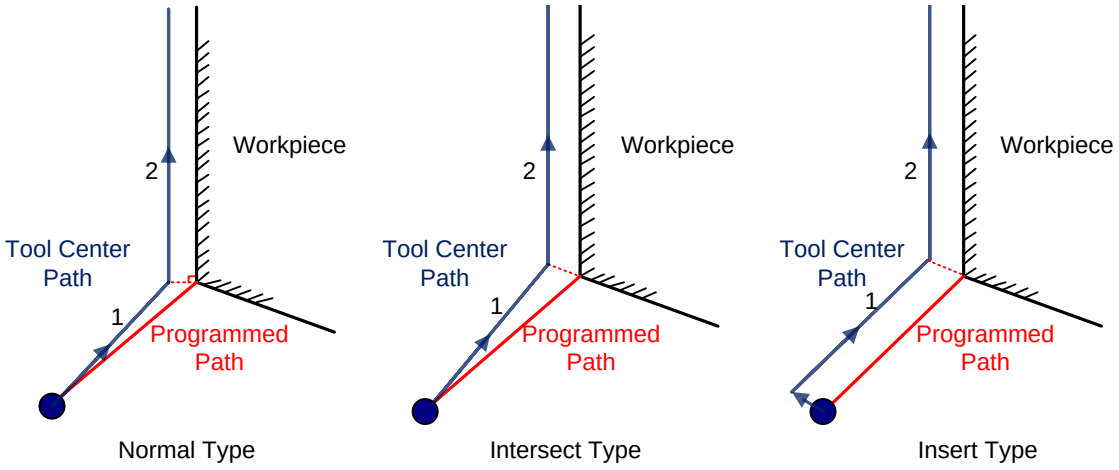
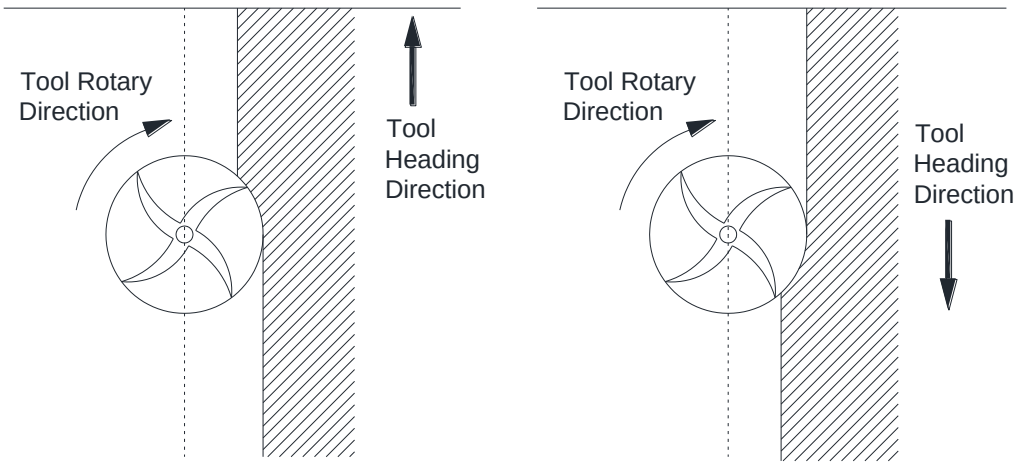


Fig. 3-51 Types of Establishing Tool Compensation

3.12.2.3 Tool Compensation Direction

The schematic diagram of tool compensation direction is as shown in Fig. 3-52.



(a) Left Compensation (b) Right Compensation

Fig. 3-52 Direction of Tool Compensation (A: Left Compensation; B: Right Compensation)

Programming for tool radius compensation is as shown in Fig. 3-53:

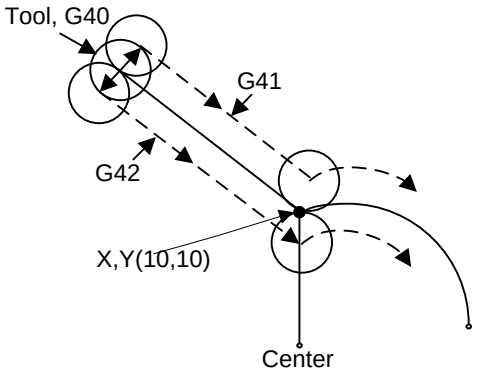


Fig. 3-53 Schematic Diagram of Tool Compensation Machining

```
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*.

3.12.3 Across Quadrant Error Compensation

Across quadrant error compensation refers to the distortion, the most commonly seen is the closed angle, on the conversion part of two adjacent quadrants in circle machining of machine tool. To eliminate this kind of distortion, error compensation is necessary.

Across quadrant compensation parameter is used for the closed angle compensation when processing arc passes quadrants. The setting method along positive and negative directions of X/Y/Z is similar.

- **Related Parameters**

Parameter		Details	Setting Range
N12020	Turn On AQE Compensation	Turn On the across quadrant error compensation for circular	YES: enabled; NO: disabled
N12030	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.	0~0.1
N12031	Distance(Group 0)	-	0~999
N12032	Delay(Group 0)	-	0~0.1
N12033	Intensity(Group 0)	-	0.01~0.99

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).

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.

3.13 Log and Diagnosis

3.13.1 Log

Press SF6 to access functional area **Diagnosis**, and then press key A to open interface **Log**.

The interface **Log** 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. 3-54.

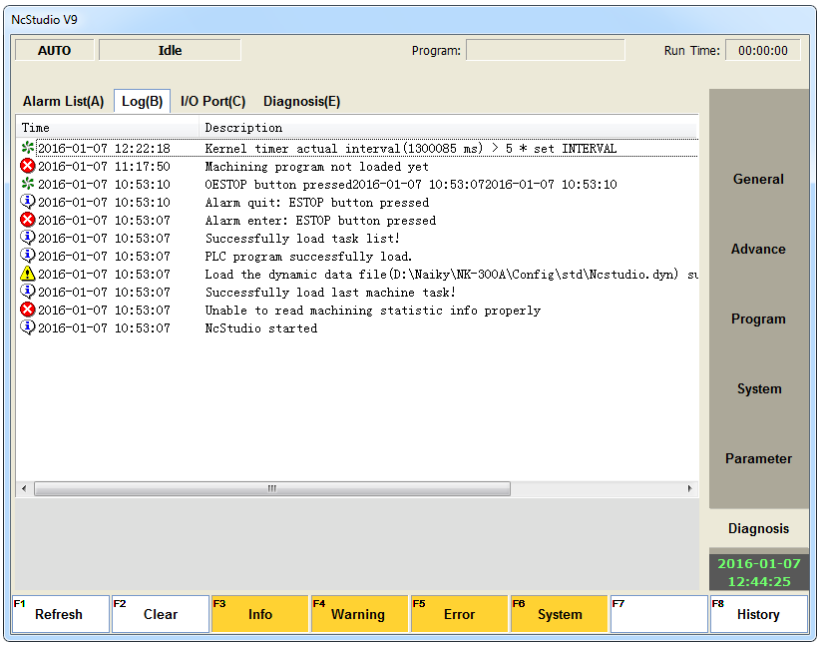


Fig. 3-54 Interface Log

- **Refresh/Clear**

Press key “F1” to refresh the log list in order to make it synchronize with the system. Press key “F2” to remove all the current log information.

- **Info, Warning and Error**

The shortcut keys are F3, F4 and F5 respectively for function **Info**, **Warning** and **Error**.

The default state is checked and highlighted in orange, namely the system displays normal information, warnings and error info by default. If you don’t need certain info displayed, you can press the corresponding shortcut key to eliminate the highlight. For example, you can press F5 to make the button bounced and the system will hide the error info.

- **System**

Press key F6 to view the system info, which need password.

- **History**

Press key F8 to display all the logs since recording.

3.13.2 Diagnosis

Press SF6 to access functional area **Diagnosis**, and then press key E to open interface **Diagnosis**.

The interface **Diagnosis** 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. 3-55.

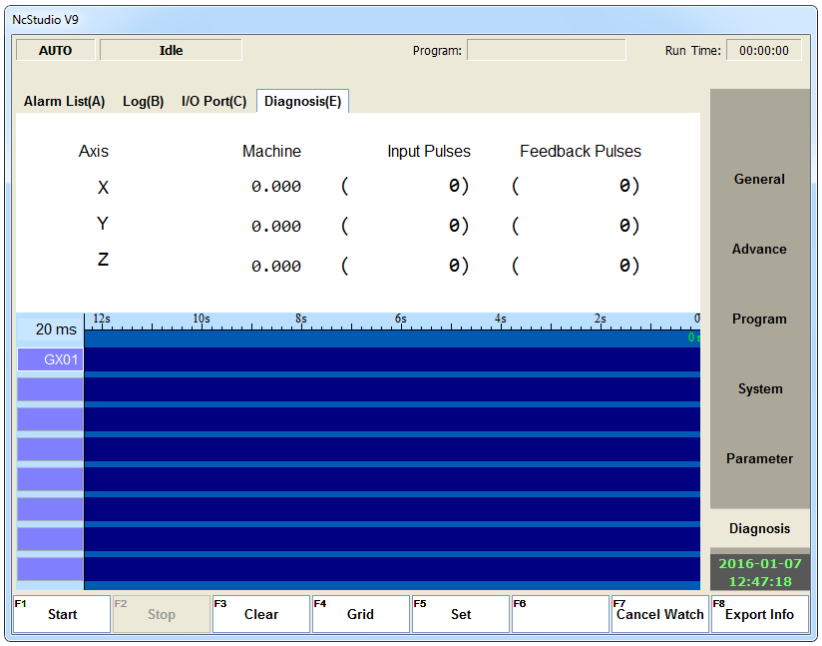


Fig. 3-55 Interface Diagnosis

- **Start/Stop/Clear**

Press F1/ F2 to start/stop diagnosing the corresponding port. Press F3 to clear the diagnosis result of the corresponding port.

- **Grid**

Press F4 to bring grid lines into the sampling window.

- **Set**

Press F5 to set the sampling interval, as shown in Fig. 3-56.

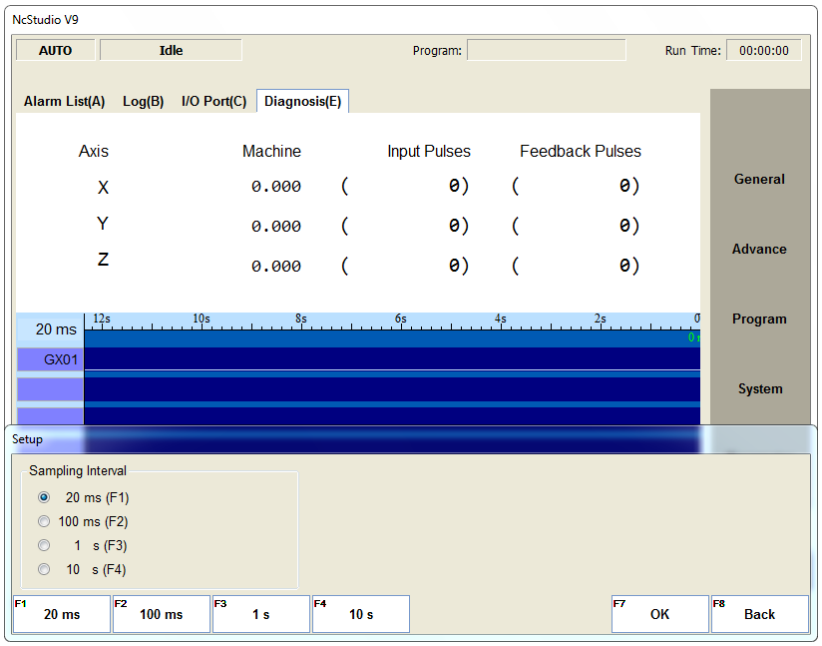


Fig. 3-56 Dialog Box of Setting Sampling Interval

Select a sampling interval in range (20ms, 100ms, 1s and 20s), and press F7 to confirm. And then back

to interface Diagnosis, and press F1 to start sampling periodically the corresponding port or PLC address, realizing tracking detection of the port.

- **Cancel Watch**

Press F7 to cancel the monitoring of the corresponding port.

- **Export Info**

Press F8 to export the system information to D:\Naiky\NK-300A\Config\std (the folder name varies with different system configuration, the folder will be “rev” if the system adopts “rotary table” configuration) or to the removable disk if inserted.

3.14 Program File Management

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

3.14.1 Machining Wizard

NK300 offers 4 basic machining program wizards: circular contour, circular pocket, rectangular contour, and rectangular pocket. 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 SF3 to enter functional area **Program**, then press letter key C to enter [Program Wizard] screen, and then press shortcut key H to switch to Circular Contour screen, as shown in Fig. 3-57. To achieve the desired results, users can set parameters for the selected processing graph, such as milling inner frame or outer frame (milling inner frame mills the region inside the frame, and milling outer frame mills on the frame), workpiece diameter, initial (workpiece) coordinate X/Y (of processing), layer depth (of each cutting), engraving depth (of several accumulated cutting) and cutter diameter.

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

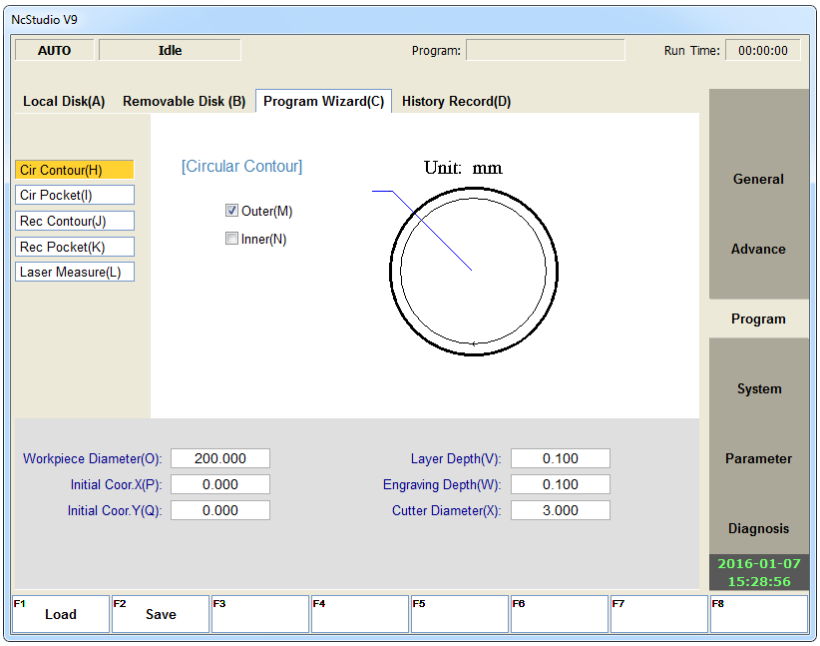


Fig. 3-57 Interface Circular Contour

3.14.2 Statistic

Press SF1 to enter functional area **Program** and then press letter key D to enter interface statistics. The main function of statistics screen is to display the statistics of processing file being processed and processed. As shown in Fig. 3-58, the upper part of this screen displays processing statistics of current machining file, including loaded program, start time, total parts, finished parts, runtime, part run time, part total len, and finished len. After a new program is loaded, all the statistics will be cleared.

The lower part of the screen displays the statistics of processed files, including program (name), start time, total time, total length and count.

Press F1 to you can clear the selected history statistics record in the list.

Press F3 to save current processing statistics as a new processing statistics file with default name “PartStat.txt” and saved path “D:\Naiky\NK-300A\Config\std”. the folder name of the last layer varies with system configurations. Currently being processed file cannot be saved to statistics in the lower part of the screen, while processed file will be automatically saved to statistics in the lower part of the screen.

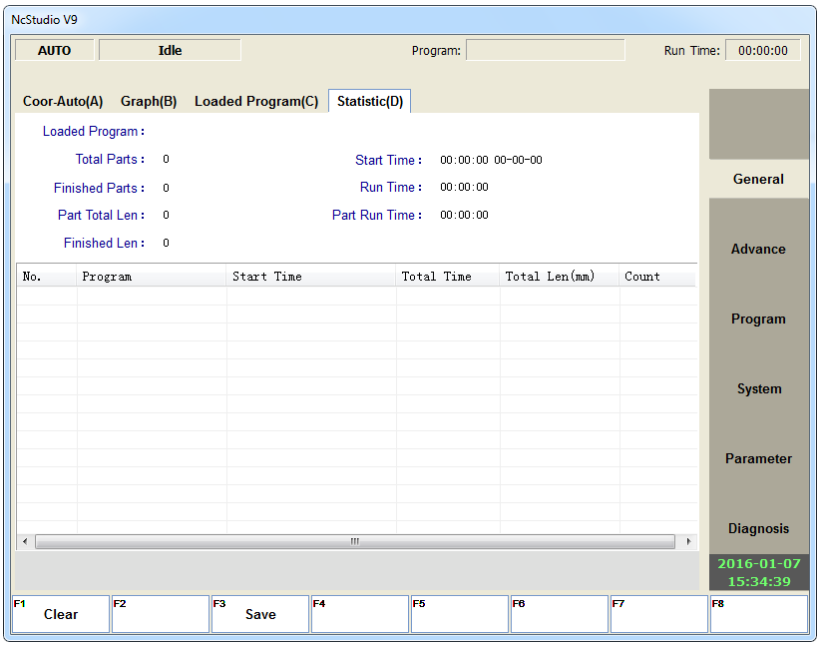


Fig. 3-58 Interface Statistics

3.14.3 Program File

Press SF3 to enter functional area **Program**, and interface Local Disk, Removable Disk, and History Record will be introduced as follows.

3.14.3.1 Local/Removable Disk

Press letter key A to enter interface **Local Disk**, as shown in Fig. 3-59. On the upper part of the interface, there is file list box displaying the processing files under the path D:\NcFiles. On the lower part, the prompt box shows the path of the currently selected file and available space of driver. Folders can be opened by pressing Enter.

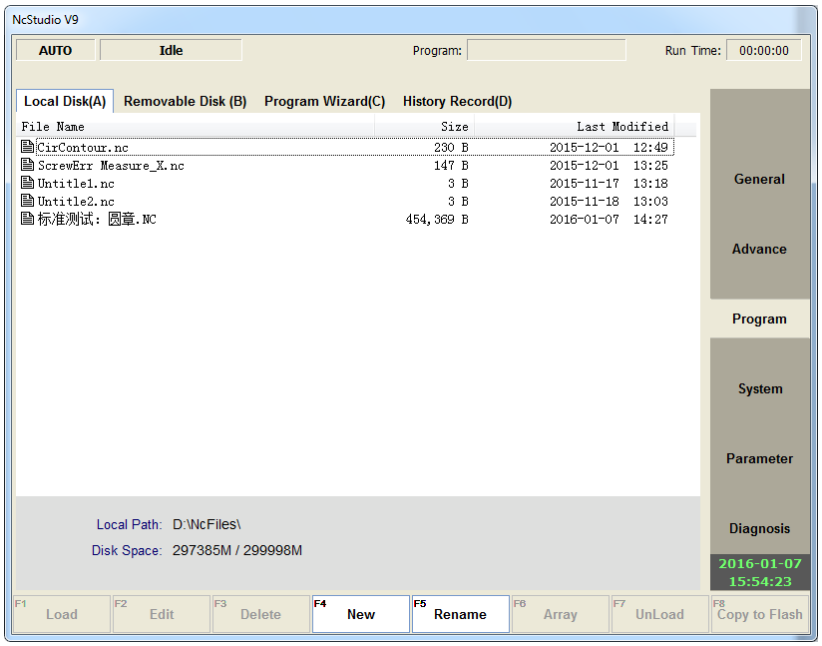


Fig. 3-59 Interface Local Disk

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.

- **Load**

After the processing file is selected (press key " ↑ " or " ↓ " to move cursor to the target file), press F1 will to make the system load and check the file automatically, information bar displaying the loading progress bar on the upper part. If there are errors in the program loaded, information prompt bar will display the specific error information. After loading finishes, other operations are available.

- **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**.

CAUTION

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

- **Delete**

After selecting a file, press F3, a prompt box asking whether to delete the file will pop up, as shown in Fig. 3-60.

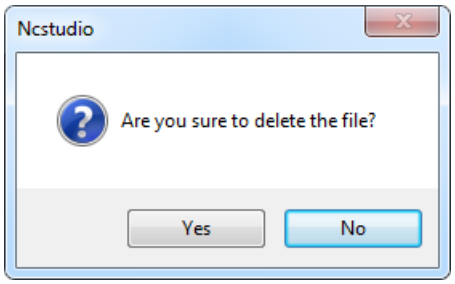


Fig. 3-60 Dialog Box Delete

CAUTION

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

- **New**

Press F4, 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 F5. A file name input box will pop up. After entering the new name, press F1 to complete the operation.

- **Array**

This function executes array machining for a machining file. Press F6 to access the dialog box, as shown in Fig. 3-61.

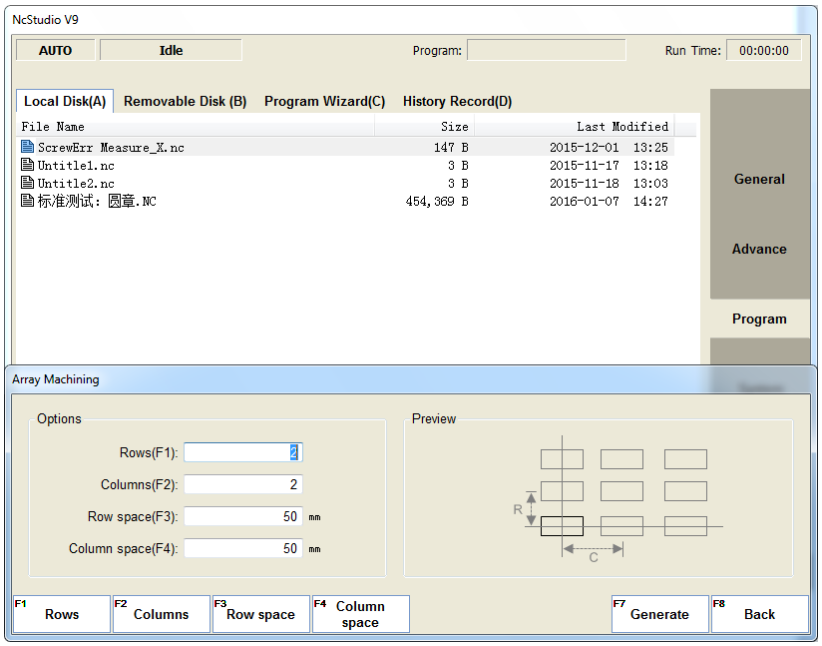


Fig. 3-61 Array Machining

Correctly set "Rows", "Columns", "Row space" and "Column space" by pressing corresponding shortcut keys. After finishing array setting, press F7 "Generate". Then you can name the set array machining file in the input box popping out, as shown in Fig. 3-62. And press F1 to confirm. The new file will be

displayed in the program list box.

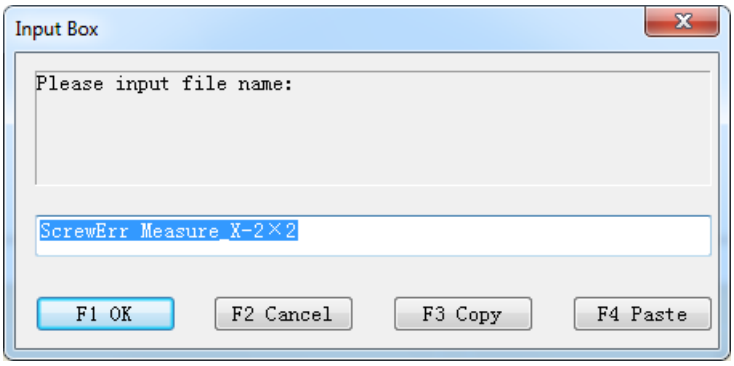


Fig. 3-62 Input Array Machining File Name

To cancel setting, press shortcut key F8 to return to the previous mode.

- **Unload and Copy to Removable Disk**

Press F7 to unload the currently being processed file, corresponding to the operation **Load**.

Press F8 to copy the file selected to the removable disk after inserting a removable disk.

Press letter key B to enter interface Removable Disk, as shown in Fig. 3-63, in which files in the USB flash disk can be read. For example, the file in the picture is under Naiky file in the USB Disk. You can also do the following operations to it, such as **Load**, **Edit**, **Delete**, **New**, **Rename**, **Copy to Local**, **Array** and **Unload**.

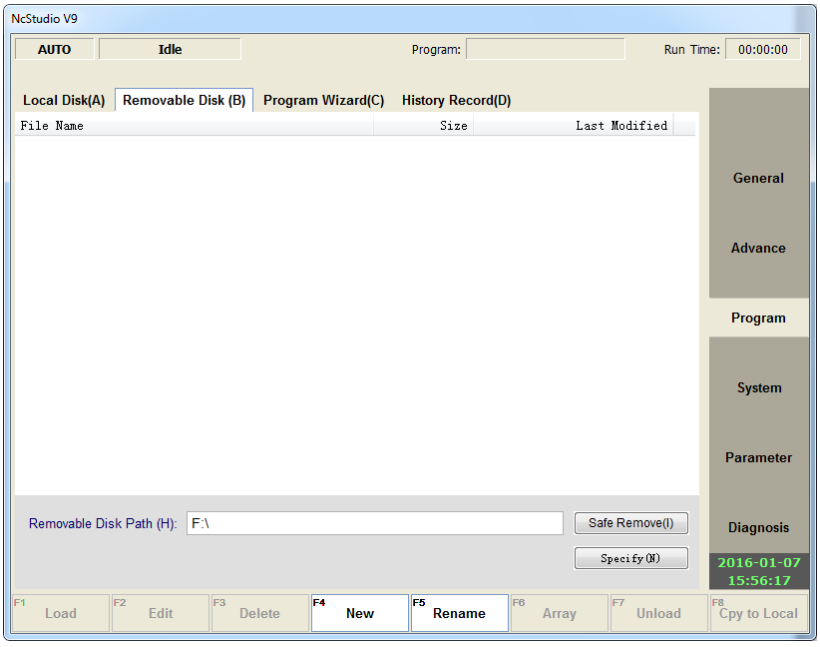


Fig. 3-63 Interface Removable Disk

The layout of program list in removable disk is the same as that in local disk, so is its operation, which will not be introduced here. After finishing using external storage, press letter key I to cut off the connection between the system and the external storage. At this time, the external disk can be removed safely. If there is a file in the external storage being processed or loaded into buffer zone, this button will be disabled.

Press letter key N to visit the shared files by other hosts or computers in the same LAN, as shown in Fig. 3-72.

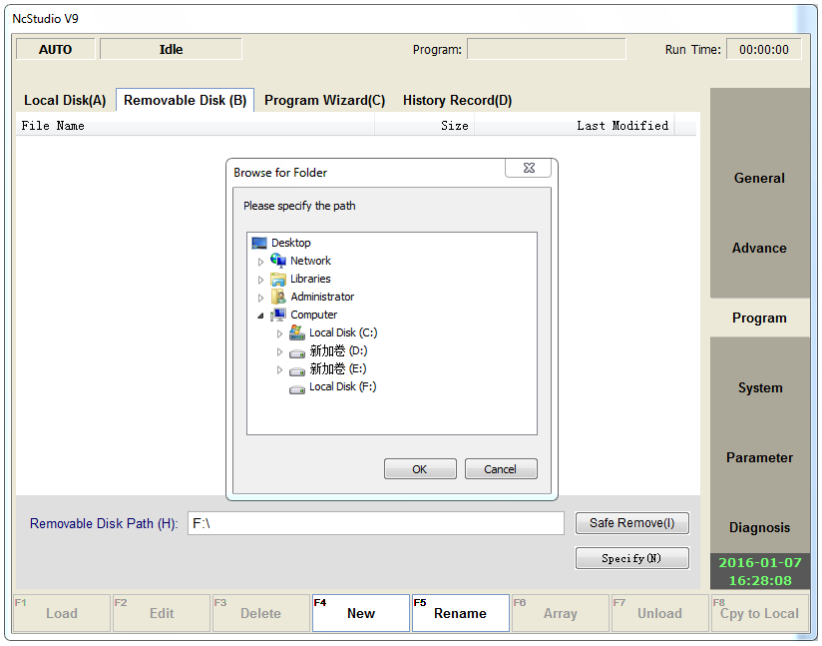


Fig. 3-64 Interface Specify Path

3.14.3.2 History

Press letter key D to enter interface **History Record**, as shown in Fig. 3-65. To replace the current machining file with a recently processed file, you can switch to the interface, in which all the recently loaded processing files are recorded. You just need to press the manipulation button corresponding to the desired processing file to load it and be ready for processing.

The system can record at most 8 processing files. When the file number exceeds 8, the file saved earliest will be overwritten.

The method to delete all the history records is to press H and then click “OK” in the pop-up prompt box.

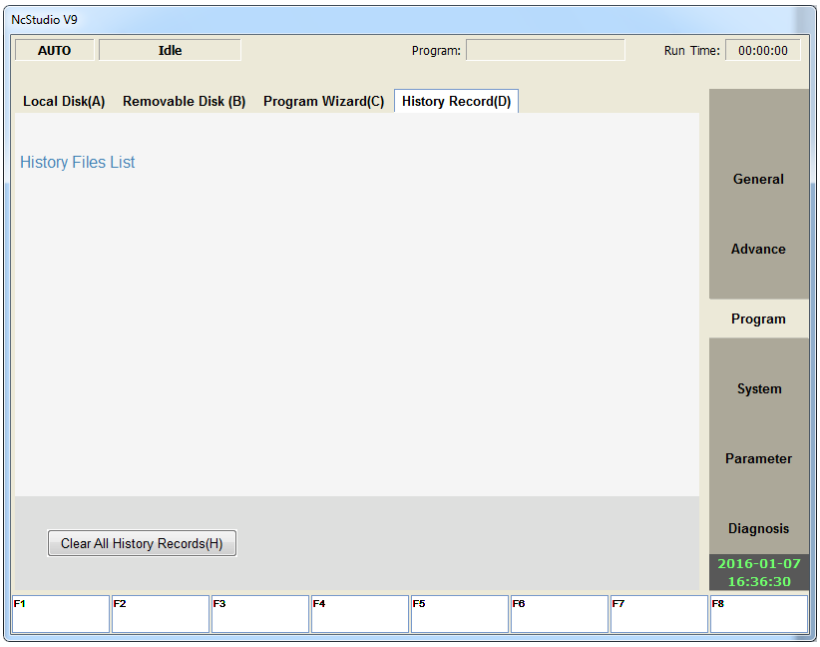


Fig. 3-65 Interface History Record

3.14.3.3 Program Task

You can decide whether interface **Program Task** shows in functional area **Program** by modifying the value of parameter N80003. When the parameter is set as “YES”, interface **Program Task** will shows as Fig. 3-66.

The interface consists of current edit task area and file list area. The current edit task area shows file order, file name, coordinate system and time interval. The file list area shows file name, file size and last modified time. In this interface, you can create tasks in which there is one or more machining files to machine the workpiece, saving you from loading machining files several times.

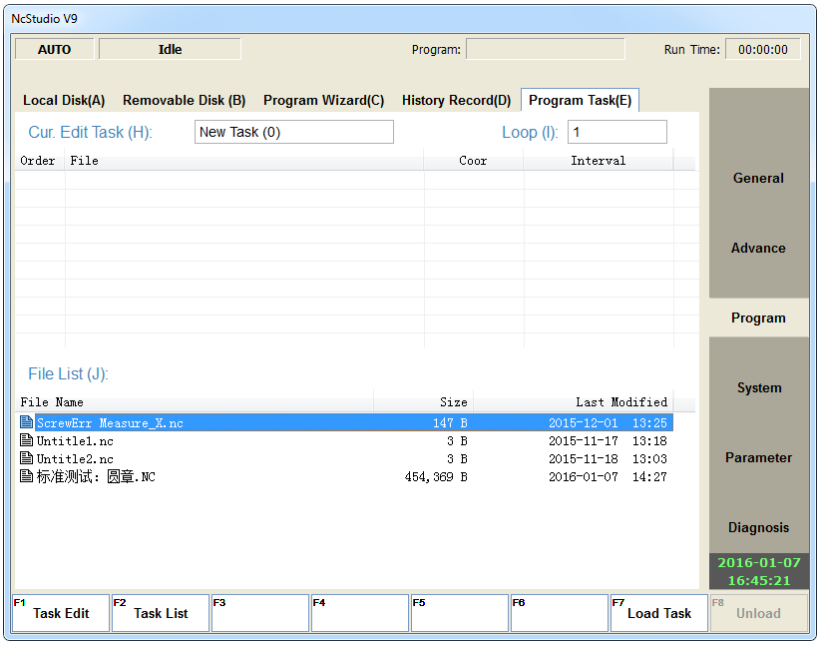


Fig. 3-66 Interface Program Task

- **Task List**

In interface Program Task, you can press F2 or letter key H, and dialog box “Task List” will pop out, as shown in Fig. 3-67. Then you can press F1, F2, F3 and F7 to create new task, delete, rename and open tasks. You can press F8 to cancel the operations and close the dialog box.

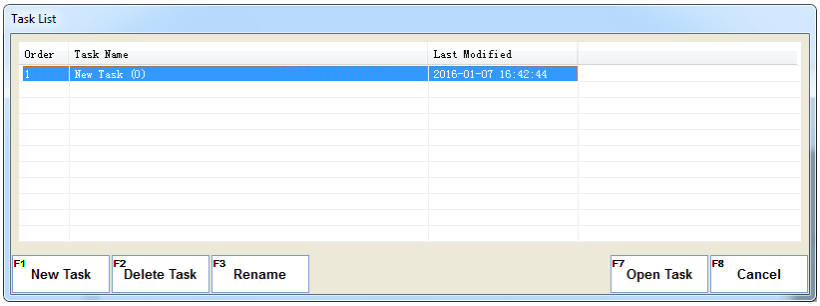


Fig. 3-67 Dialog Box of Task List

- **Edit Current Task**

After creating a task, press F1 to edit the current task, new manipulation buttons will appear in the bottom of the interface, as shown in Fig. 3-68.

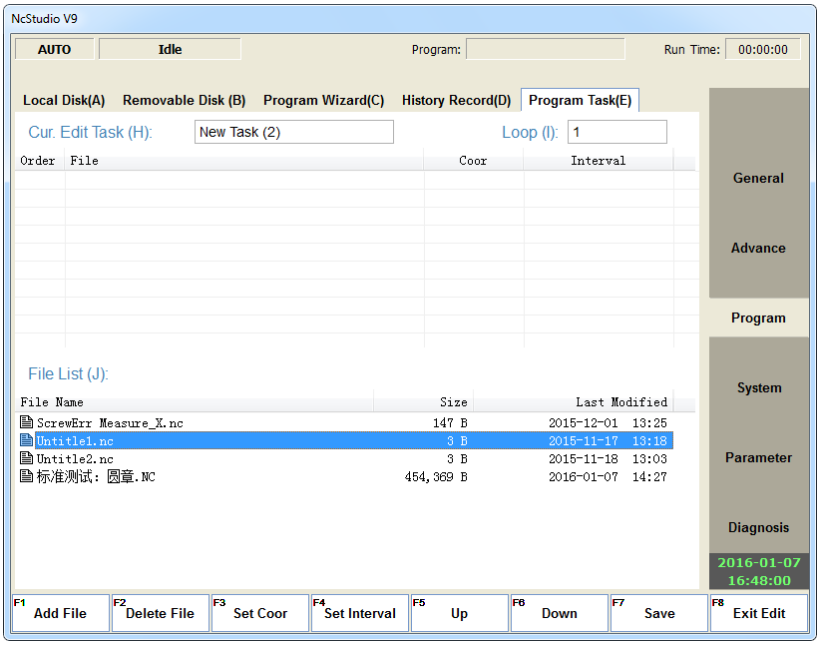


Fig. 3-68 Edit Current Task

Add File

In **Task List** area, press “↑” or “↓” to select a folder, and then press Enter to open a folder. Select a program file in .tap、.nc、.eng format and press F1 to add the file into **Cur. Edit Task** area. Several files can be added in the same task, and the same file can be added repeatedly.

After you press F1 **Add File**, dialog box “Set Coor” will pop out as shown in Fig. 3-69. You can set the coordinate system as any value from G54 to G59. The default coordinate system is G54. Or you can press F8 to set extra coordinate systems. Press F7 to confirm setting or F8 to cancel setting.

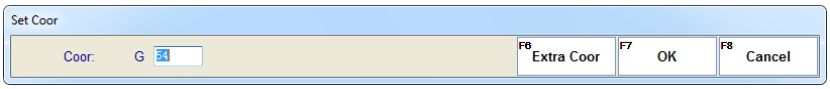


Fig. 3-69 Dialog Box Set Coor

Delete File

In **Cur. Edit Task** area, press " ↑ " or " ↓ " to select a program file and then press F2 to delete the file.

Set Coordinate

In **Cur. Edit Task** area, press " ↑ " or " ↓ " to select a program file and then press F3 to set the coordinate systems.

Set Interval

The machining interval between two adjacent machining files with second as unit. After selecting a task in **Cur. Edit Task** area, press F4 to set the interval. Press F1 to confirm setting or press F2 to cancel setting.

Up/Down

In **Cur. Edit Task** area, pressF5/F6 to move the file you selected up or down to change its order.

Save

Press F7 to save the task you have edited.

Exit Edit

Press F8 to exit editing of the current task.

- **Loop**

Click the input box on the right of **Loop** or press letter key I, and you can set the task loop in the dialog box popping out. The default is 1. Press F1 to confirm setting or press F2 to cancel setting.

- **Load Task**

Press F7 to load tasks. During machining, the machining files will be loaded in order.

- **Unload**

Press F8 to unload the current task.

3.14.3.4 Parameter Specification

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

- **Parameters Related to G Code**

Parameter		Meaning	Setting Range
N62000	Deceleration distance	To protect tools, machine tool will decelerate at a certain position nearby the target (at [approach speed]). This parameter indicates the distance between the position beginning to decelerate and target position.	0~999
N62001	Approach speed	Feed speed of tool approaching workpiece (its distance away from workpiece is smaller	0.001~99999

		than deceleration distance) during positioning	
N62021	Arc radius tolerance	In the IJK incremental representation of G02 and G03, the circle radius is calculated twice. Generally, the two values calculated are not the same and their D-value is called arc radius tolerance. Typically, arc instruction does not incur too large tolerance, and the recommended value is about 10 microns.	0~9999

- **Parameters Related to 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 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 to 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	Machining only one shape each time and go to next shape only after last one finished.	YES: enabled; NO: disabled

Parameter		Details	Setting Range
N65105	Enable Bottom Cutting	Valve operation is enabled only when [3D cutting] is on the workpiece surface.	YES: enabled NO: disbled
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 to 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: enabled; NO: disabled
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: do not use
The parameters above are related to translating ENG file.			

- **Related Parameters**

Parameter		Details	Setting Range
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
The two parameters above are related to deep hole cutting.			

3.15 Handwheel Operation

3.15.1 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. 3-70.

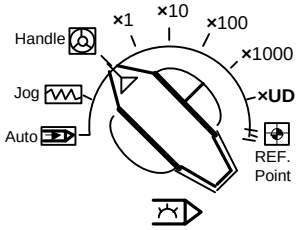


Fig. 3-70 Mode Selection Knob

In handwheel mode, you can configure a handwheel to control the machine tool. As shown in Fig. 3-71, 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. 3-71 A Handwheel

- **Related Parameters**

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.			

Parameter		Details	Setting Range
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	0.001~10

Parameter		Details	Setting Range
		received when a handwheel sends 1 pulse.	
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: do not decelerate

3.15.2 Handwheel Guide

NK300 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.

The software interface for handwheel guide is as shown in Fig. 3-72.

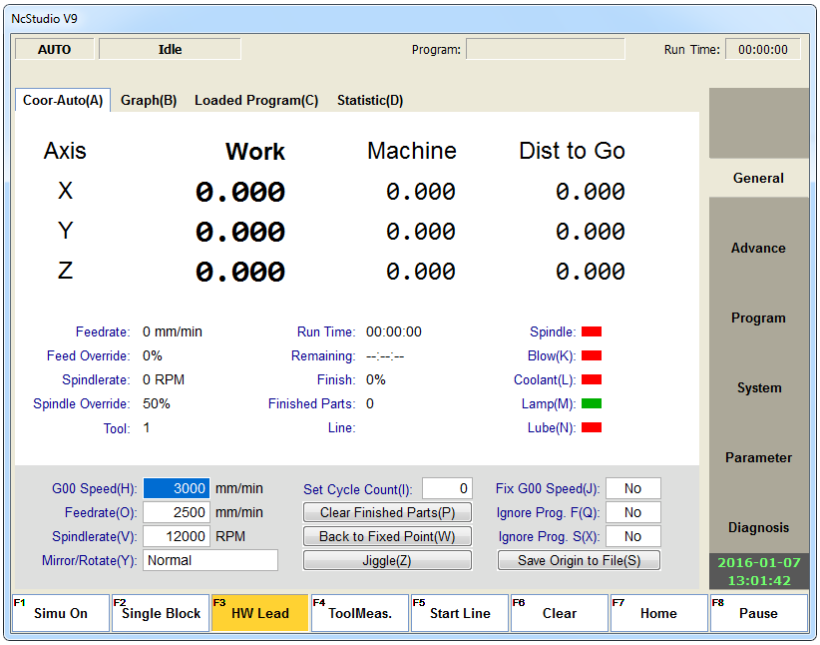


Fig. 3-72 Handwheel Guide Interface

In auto mode, press F3 in interface **Coor-Auto** in functional area **General**. 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.

NK300 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**

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)		
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

3.16 System Management

In functional area **System**, you can press F6 to restart the software, F7 to restart the system and F8 to shut down the system.

3.16.1 Configuration and Language Setting

1) Configuration Setting

NK300 currently supports “Standard” and “Rotary Y” configuration. In addition, you can add other configurations according to your needs. You can switch configuration between “Standard” and “Rotary Y” in interface **Configuration**.

See Fig. 3-73. You can press “↑” or “↓” to move the cursor to the desired one, and then press F1 to confirm. A dialog box will pop out asking “Configuration changes may cause damage to your machine, are you sure to change it?” Select “Yes” and another dialog box will pop out asking “New active configuration has been selected, please restart NcStudio to make it valid, do you want to go on?” Slect “Yes” and the software restart to complete configuration switchover. After system restarted, you need to set relevant parameters again.

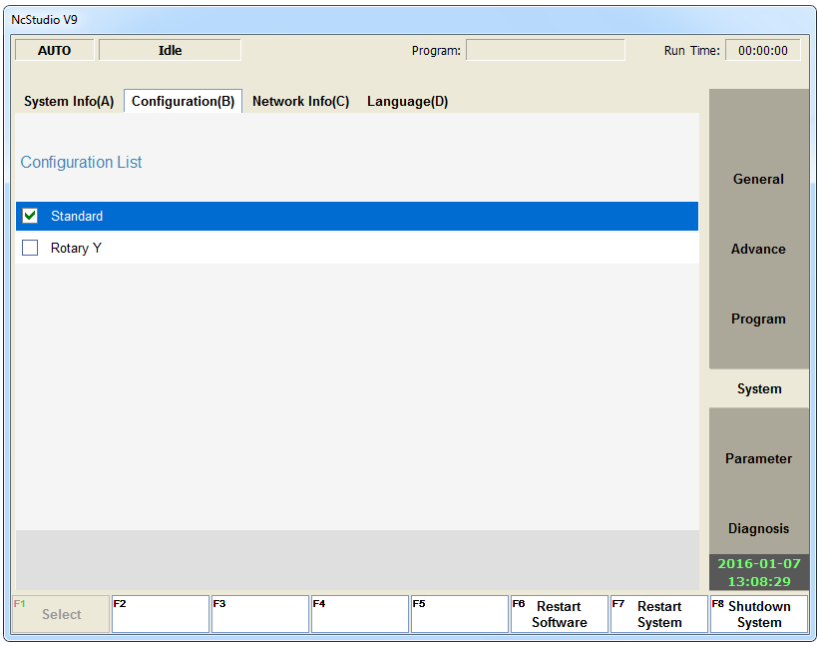


Fig. 3-73 Dialog Box Configuration

If you change configuration from “Standard” to “Rotary Y”, you need to set the pulse equivalent. Refer to section 3.2.2 for the details.

- **Related Parameters**

Parameter		Meaning	Setting Range
N14001	Programming Units	The measuring unit for rotary axis data in the machining file under Rotary Y configuration.	0: angle (degree) 1: the distance of rotary workpiece surface (mm)
N14002	Workpiece diameter	Program diameter of workpiece to be processed under turntable mode	1~3000
N14003	Rotary Y max velocity	This value is decided by mechanical and electrical characteristics of machine tool. At any time, the rotate speed of rotary axis will not exceed this value.	0.001~9999
N14004	Angular acceleration	This value is decided by mechanical and electrical characteristics of machine tool. At any time, the acceleration of rotary axis will not exceed this value.	-
N14005	Coordinate units	The unit of rotary axis value displayed on the program interface	0: degree 1: millimeter
Under turntable mode, Y axis works as rotary axis. Parameter N14002 refers to the diameter value of workpiece to be processed, related to the diameter programmed in the machine file. If this parameter is not set properly, the size dimension of workpiece in actual processing will be affected. Parameter N14003 and N14004 set the maximum rotate speed and acceleration of rotary axis. Their setting values are related to the mechanical characteristics of machine tool.			

Parameter N14005 sets the unit for the value of rotary axis, related to the unit in the programming of machine file. If angle is used in programming, it should be set as “0”; if millimeter is used in programming, it should be set as “1”.

2) Language Setting

As shown in Fig. 3-74, interface **Language** of functional area **System** displays the language list. You can select English or Chinese interface according to your needs. Press F6 to restart the software after selecting the desired language to make your selection effective.

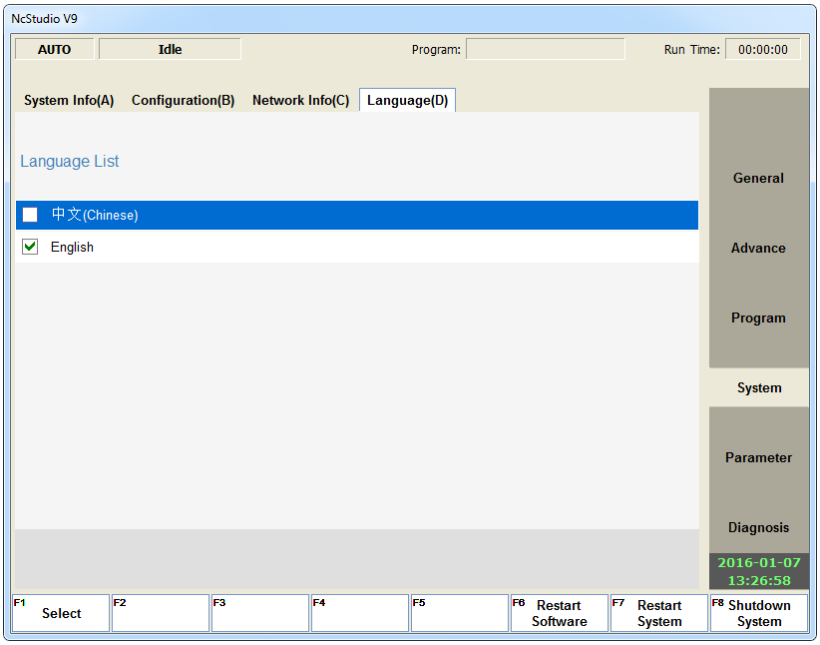


Fig. 3-74 Interface Language

3.16.2 IP Setting

NK300supports network connection. You can obtain IP address automatically or set it manually.

In interface **Network Info** under functional area **System**, “OS OEM Version”, “Controller Name’, “Work Group”, “network connection state”, etc. are displayed. Pressing letter key J “setting” button, and you can obtain IP address automatically or manually in the dialog box popping out.

3.16.3 System Info

In functional area **System**, the default interface displays **System Info**, including CNC software info and hardware info, as shown in Fig. 3-75.

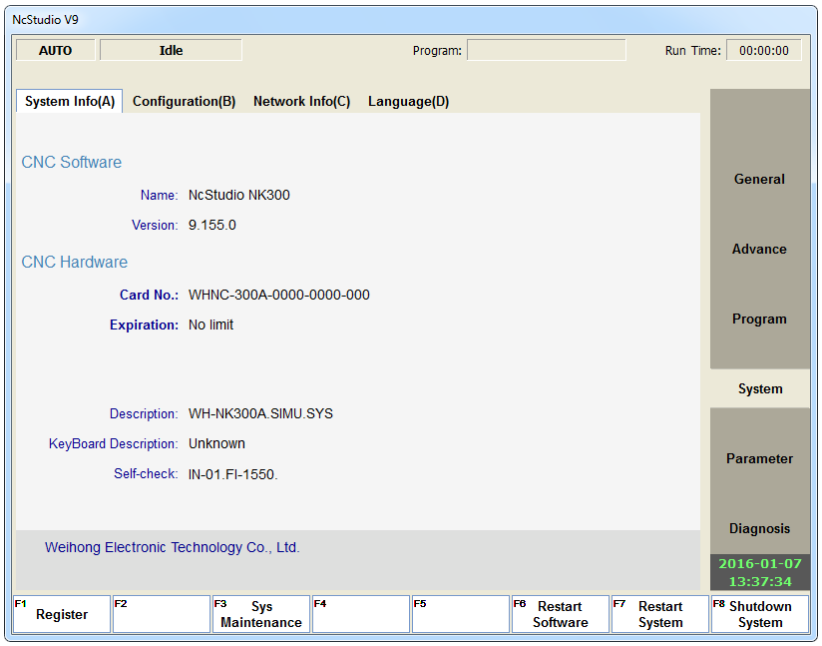


Fig. 3-75 Interface System Info

3.16.4 Registration

In interface **System Info** of functional area **System**, “Registration code” can be used to register the system and limit the system service time. Registration code is generated by registration code maker.

The software supports registration by hour or by day. If you choose register by hour,

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.

1) View the current board No. in interface **System Info** in functional area **System**, as shown in Fig. 3-76. Press F1 to open dialog box “Register” and you can also see the board number as shown in Fig. 3-77.

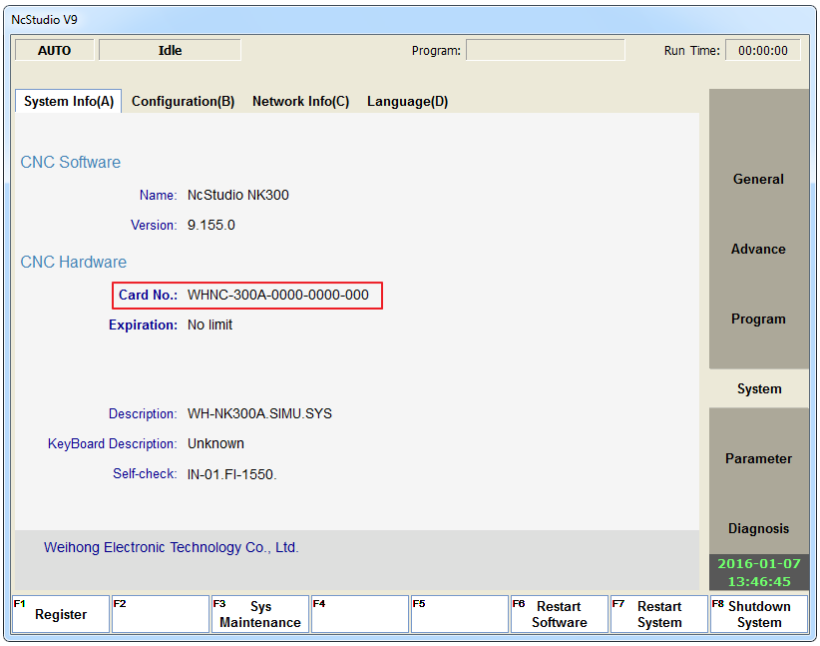


Fig. 3-76 Board No. Display-1

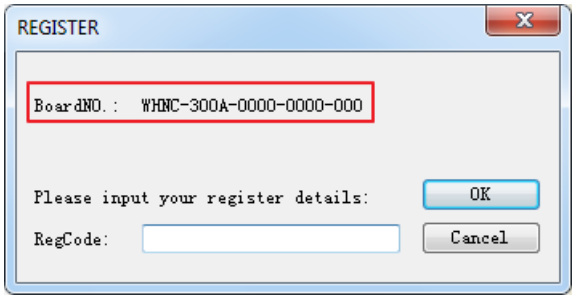


Fig. 3-77 Board No. Display-2

2) Double click registration code maker “GetRegCode.exe”, and then enter the password “ncstudio” (revisable) in the dialog box as shown in Fig. 3-78. Then press “ok”, input control card serial number and limited service time, and then click “generate” to generate the new code displayed at the lower part, as shown in Fig. 3-79. If service time does not need limiting, input “-1” in the “limited time” bar to generate an unlimited code.

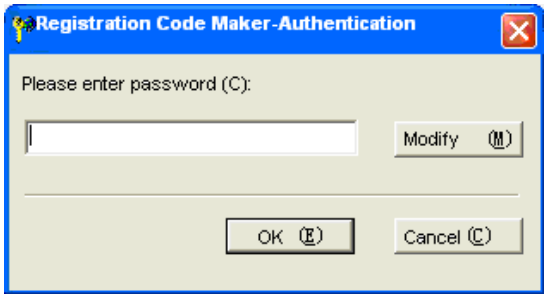


Fig. 3-78 Registration code maker-1

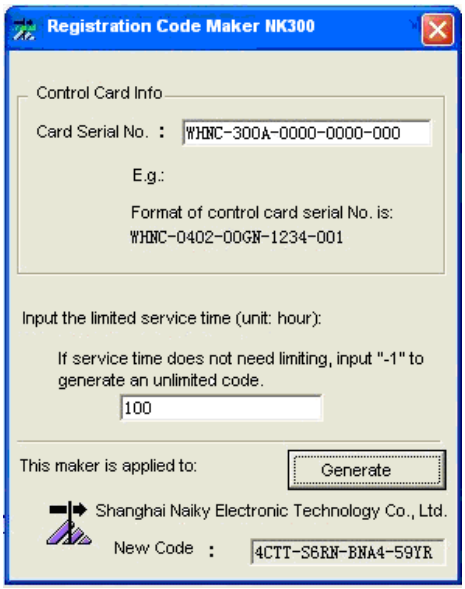


Fig. 3-79 Registration code maker-2

3) Press F1 Register in interface **System Info** of functional area **System**, then input the registration code generated in the second step, and then click “ok”. System prompts “register successfully”.

CAUTION

ID of board 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.

3.17 Auxiliary Function

3.17.1 Single Block

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.

Press SF1 to enter functional area **General** and then press F2 to open single block mode, as shown in Fig. 3-80.

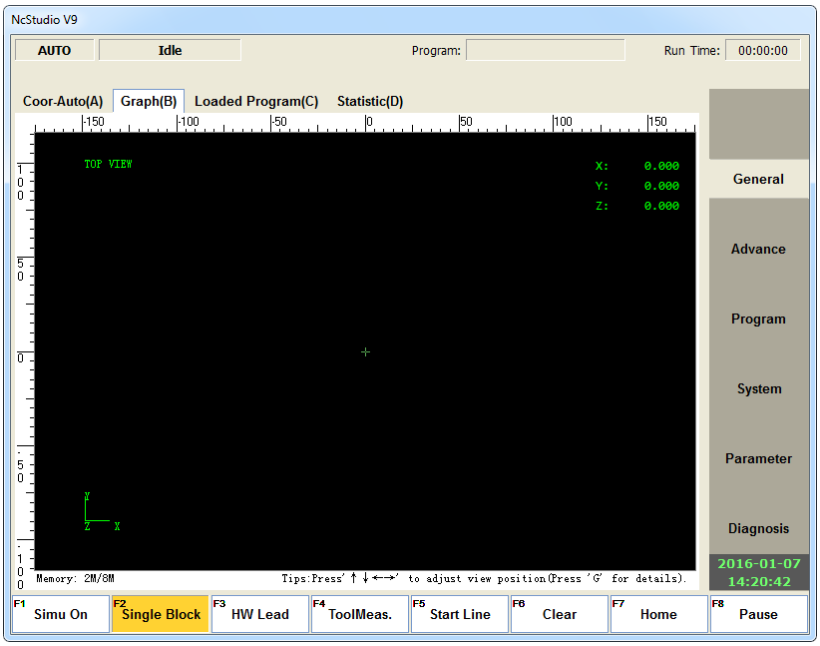


Fig. 3-80 Single Block

3.17.2 Start Line

Function “Start Line” is used for select any blocks for machining.

Press SF1 to enter functional area **General**, and then press F5 to process the machining task in single step mode, as shown in Fig. 3-81.

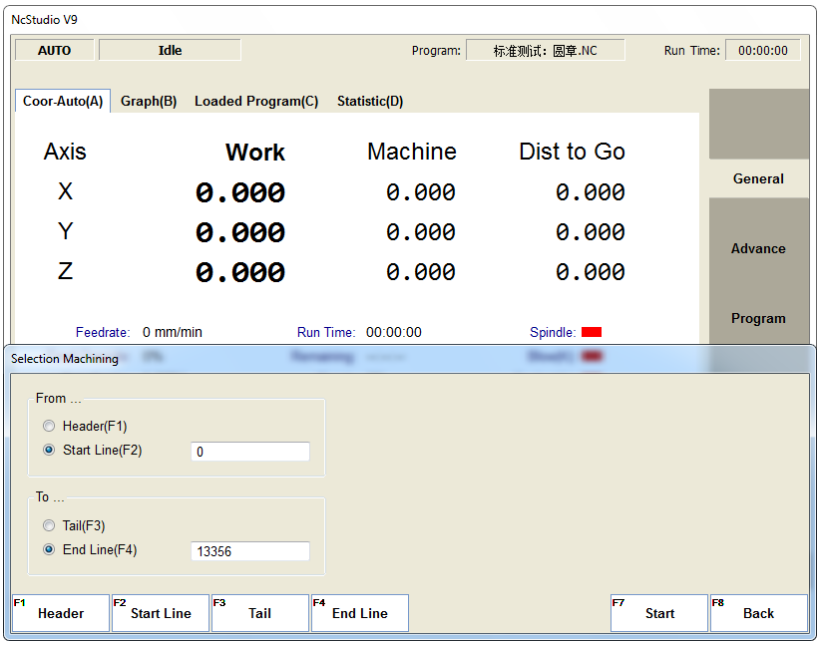


Fig. 3-81 Dialog Box of Start Line

Users can process the specified program segment to be executed (via shortcut key F7) by entering its segment number. With this function, users can process any segment freely.

3.17.3 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.

3.17.4 Parameters Auto Backup

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 SF5 to enter functional area **Parameter**, and then press letter key F to enter interface **Auto Backu**p, as shown in Fig. 3-82. And press " ↑ " or " ↓ " to select valid parameters, then press F1 to recover the selected backup parameters.

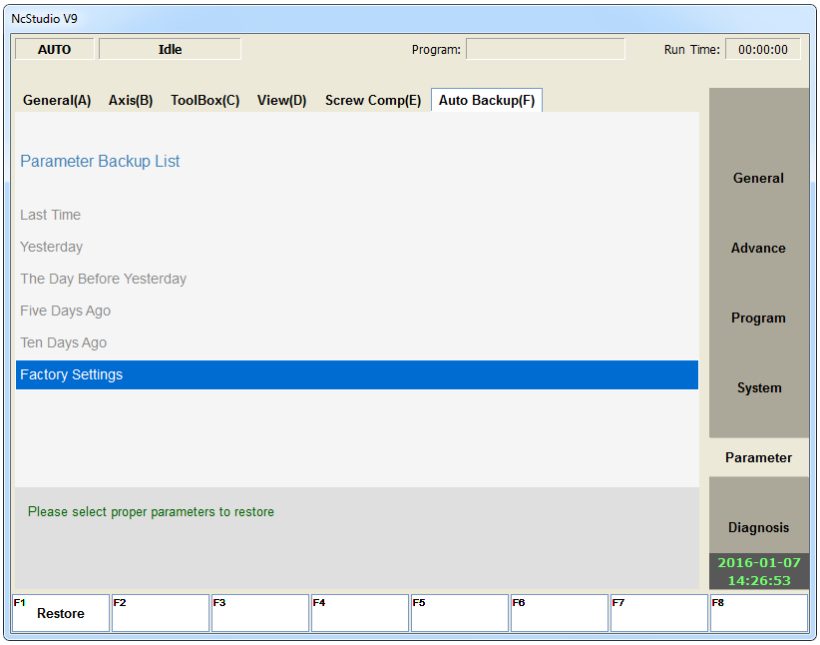


Fig. 3-82 Dialog Box "Restore Parameter"

3.17.5 Manual Data Input (MDI)

Press SF2 and then press key C to access the MDI interface, as shown in Fig. 3-83.

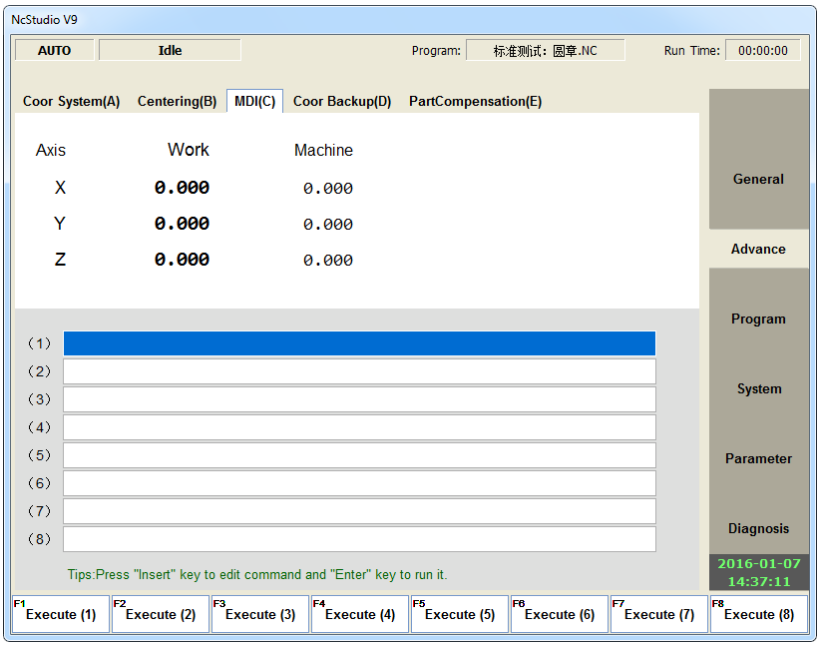


Fig. 3-83 Manual Data Input Screen

In the upper part of the interface are the workpiece coordinates and machine coordinates of each axis. In the lower part are command lines where you can enter commands.

Press key " ↑ " or " ↓ " to select a line and then press insert to open 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 F1~F8 to execute commands in the corresponding input box.

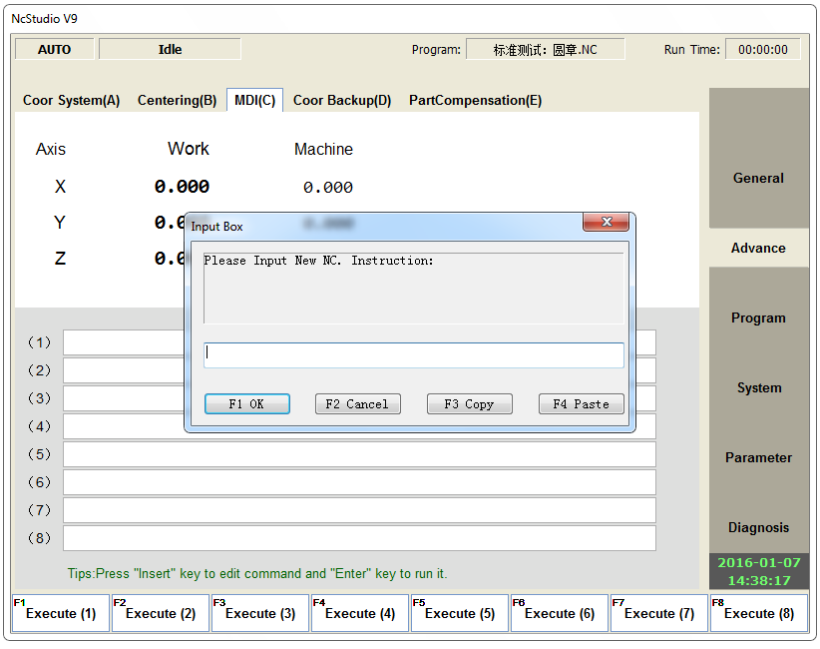


Fig. 3-84 Input Box of Manual Data Input

3.18 Tool Magazine

3.18.1 Auto Tool Change for A Linear Tool Magazine

Linear tool magazine stores tools in the form of array. 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:

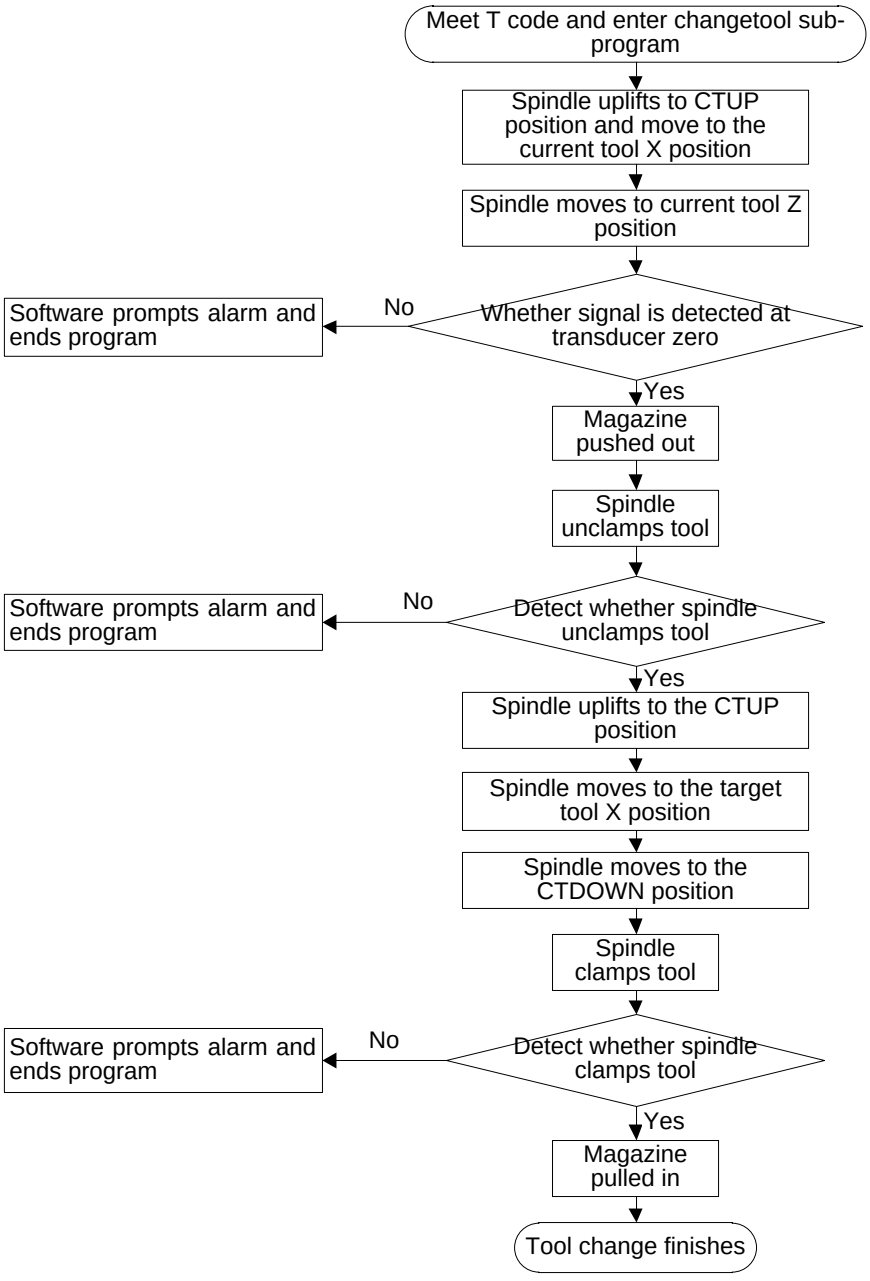


Fig. 3-85 Process of Auto Tool Change for A Linear Tool Magazine

- **Related Parameters**

Parameter		Description	Setting Range
N66074	Group 1 Tool Position on X-axis	Group 1 tool coordinated position of on X-axis	-
N66075	Group 1 Tool Position on Y -axis	Group 1 tool coordinated position of on Y-axis	-
N66076	Group 1 Tool Position on Z-axis	Group 1 tool coordinated position of on Z-axis	-

There are several groups of tool are provided in the system, which will not be mentioned here.

3.18.2 Auto Tool Change for A 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 follows.

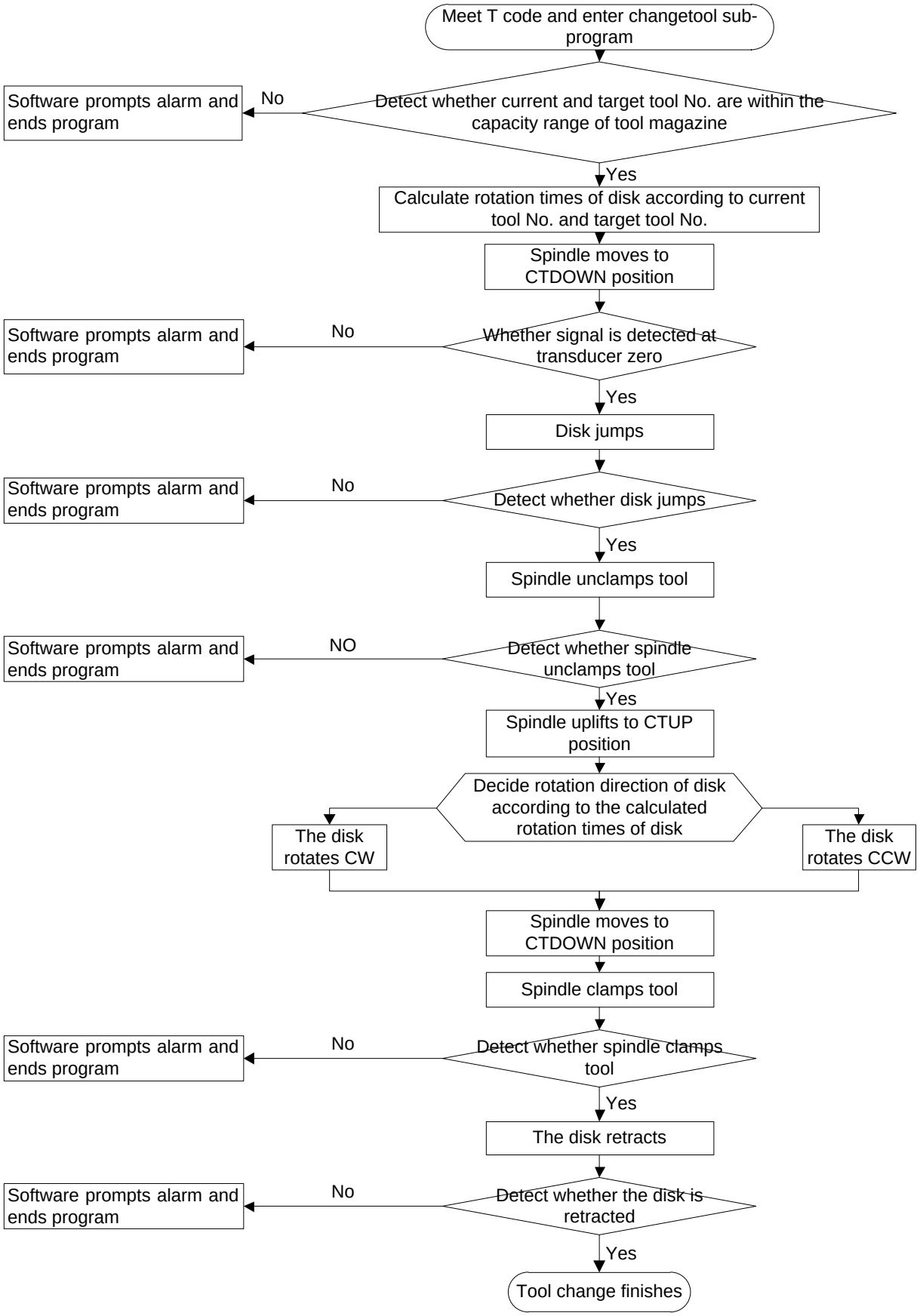


Fig. 3-86 Process of Auto Tool Change for A Circular Tool Magazine

3.18.3 Parameter Specification

- **Related Parameters**

Parameter		Details	Setting Range
N66020	Tool Magazine Type	-	0: Null; 1: Disk Tool Magazine 2: Linear Tool Magazine
N66021	Tool Magazine Capacity	-	1~255
N66022	Check Change ToolNo	-	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.
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.			

- **Related Parameters**

Parameter		Details	Setting Range
N66000	Prompt For Tool Change	Whether to pause and prompt tool change when meeting tool change command.	YES: Enable; NO: Disable
N66002	Pause in ToolChnage for Same Active Target Tool NO.	On condition that “Prompt for ToolChange” is enabled and this parameter set to YES, machine will pause when reach the same T number command in tool change; when set to NO, machine will not pause if the active and target T No. are the same.	YES: Pause; NO: Do not pause
N66005	Upper Position	Z-axis machine coordinate when a tool moves to tool magazine for tool change, or	-99999~99999

Parameter		Details	Setting Range
		CTUP position	
N66006	Lower Position	Z-axis machine coordinate of tool change position when a tool moves downwards from Upper Position, or CTDOWN position	-99999~99999
N66007~ N66009	Spindle Position in Tool Change X/Y/Z	Machine coordinate value when spindle changing a tool, usually used for circular tool magazine parameter setting.	-99999~99999
N66010~ N66012	Deceleration Position X/Y/Z	Machine coordinate value of spindle position before tool change	-99999~100000
N66015	Tool Change Speed	Movement speed of spindle during tool change	0~100000
N66016	Z Axis Speed	The speed of Z-axis moving to Upper and Lower Position during tool change	0~13740
N66017	Automatic Tool Measure	Whether to execute auto tool measurement after tool change	NO: Not execute; YES: Execute

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 Double Z Axes CNC System

Matched with double Z axes software, NK300 series CNC system can be used to control double Z axes machine tools.

4.1 Configuration Selection

Currently, double-Z axes control systems have been equipped with the following configurations: “Unite Config”, “Turn Config”, “Unite Rev Config” and “Turn Rev Config”. In addition, other configurations can be added according to users’ needs. Switching between configurations is performed in [Configuration] screen of [System] function section, shown in Fig. 4-1:

Press shift key “ ↑ ” or “ ↓ ” to move to the target one in [Configuration] screen. After pressing F1, select “Yes” successively in the pop-up dialogues, and then press [Enter] key to restart the system and finish configuration selection when the system shuts down and “Operation Succeed” dialogue pops up. Set the related parameters again after switching configuration.

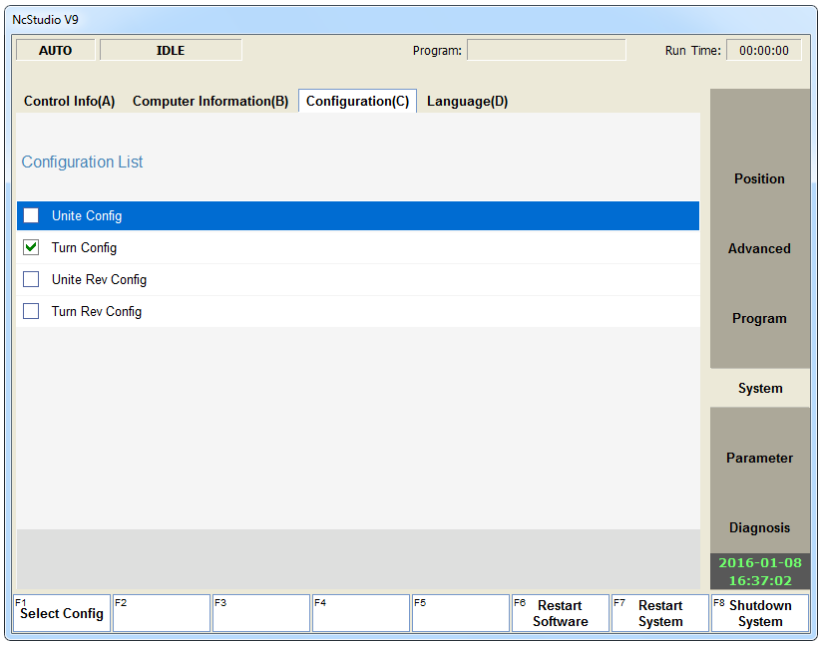


Fig. 4-1 Configuration selection interface

In **Unite Config**, you can select Z1 mode, Z2mode, or Z1Z2 mode. In Z1Z2 mode (unavailable in T**urn Config**), double Z axes move simultaneously and uniformly.

Select one of the modes in interface **SwitchAxis** of functional area **Position**, as shown in Fig. 4-2.

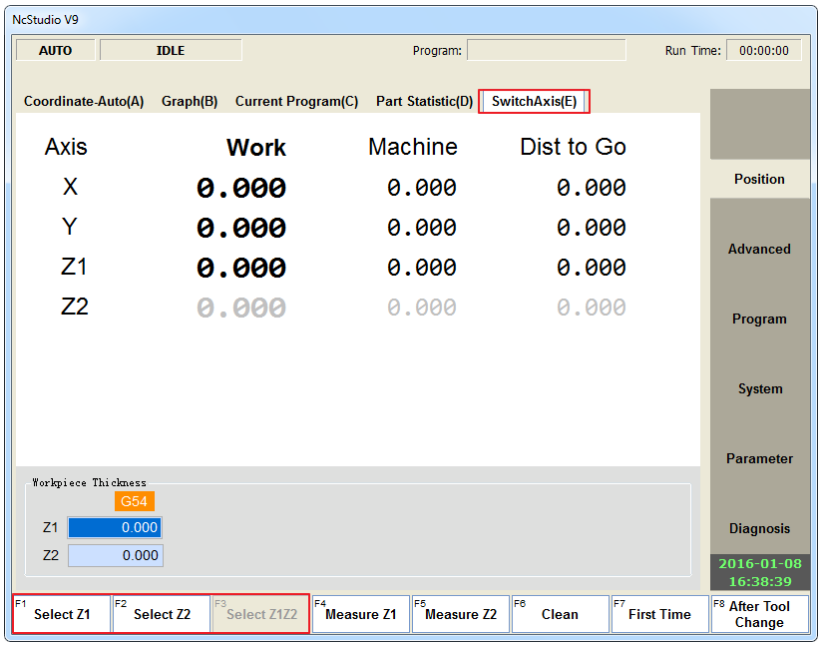


Fig. 4-2 Axial switching interface

- **Unite Config**

Under this configuration, a machine tool will not move when you switch between **Select Z1** and **Select Z2**, i.e. the heights of Z1-axis and Z2-axis will remain unchanged.

Under this configuration, when button **Select Z1Z2** is pressed, if the workpiece coordinates of Z1 and Z2 are not identical, the machine tool may generate motions; if identical, the machine tool will keep still; when the machine tool stops, the workpiece coordinates of Z1-axis and Z2-axis in Z direction will be identical.

- **Turn Config**

Button **Select Z1Z2** is disabled in this configuration.

Press F1 or F2 to activate button **Select Z**1 or **Select Z2**; then the current axis will move to machine origin while the target axis will not move. For instance, current axis is Z1, while Z2 is selected now; Z2-axis will not be activated until Z1 returns to machine origin.

In **Turn Config**, if T command is encountered, the machine tool will move as below (e.g. shift to T2 from T1):

1) Z1 spindle stops
2) Z1 uplifts to machine origin
3) Z1 enable is closed
4) Select Z2, and Z2 is enabled
5) Z2 moves to the workpiece coordinates position of Z1 before tool change

- **Unite Rev Config and Turn Rev Config**

Y-axis will serve as rotary axis under these two configurations; please refer to section 3.2.2 in *NK300 Integrated CNC System Manufacturers' Manual* for pulse equivalent setting.

4.2 Tool Calibration

Tool calibration in double Z axes control systems includes “first calibration”, “after tool change”, and “automatic measuring”. Please refer to section 3.7.4 in *NK300 Integrated CNC System Manufacturers' Manual* for the principle of “first calibration” and “exchanged calibration”, while “automatic measuring” will be mainly introduced in this section.

4.2.1 Software Interface

In interface **SwitchAxis** of functional area **Position**, press F7and F8 to execute **First Time** and **After Tool Change** respectively. Press F4 and F5 to execute **Measure Z1** and **Measure Z2**, as shown in Fig. 4-3.

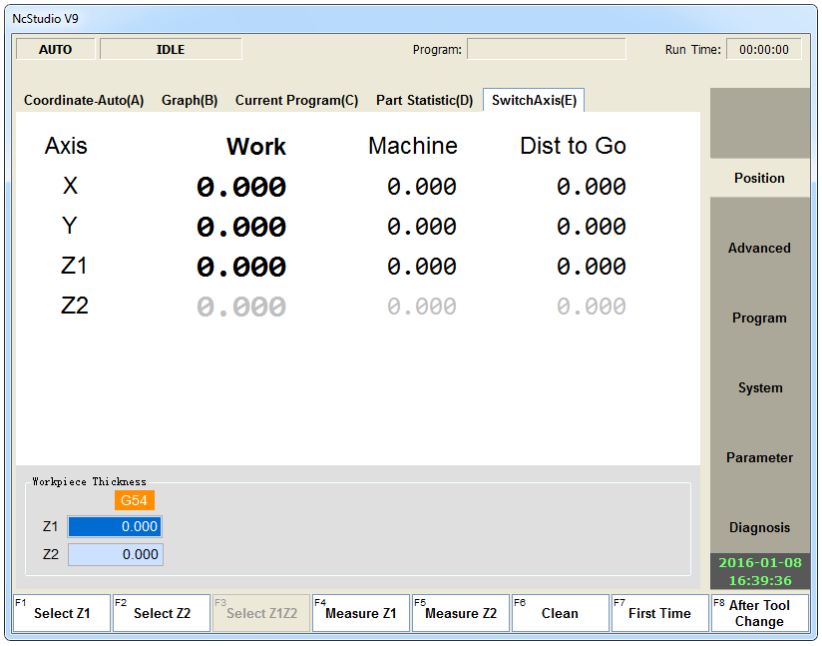


Fig. 4-3 Sub-function interface of axial switching

Press F1, F2 to select Z1-axis, Z2-axis respectively in this interface, and press F3 to select both Z1 and Z2. The corresponding Z-axis can be measured automatically after selecting Z1 or Z2.

4.2.2 Principle and Operation Steps

Automatic measuring is similar to fixed presetting, and the presetting result will be saved to the position offset of tool, the sketch map is shown in Fig. 4-4.

Tool offset= Machine coordinate – Thickness of tool presetter

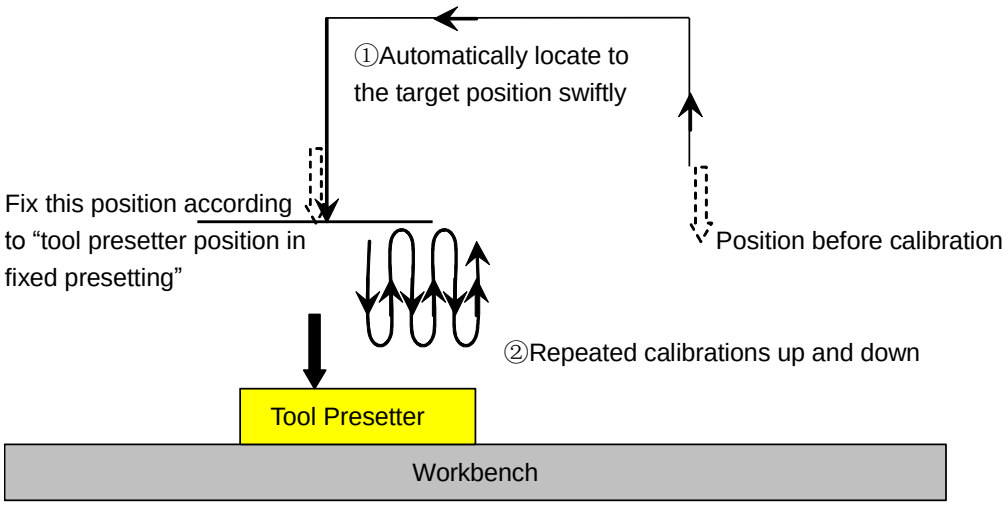


Fig. 4-4 The process of automatic measuring

After button **Select Z1** is selected in **Turn Config**, the steps of using a tool for the first time are as below:

1) Press F4 **Measure Z1** to begin tool calibration, and the system will record tool offset value automatically;
2) Manually move the tool to workpiece surface to clear workpiece coordinate, and the system will record the D-value of tool presetter and workpiece thickness automatically;
3) You only need to execute step 1 to begin normal machining after tool change.

The relationship between workpiece coordinate, machine coordinate, tool offset and workpiece thickness is as follows.

Workpiece coordinate= Machine coordinate- Tool offset - Workpiece thickness

5 Punching & Tapping CNC System

This chapter introduces the special function of NK300, i.e. punching & tapping function.

5.1 Related Concepts

- **Spindle Exact-stop**

To finish the motion process of ATC (Automatic Tool Change), the CNC machine should hold a structure for setting spindle exact-stop. Since the tool is installed on the spindle, the cutting torque cannot be transferred solely by friction of taper hole in cutting, thus a bump key should be specified on the front end of spindle; when the tool is installed on the spindle, the key slot of tool holder should aim at the bump key to realize tool change. Therefore, the spindle must accurately stop at a certain fixed angle.

Set the spindle stop position during spindle exact-stop by adjusting the related parameters of servo spindle.

For repeated tapping of the same hole, the “spindle exact-stop” is executed once before tapping command each time, so the system will not begin tapping until receiving “The signal of spindle exact-stop”.

This system achieves spindle exact-stop by sending control signals to the servo spindle with self-contained exact-stop function. After the output port “Spindle Exact-stop (OUT2)” outputs signals and the input port “The Signal of Spindle Exact-stopped (IN4)” receives signals, the exact-stop finishes.

- **Pulse-pos Mode:**

This mode is used for tapping, and the system controls A-axis by pulse-position mode.

5.2 Software Interface

Set parameter N21000 “Tapping Type” as 1 or 2, and restart the software. Press key SF1 to enter functional area **General** and then press letter key E to enter interface **TappingDebug**, as shown in Fig. 5-1.

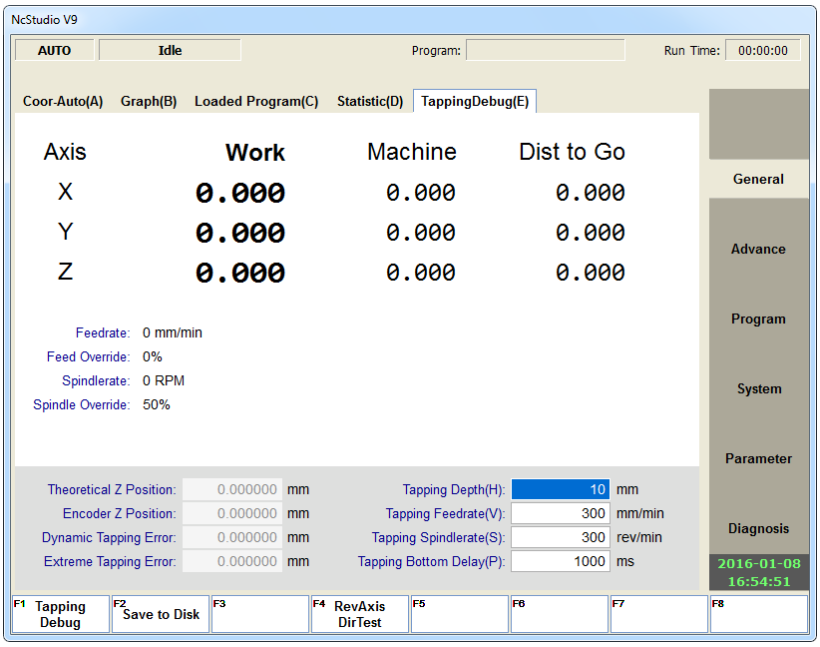


Fig. 5-1 Interface Tapping Debug

Before machining, view the tapping dynamic error with tapping debug function, and adjust tapping parameters to make tapping precision reach requirement and improve the quality of tapping product and avoid wasting.

- **RevAxis Dir Test**

Press F4 to verify the correctness of the spindle and calculate the value of parameter N11160 “Frequency Division Pulses of PG(X4)”. Watch and collect the value of pulse number (A). If the number is positive, it means the direction is correct. Otherwise, you need to adjust the parameter N11160. Get the Max. pulse number and divide it by 2. Set the result as the value of parameter N11160.

- **Tapping Debug**

Set the customized punching and tapping parameters, and then press F1 to begin debugging. Watch the ideal position, feedback position, tapping dyamic error and the Max dynamic error. Then adjust the parameters to minimize the punching and tapping error, improving product quality.

- **Save to Disk**

Press F2 to save the debug error data into USB for later analysis.

5.3 Related Parameters

Parameter		Details	Setting Range
N10010	Pulse Equivalent	Displacement or angle on corresponding axis per pulse.	Depend on motor parameters, resolution and bandwidth, of rotary axis.

Parameter		Details	Setting Range
	The formula of pulse equivalent is： $\text{Electronic gear ratio}\frac{B}{A}=\frac{\text{Encoder resolution}\times\text{Pulse equivalent}}{360}\times\text{Deceleration ratio}$		
N11110	Axis Encoder Dir	The direction of encoder.	1: positive; -1: negative
N11160	Frequency Division Pulses of PG(X4)	Encoder feedback pulse No. via frequency division of servo per resolution of motor.	Same as the setting value of encoder on rotary axis.
N11180	Position Loop Gain	The responsiveness of the position loop id determined by the position loop gain.	Same as that of drivers.
N11190	Velocity Feedforward Ratio	The ratio of the velocity feedforard, position lagging cause by velocity can be reduced by adding up this value.	0~1000(0%~1000%)
N21000	Tapping Type	0: Not tapping; 1: Synchronize Tapping; 2: Following tapping	0; 1; 2
N21001	Spindle Acc while tapping	-	0.001~100000
N21002	Spindle Acc while tapping back	-	0.001~100000
N21003	Spindle jerk while tapping	-	-
N21004	Spindle jerk while tapping back	-	-
N21005	Start speed of spindle while tapping	-	0~6000
N21006	Backward distance of relapse tapping	The backward distance of Q additional tapping.	0~999
N21007	The direction of followed axis of tapping type	Valid under following tapping mode.	-
N21008	Confirm exact stop before tapping or not.	Confirm exact stop before tapping or not	YES: exact stop; NO: not exact stop
N21009	Variable-frequency Drive mode switch output	PLC address of output port which switch between analog mode and pulse mode.	-
N21010	Spindle exact stop confirm signal	PLC address of spindle exact stop confirm input port.	-
N21011	Spindle exact stop	PLC address of spindle exact stop output	-

Parameter		Details	Setting Range
	output port	port.	
N62730	G73_G83 Lifting Distance	Tool Lifting Distance for each drilling	-99999~99999(mm)

5.4 General Function Keys

General function keys K1 & K2 on the operation panel WH106 are specially defined for tapping function when configured with software available of punching and tapping functions.

- **K1 Function Key "Tapping Retracting"**

Press "K1" to control "Tapping Retracting". Only "Tapping Retracting" is allowed by the system after E-stop or program stop during tapping.

- **K2 Function Key "Spindle Exact-stop"**

K2 is used to control "Spindle Exact-stop". Spindle locating begins after "Spindle Exact-stop" port outputs signals, and stops after K2 is pressed again. This function is used for adjusting the spindle position and angle in "Spindle Exact-stop".

5.5 Tapping Instructions

Tapping function is compiled in a machining file by G74 (left-handed tapping cycle) and G84 (right-handed tapping cycle) G codes.

- **G74 Command**

Format: G74 X_Y_Z_R_P_K_

Description:

X_Y_: hole position data (absolute / incremental coordinate)

Z_: the position of point Z of hole bottom (absolute programming); the distance from point R to point Z at the bottom of the hole (incremental programming)

R_: the position of point R (absolute programming); the distance from the initial point to point R (incremental programming)

P_: the dwell time at the bottom of the hole, with unit "ms" and no decimal point

K_: number of repeats (repeated movements and drillings, valid under G91 incremental input)

CAUTION

In the process of tapping, after program stop or E-stop, only "tapping retracting" is allowed, and its functional key is "K1" on the operation panel WH106A2.

Hole-machining process is shown in Fig. 5-2. In G98, the cutter will return to the initial point after the

hole-machining is finished, while in G99, the cutter will return to point R after the hole-machining is finished.

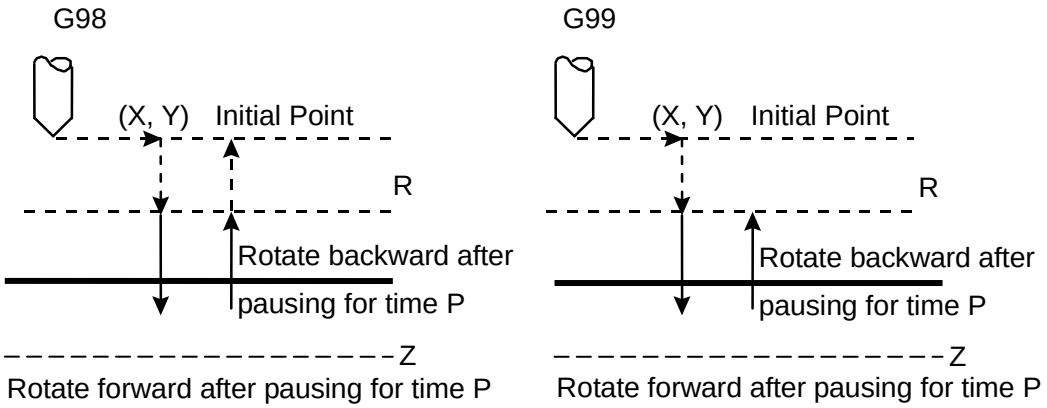


Fig. 5-2 G74 Machining Process

Description of Machining Process:

1) At the beginning of machining, the cutter moves to the specified hole position (X, Y) at G00 speed;
2) Execute “Spindle Exact-stop”;
3) Go down to the specified point R at G00 speed;
4) Tap down to the point Z at the bottom of the hole at G01 speed;
5) Rise to the point R at G01 speed;
6) Rise to the initial point (G98) or point R (G99) at G00 speed.

Programming Example:

```
G90
G00 X0. Y0. Z10. ’Move to the initial point
G17
G90 G99
’Specify the coordinates of point R, point Z and hole 1, pause for 2s,
G74 X5. Y5. Z-10. R-5. P2000.
X25. ’hole 2
Y25. ’hole 3
G98 X5. ’hole 4, and set to return to the initial point
X10. Y10. Z-20. ’hole 5, and set a new point Z to be “-20”
G80
M02
```

- **G84 Command**

G84 has the same format and description as G74, except that it is right-handed tapping cycle.

6 Double Y Axes CNC System

Matched with double Y axes software, NK300 series CNC system can be used to control double-Y axes machine tools.

The processes of returning to X-axis and Z-axis machine origin in double Y axes system are identical with those under three-axis standard configuration, but the processes of returning to Y-axis machine origin mare slightly different under the two configurations.

On each Y-axis of double Y axes machine tool, an encoder is installed. And the line connecting the two encoders must be parallel to X-axis to ensure that the line connecting the machine origins of two Y axes is parallel to X-axis after returning to machine origin is completed. However, due to inherent errors or installation mistakes, the line connecting two encoders may not be parallel to X-axis. Thus, it is necessary to execute origin detection, measuring the position offset of the two encoders and then record it in parameter N74121. And offset compensation will be executed during returning to Y-axis machine origin to ensure that the line connecting the machine origins of two Y axes is parallel to X-axis. The whole process is achieved by **Y Origin Detect** function.

The detection of Y-axis origin is illustrated in Fig. 6-1.

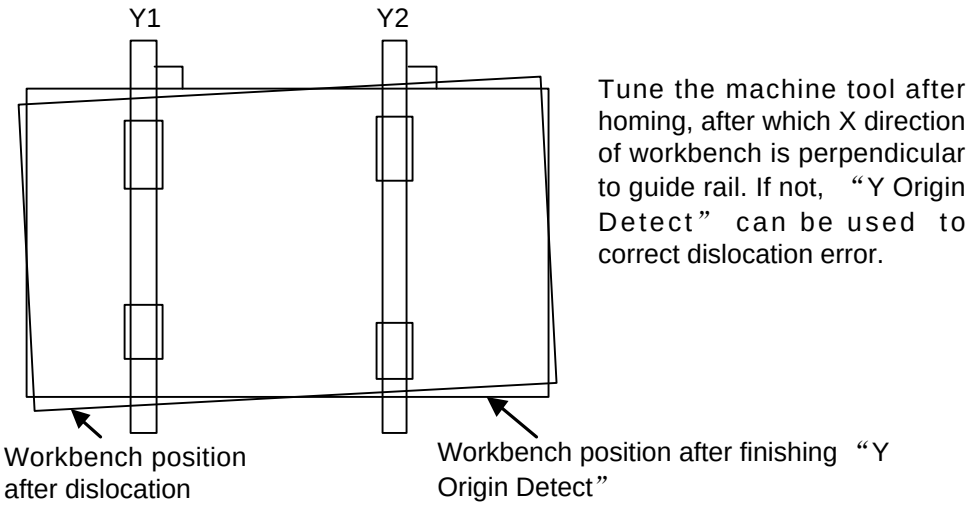


Fig. 6-1 Principle of Y-axis Origin Detection

The operation interface in software is shown in Fig. 6-2.

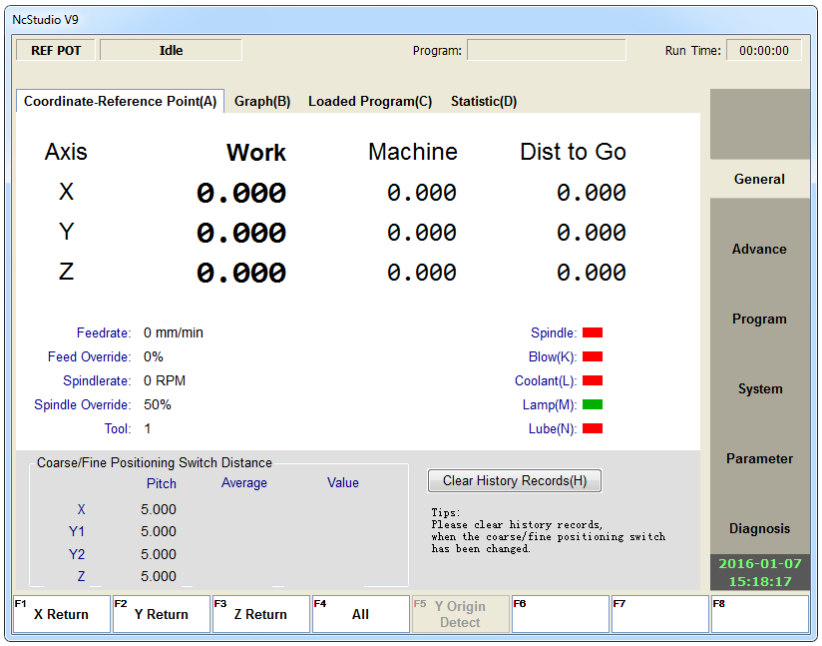


Fig. 6-2 Homing Interface of double Y Axes Control Systems

CAUTION

Parallel between the gantry and X-axis must be strictly guaranteed before “Y Origin Detect” is executed. Once is enough for “Y Origin Detect” execution after software installation.

7 Maintenance

7.1 Operating System Maintenance

When you receive product NK300C, it is ready for use with all systems installed and setup. If any failure occurs, you can restore the software to factory state.

7.1.1 Hardware Configuration

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

7.1.2 Creating System Installation Disk

- **Creating Startup Disk of USB Flash Disk**

Insert the CD to the CD driver, open the root directory of CD and find “NK300 system recovery\USB flash disk startup disk making\hpUpgsh\hpUpgsh.exe”. Then select the USB disk drive for formatting in the opened interface, choose formatting type and start file path (G:\hpUpgsh\boot), and press “start” to start formatting. Press “Yes” continuously to finish the startup disk creation of USB flash disk. The interface of making USB disk startup disk is shown in Fig. 7-1.

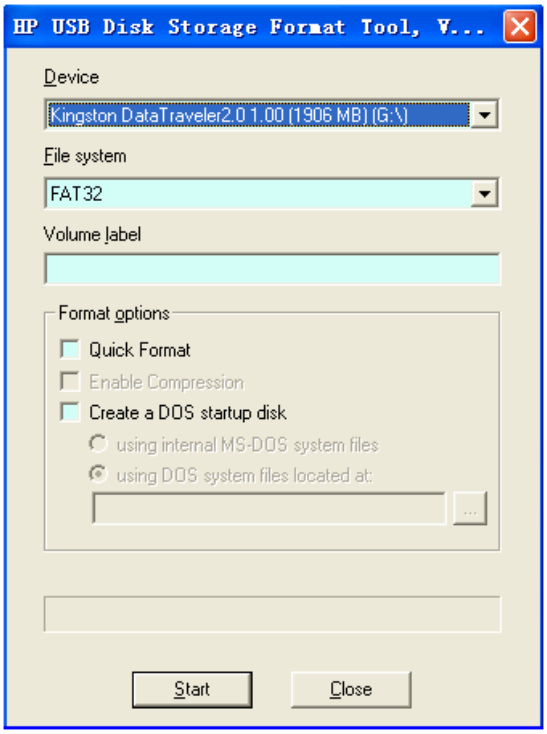


Fig. 7-1 The Interface of Making USB Startup Disk

- **Tool Kit Installation**

After finishing last step successfully, double click the folder following the path “G:\NK300 system recovery\USB flash disk startup disk making\USB flash disk recovery package.exe”. Select “Browse”

under extracting option (shortcut key: ALT+W), and define the “Destination folder” as the root directory of the current USB disk. After “Install” is pressed, file case will start extracting.

Fig. 7-2 Installation

After decompression, installation of recovery backup tool is finished.

7.1.3 Set BIOS Booting from USB Flash Disk

Insert startup disk with backup files into the USB slot on NK300 operation panel→ restart NK300→ press “Delete” to enter into “BIOS SETUP UTILITY”:

1) Press direction key “→” to BOOT→ press direction key “↓” to “1st Boot Device”→ click “enter” key on panel→ “options” dialogue will pop up→ press “↓” to select “USB:…”→ press “Enter” again to finish setting;
2) Likewise, press direction key “↓” to “2nd Boot Device”→ click “enter” on panel→ “options” dialogue will pop up→ press “↓” to select “PM flash card”→ press “Enter” again to finish setting;
3) Press direction key “→” to “EXIT” → select “Save changes and exit” → press “Enter” on panel→ a dialogue will pop up→ click “OK” to validate the setting.

7.1.4 One Key Recovery and Backup

After finishing the above setting, insert the USB flash disk, system will enter one key recovery DOS environment after restarted, in which there are two options, “1” is one key recovery, and “2” is one key backup.

1) Select “1” twice continuously→ enter system recovery interface→ system recovers automatically. Remove the u-disk after recovery. Then set BIOS booting from CF card after restarting of CNC system→ double-click “My computer\ Disk C\ setmappingfiles.bat”→ enter DOS with auto file copy → after file copy completers, restart to finish recovery;
2) Select “2” to DOS environment, input “back” and press “Enter”, enter backup selection menu, in which there are two options, “1” is back to DOS; “2” means auto system data backup. Select “2” to

automatically enter backup interface for system backup. When backup is finished, press "CTRL+ALT+DEL" to exit from UCDOS.

CAUTION

If backup file "diskback.gho" has already existed in the USB flash disk (the image file has already existed), it must be moved or renamed, or system will quit backup automatically.

7.1.5 Set BIOS Booting from CF Card

Insert startup disk with backup files into NK300→ restart NK300→ press "delete"→ enter "BIOS SETUP UTILITY":

1) Press direction key "→" to BOOT→ press direction key " ↓ " to "1st Boot Device"→ click "enter" key on panel→ "options" dialogue will pop up→ press " ↓ " to select "PM Flash card" → press "Enter" on panel again to finish setting;
2) Likewise, press direction key " ↓ " to "2nd Boot Device" → click "Enter" on panel→ "options" dialogue will pop up→ press " ↓ " to select "PS Flash card" → press "Enter" on panel again to finish setting;
3) Press direction key "→" to "EXIT" interface→ select "Save changes and exit" → press "Enter" on panel→ a dialogue will pop up→ click "OK" to validate the setting.

CAUTION

If abnormal condition occurs in the process of system recovery and backup, the causes may be as following:

1) Whether the "boot sequence" of hard disk in BIOS is correct or not; the sequence of USB flash disk startup is USB disk/ PM/ PS, while the sequence of CF card startup is PM/ PS/ USB disk;
2) Whether system has already got problems before backup;
3) Whether the space in USB flash disk is enough or not;
4) When system exits abnormally during backup or recovery, it may also be caused by breakdown of CF card. To solve it, hardware repair to CF card is necessary.

On account of the above problems, it is recommended to check and repair the system disk during recovery and backup, or system performance may be affected, so may data disk.

7.2 Installation and Package of Ncstudio

7.2.1 Initial Installation

Insert the USB flash disk with the saved software into the corresponding USB slot on operation panel of

NK300 series host machine, then press “Ctrl + Alt + Delete” to enter the task manager interface, and then press “Alt + F” key to select “new task”. In the new task dialogue, input “explorer” and press “enter” to confirm it. Then press “Alt + Tab” to switch to Ncstudio. Then Press “Alt + F4” to close it and enter the desktop. At last, enter the U disk, find the corresponding file and install it.

7.2.2 Package and Upgrade

The system holds the function of backup. After installing the software and setting various parameters corresponding to machine tool, you can backup and save the software with proper parameter setting as the original data. And the backup software can be directly installed on machine tool of the same type. This function is realized in system maintenance. The following will introduce system maintenance in detail.

Enter functional area **System** by pressing SF1, and its default sub interface is **System Info**. Press F3 to open dialog box “Upgrade”, as shown in Fig. 7-3.

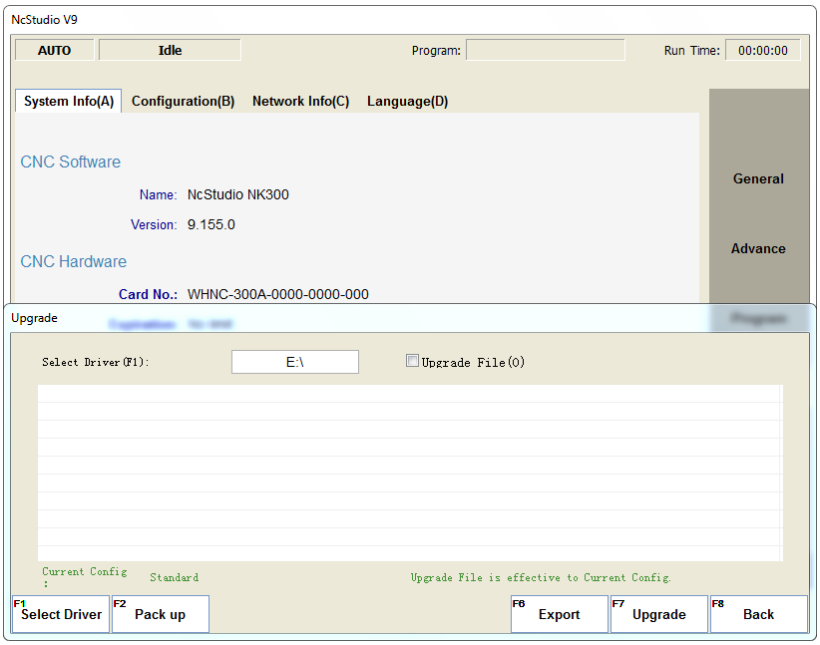


Fig. 7-3 Dialog Box “Upgrade”

There are two types of system maintenance. The default type is “Upgrade” software. After you upgrade the software, the current parameter settings can be saved. And when you click the box before “Upgrade File”, in the file list box, you can select “Public”, “Plc”, “Amend” and “String” files to back up, which is only enabled under three-axis standard configuration.

The dialog box “Upgrade” is shown as Fig. 7-3. Below is an introduction of the buttons in this dialog box.

Select Driver

Press F1 in this dialogue box and enter the driver of disk in the input box popping up. Press F1 to confirm, and all software upgrade packs in the selected disk will the shown in the file list area.

Pack Up

Press F2, system will pack the software automatically and save packaged softwares to the selected disk.

Export

Press F6 to export software packs to the selected disk.

Upgrade

All the software update packs are listed in “update list under the disk directory”. Press “ ↑ ” and “ ↓ ” to select the desired update. Then press F7 to start software installation.

Back

Press F8 to back to interface **System Info**.

Dialog box “Upgrade” with “Upgrade File” selected is shown in Fig. 7-4.

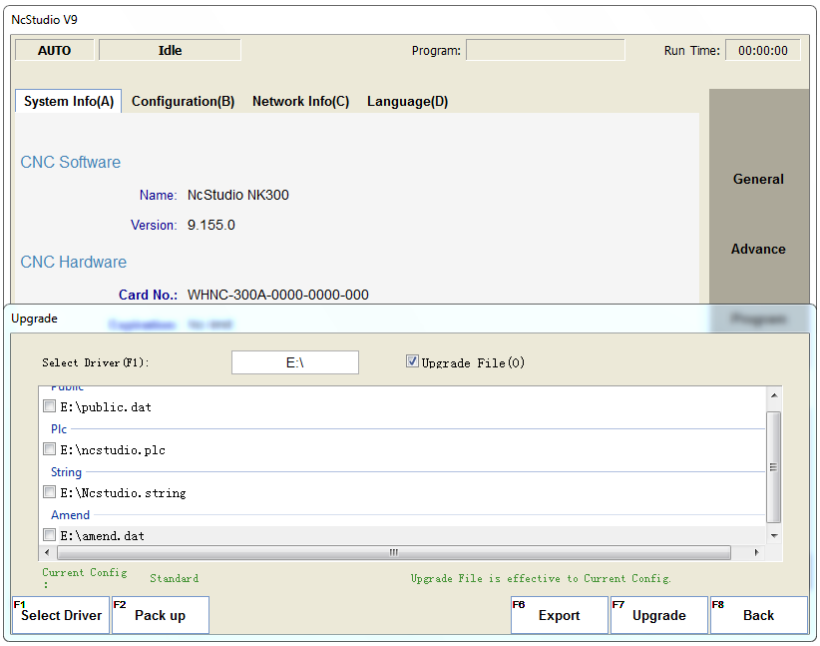


Fig. 7-4 Dialogue Box “Upgrade File”

Select Driver

Press F1 in this dialogue box and enter the driver of disk in the input box popping up. Press F1 to confirm, and all public files in the selected disk including “Public”, “Plc”, “Amend” and “String” will the shown in the file list area.

Pack Up

Press F2, system will pack the software automatically and save the packaged software to the selected disk.

Export

Press F6 to export the public files under current configuration to the selected USB disk. Prompt “Export Common Files Successed!” will appear when export process is successfully completed.

Upgrade

Press F7 to update the common files under configuration to the selected common files. If upgrade process is successful, a dialog box “You must reboot computer to make changes effective” will appear. Click “OK” to reboot the system.

Back

Press F8 to back to interface **System Info**.

7.3 Warning Information

Type	Warning Content	Cause	Solution
Warning prompt information	Simulation result shows that running scope of the program exceeds the mechanical stroke.	The scope of machining file exceeds upper and lower limits of work table, decided by setting “N10020”& “N10030” in system.	Modify the value of “N10020” and “N10030” to expand stroke range of work table (see section 3.2.3).
	The function can’t be used since backing to machine origin has not completed.	System has not returned to machine origin yet. If “N74001” is set “yes”, returning to machine origin before machining is a must.	Execute [back to machine origin] before using this function.
	The error of X (Y\Z) axis backing to machine origin is out of allowable range.	An error occurs in the precision of origin switch.	Detect the precision of origin switch.
		An error occurs in the precision of encoder origin.	Detect if origin signal of system encoder is right.
	This operation can’t be executed under current processing state.	Such illegal operation is executed in processing state as adjusting parameter.	Stop processing or execute those operations under idle state.
	This operation can’t be executed under simulation state.	Such illegal operation is executed in simulation mode as modifying parameters or pressing some shortcut keys.	Quit simulation mode or execute those operations under idle state.
Limit alarm	Positive (negative) limit of X (Y\Z) axis	The polarity of X-axis positive limit port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (refer to section 3.6).
		X-axis runs into limit switch directly during motion.	Manually move X-axis away from limit switch.
		There is an error in limit switch itself.	Check if limit switch works normally.
Return to machine origin alarm	The distance is too close between coarse and fine positioning switches if X (Y\Z) axis backing to machine origin.	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 origin switch and encoder origin to make the space within the range: [0 + “N74110” ~ screw pitch – “N74110”] (see section 3.4).

Type	Warning Content	Cause	Solution
Servo alarm	Servo alarm of X (Y\Z) axis	The polarity of X-axis servo alarm port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (refer to section 3.6).
		There is an error in X-axis servo driver itself.	Check if X-axis servo driver works normally.
E-stop alarm	E-stop button is pressed.	The polarity of E-stop port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (refer to section 3.6).
		The E-stop button is pressed.	Turn the E-stop button clockwise make it pop-up.
Oil level alarm	Oil level alarm	The polarity of oil level alarm port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (see section 3.6).
		When the oil level line in the oil pump is below a certain value, a signal will be sent to the system to give an alarm.	Check if the oil mass is too small in the oil pump.
Spindle alarm	Spindle alarm	The polarity of spindle alarm port is wrong.	Enter [I/O Port] function screen under [Log and Diagnosis], and modify the port polarity (see section 3.6).
		There is an error in inverter.	Check if the inverter works normally.
File error	There is no file.	Start file machining with no file loaded in the system in advance.	Load a machining file before start machining.
Pulse feedback alarm	The pulse feedbacks exceed the setting range of parameter.	It is used to detect if the D-value between the sent pulses and received pulses exceeds the setting value of parameter.	Check if the servo system is stable or the motor encoder is damaged.
	Alarm for over-travel	Alarm signal occurs in tool change over-travel	Check if the tool presetter

Type	Warning Content	Cause	Solution
Change tool over-travel alarm	in tool change	protection port	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 NK300 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 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.

7.4 Common Troubleshooting

7.4.1 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 is conducted with a multimeter and whether the analog voltage between SVC and GND is normal. 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, shut down 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 NK300 host.

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

1) Check whether the polarity of output port “Axis X/Y/Z Servo Enables”, in interface **I/O Port** of functional area **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.

7.4.3 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 connected to output port brake, BRAKE-COM, start the system, power on the machine tool with system alarm signal removed, and measure whether the port is conducted with a multimeter. If it is not conducted, please check the host machine; and if it is conducted, then the output of the port is normal.
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 terminal board.

7.4.4 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) In interface **I/O Port** of functional area **Diagnosis**, check and make sure the polarity of input port "Reference Point of X/Y/Z-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 X 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 X axis" changes.

3) Enter interface **Axis** of functional area **Parameter**, and check whether parameter "Home Search Dir.", "Home Latch 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.

   In interface **I/O Port** of functional area **Diagnosis**, check whether the polarity of input port "Reference Point of X-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 X/Y/Z-axis" in interface **I/O Port**.

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.

7.4.5 What should you do if band switch “Control Mode”, “SpindleRate” and “FeedRate” are not in accordance with the actual situations?

1) Firstly, check whether the polarity of band switch input port “Control Mode”, “SpindleRate”, and “FeedRate” are set as “NO”.
2) Secondly, check if the wiring between the panel and host machine is firm: close the host machine, then unplug and re-plug the connection line between the panel and host machine, and then see if the host machine works normally after restarted.
3) Finally, exchange a panel to see whether something is wrong with the panel and system.

7.4.6 What should you do if handwheel control is abnormal?

The frequently encountered problems in handwheel operation: a certain axis moved by handwheel keeps still; a certain axis can still motion controlled by handwheel in the situation of OFF gear selected; handwheel gear does not match the actual condition.

1) Check if the polarities of input signals “Select X/Y/Z-axis by Handwheel” and “Handwheel Ratio X1/10/100” are set correctly in interface **I/O Port** of functional area **Diagnosis**.
2) Check if the software can receive the input signals of “Select X/Y/Z-axis by Handwheel” and “Handwheel Ratio X1/10/100” normally in interface **I/O Port** of functional area **Diagnosis**. If not normally, check whether the handwheel and system go wrong by exchanging another handwheel.

7.4.7 What should you do if a machine tool moves upward after arriving at tool presetter position during tool measurement?

1) In interface **I/O Port** of functional area **Diagnosis**, check whether the ploarity of input port “Tool Presetter Signal” is in accordance with the signal type of the port.
2) Manually press down the tool presetter and check whether the polarity of port “Tool Presetter Signal” in interface **I/O Port**. If it the polarity does not change, the tool presetter must have been damaged.

8 Drivers

8.1 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.

8.1.1 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	To be Calculated	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	To be Calculated	
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
$0\sim2^{20}$	_ (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	$0\sim2^{30}$	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.
	$0\sim2^{30}$	$0\sim2^{30}$	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.

8.1.2 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	To be Calculated	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)	To be Calculated	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)	To be Calculated	

8.1.3 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)	To be Calculated	$\frac{Pn20E}{Pn210}=\frac{Encoder\ resolution\times Pulse\ equivalent\times Deceleration\ ratio}{Screw\ pitch}$ 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, $\frac{PN20E}{PN210}=\frac{2^{13}\times 0.001\times 60}{360}=\frac{8192}{6000}=\frac{512}{375}$
Pn212	Pulse No. allocated by encoder	To be calculated	Range: 16~230. Concrete value is decided by PG divider ratio.

8.1.4 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)	To be Calculated	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.

Para. No.	Function	Value	Description
Pr48	1st numerator of command pulse frequency multiplication	To be Calculated Range: 1~10000	Typical value: pitch 5 mm, encoder resolution 10000, shaft coupling direct drag, pulse equivalent 0.001 mm: Pr48＝10000 Pr4B＝pitch 5mm / pulse equivalent 0.001mm＝5000 Pr48/Pr4B=10000/5000=2/1
Pr4B	Denominator of the command pulse frequency multiplication	To be Calculated Range: 1~10000	

8.1.5 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	To be Calculated Range: 0~2^{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	To be Calculated 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: $\text{Command pulse No. per motor circle} = \frac{\text{Screw pitch}}{\text{Pulse equivalent} \times \text{Mechanical deceleration ratio}} = \frac{5\text{mm}}{0.001\text{mm/p}} = 5000$ When screw pitch is 5mm and pulse equivalent 0.001, the value of Pr0.08 is "5000".			

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

Pr0.08	Pr0.09	Pr0.10	Description
$0\sim2^{20}$	_ (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	$0\sim2^{30}$	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.
	$0\sim2^{30}$	$0\sim2^{30}$	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.

8.1.6 Parameter Setting of MITSUBISHI MR-JE Servo Driver

Para. No.	Code	Function	Value	description
PA01	*STY	Operation mode	XXX0	_ _ _x: select position control mode.

Para. No.	Code	Function	Value	description
PD24	MBR	Output assignation to CN1-23 pin	XX05	_ _ xx: select MBR (electromagnetic brake interlock).
PA06	CMX	Electronic gear numerator	To be Calculated	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	To be Calculated	
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	To be Calculated	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.

8.1.7 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.

Para. No.	Code	Function	Value	Description
				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	To be Calculated	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	To be Calculated	
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	To be Calculated	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, LSP-ON and LSN-ON automatically selection	0110	Bit 0: Servo-ON selection. [0]: servo on by external input; [1]: servo on all the time inside. 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.

8.1.8 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	To be Calculated	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	To be Calculated	
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) 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	X2X1X0	100	External EMG stop input is not used.

Para. No.	Function	Format & Range	Value	Description
	setting for digital input pin DI8			
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.

8.1.9 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 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.

Para. No.	Function	Format & Range	Value	Description
				X1X0=00: position control mode
P1-44	Electronic Gear Ratio (Numerator) (N1)	1~32767	To be Calculated	$\frac{P1\text{-}44}{P1\text{-}45}=\frac{\text{Encoder resolution}\times\text{Pulse equiv}\times\text{Decelerat. ratio}}{\text{Pitch}}$ When encoder resolution is 1280000, screw pitch 5mm, pulse equivalent 0.001, in direct coupling, $\frac{P1\text{-}44}{P1\text{-}45}=\frac{1280000*0.001}{5}=\frac{256}{1}$ 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	To be Calculated	
P1-46	Detector output pulse No. setting	20 ~ 320000	To be Calculated	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.
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;

Para. No.	Function	Format & Range	Value	Description
				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.

8.1.10 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
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	To be Calculated	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
P1-45	Electronic Gear Ratio (Denominator)(M)	1~32767	To be Calculated	

Para. No.	Function	Format & Range	Value	Description
				used, P2-60 ~P2-62 are not required.
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+.

8.1.11 Parameter Setting of DELTA ASDA-B2 Servo Driver

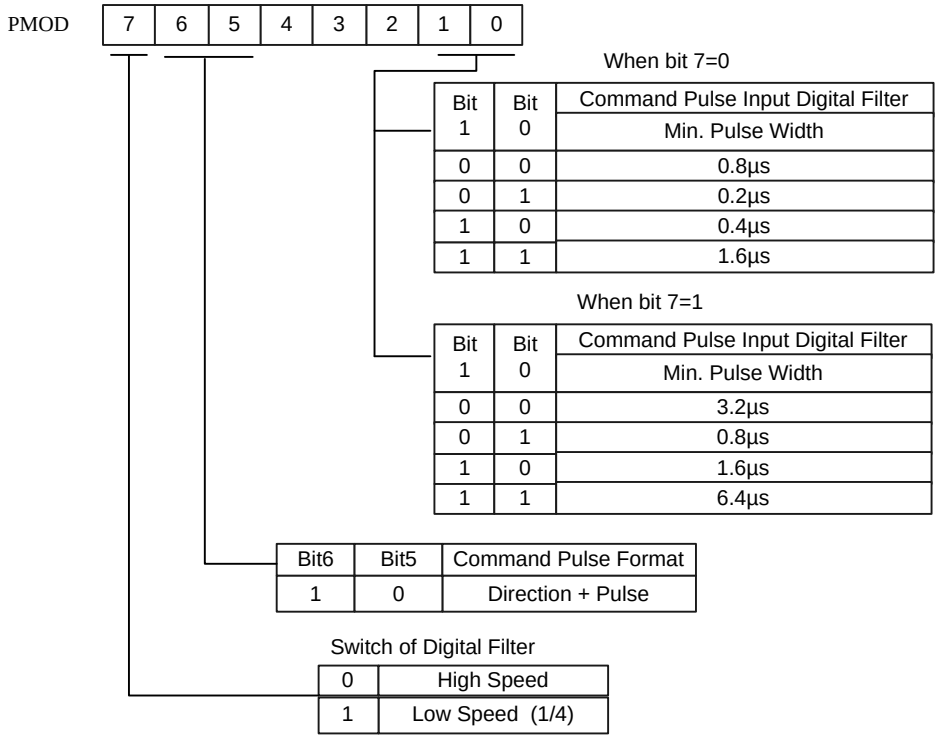
Para. No.	Function	Format & Range	Value	Description
P0-02	Driver status display	-	02	Monitor if the number of sent and received pulse is correct by setting this parameter. In Weihong control system, the correct quantity of pulse sent by control card is detected by pulse inspection in order to determine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

Para. No.	Function	Format & Range	Value	Description
				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	To be Calculated	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	To be Calculated	
P1-46	Detector output pulse No. setting	20~40000	To be Calculated	It sets detector output pulse number, whose concrete value is decided by PG divide ratio. When pulse equivalent = 0.001mm/p, pitch=10mm, without reducer, sets this parameter to 10000; when pitch=5mm, sets it to 5000.
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 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

Para. No.	Function	Format & Range	Value	Description
				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.

8.1.12 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 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;		500 to 65535	P/R	

<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="2">2. Standard configuration of the encoder pulse No. is as below.
Incremental encoder omitting wiring: --2000P/R
Absolute encoder:--2048P/R</td><td></td><td></td><td></td></tr>
<tr><td>2-0</td><td>PMOD</td><td colspan="5">Pulse format of position command:
Our system uses: direction + pulse format, the parameters are shown as following:
 
</td></tr>
</table>

<table>
<tr><th>Para. No.</th><th>Abbr.</th><th>Name</th><th>Standard Value</th><th>Setting Range</th><th>Unit</th><th colspan="2">Remark</th></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><td>Selection Item</td><td>Content</td></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>

8.1.13 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			

Para. No.	Parameter Name	Set Value	Remarks
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

8.1.14 Parameter Setting of SANYO Q Servo Driver

Para. No.	Parameter Name	Set Value	Remarks
Group 1			
GER1	Electronic gear ratio 1	1/1	Set electronic gear ratio for position command pulse. 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.

Para. No.	Parameter Name	Set Value	Remarks
03	Incremental encoder resolution	2000	500－65535, set the encoder resolution manually.
08	Control mode selection	02	Select position control mode.

8.1.15 Parameter Setting of FUJI FALDIC-β Servo Driver

Para. No.	Name	Value	Description
01	Command pulse numerator α	To be Calculated 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	To be Calculated 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).

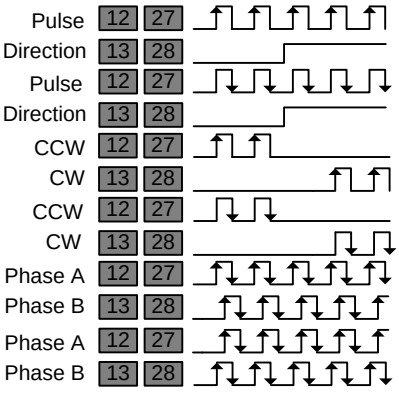
8.1.16 Parameter Setting of KT270 Servo Driver

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.
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,

Para. No.	Parameter Name	Value	Description
			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. 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.

8.1.17 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

<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>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;
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>

8.1.18 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="4">Pn010-1</td><td rowspan="4">Set control mode</td><td rowspan="4">1</td><td rowspan="2">Value</td><td colspan="2">Control mode</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>
</table>

Para. No.	Function	Value	Description		
			2	Torque control	Torque control
			3	Speed control	Speed control
			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, please refer to the driver manual of the specific brand.

8.2 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 8.2.2 and Figure 4 in section 8.2.6 (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).

8.2.1 Wiring Diagram of WISE Servo Driver

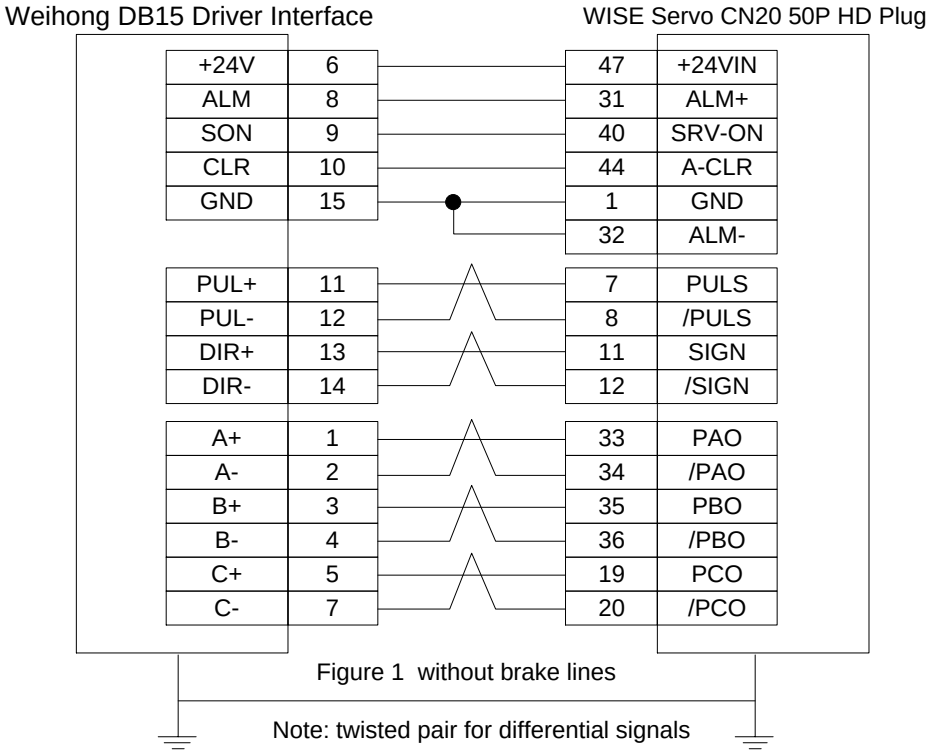


Figure 1 without brake lines

Note: twisted pair for differential signals

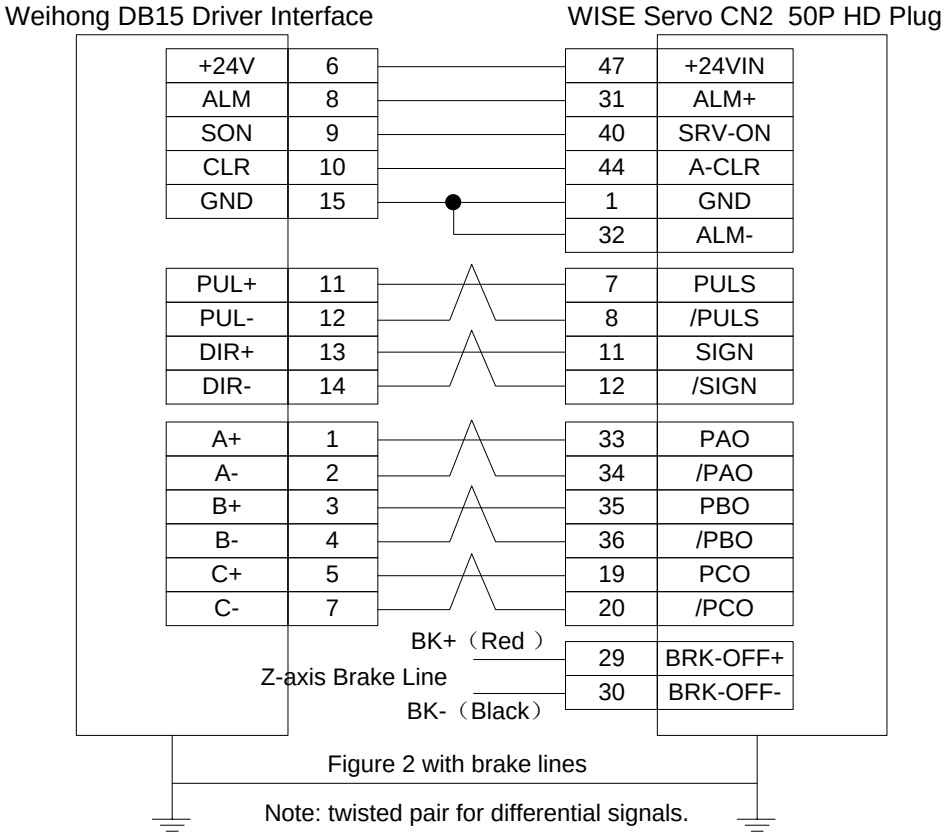


Figure 2 with brake lines

Note: twisted pair for differential signals.

8.2.2 Wiring Diagram of YASKAWA AC Servo Driver

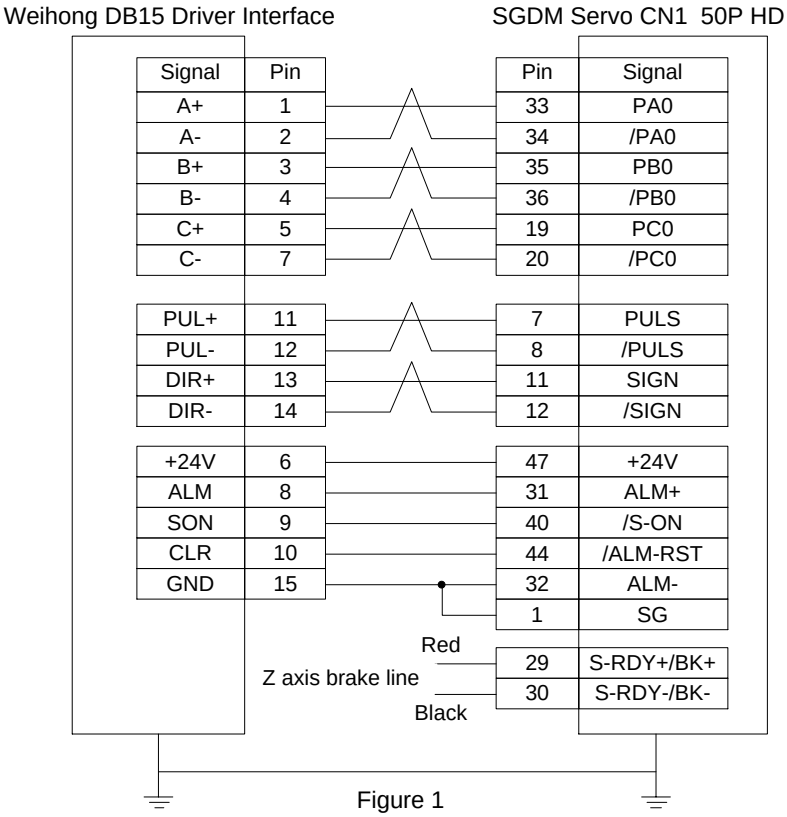


Figure 1

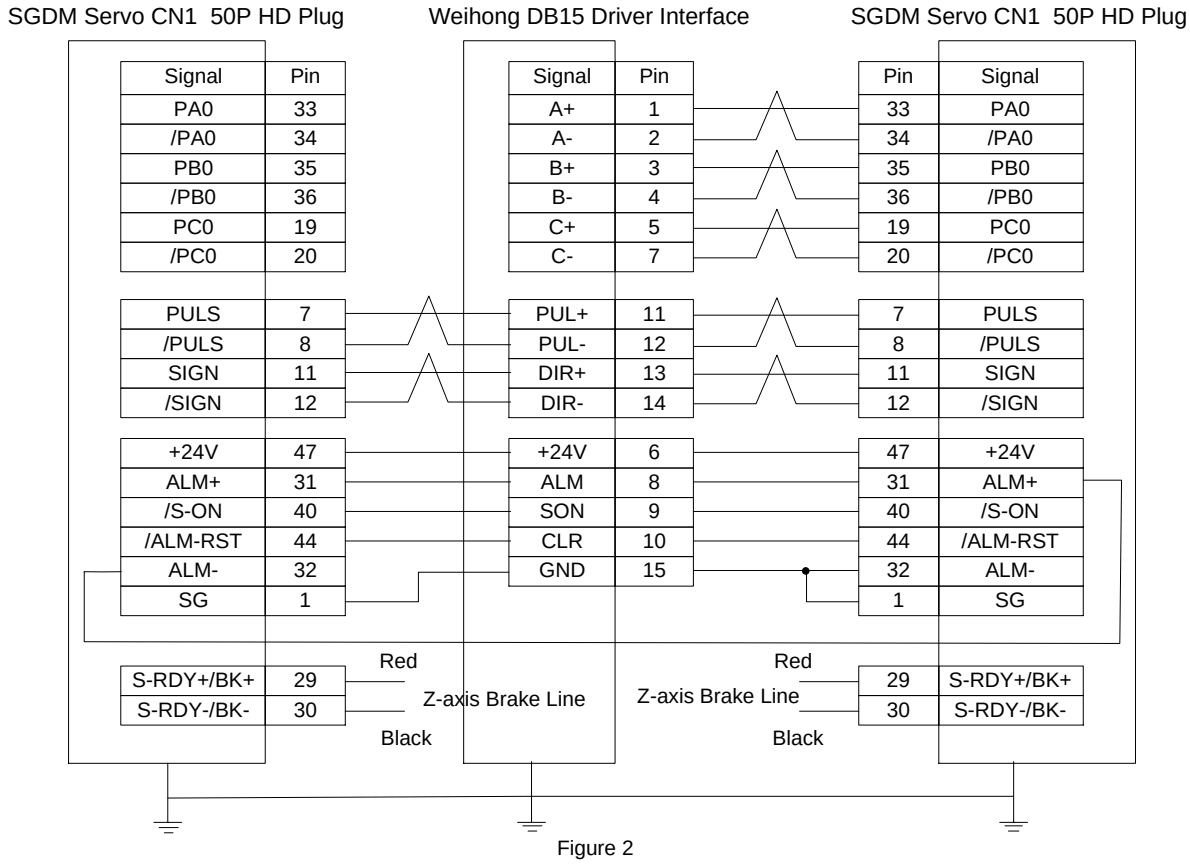
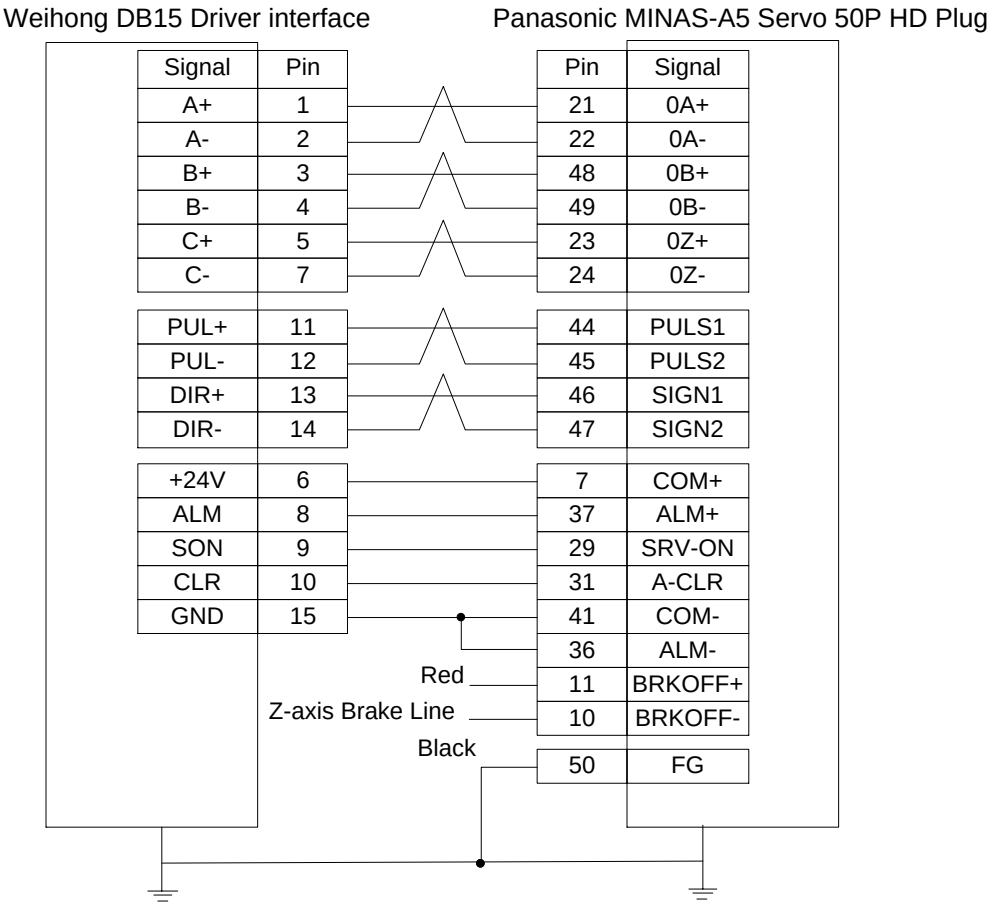


Figure 2

8.2.3 Wiring Diagram of PANASONIC AC Servo Driver



8.2.4 Wiring Diagram of MITSUBISHI MR-JE Servo Driver

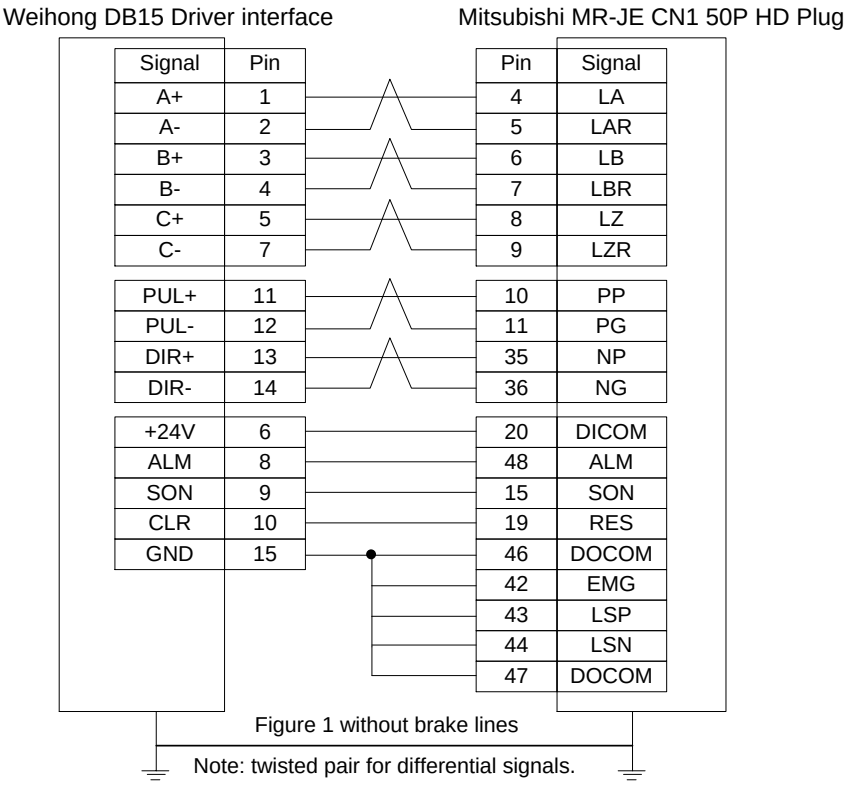


Figure 1 without brake lines

Note: twisted pair for differential signals.

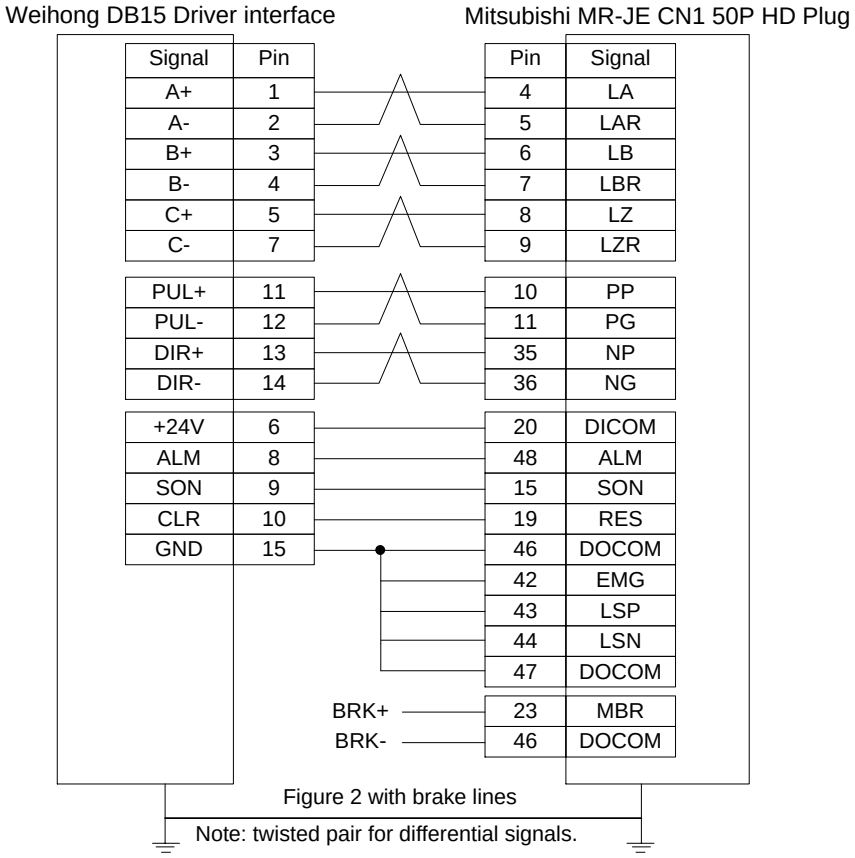
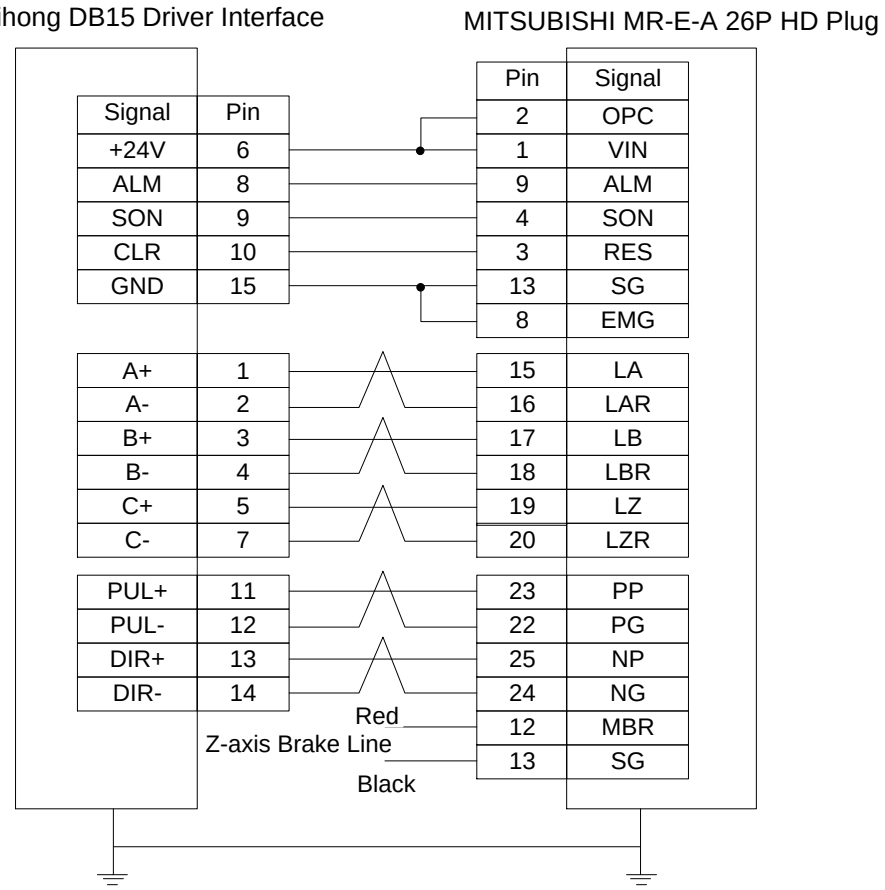


Figure 2 with brake lines

Note: twisted pair for differential signals.

8.2.5 Wiring Diagram of MITSUBISHI MR-E Servo Driver



8.2.6 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 8.1.8, 8.1.9, and 8.1.10.

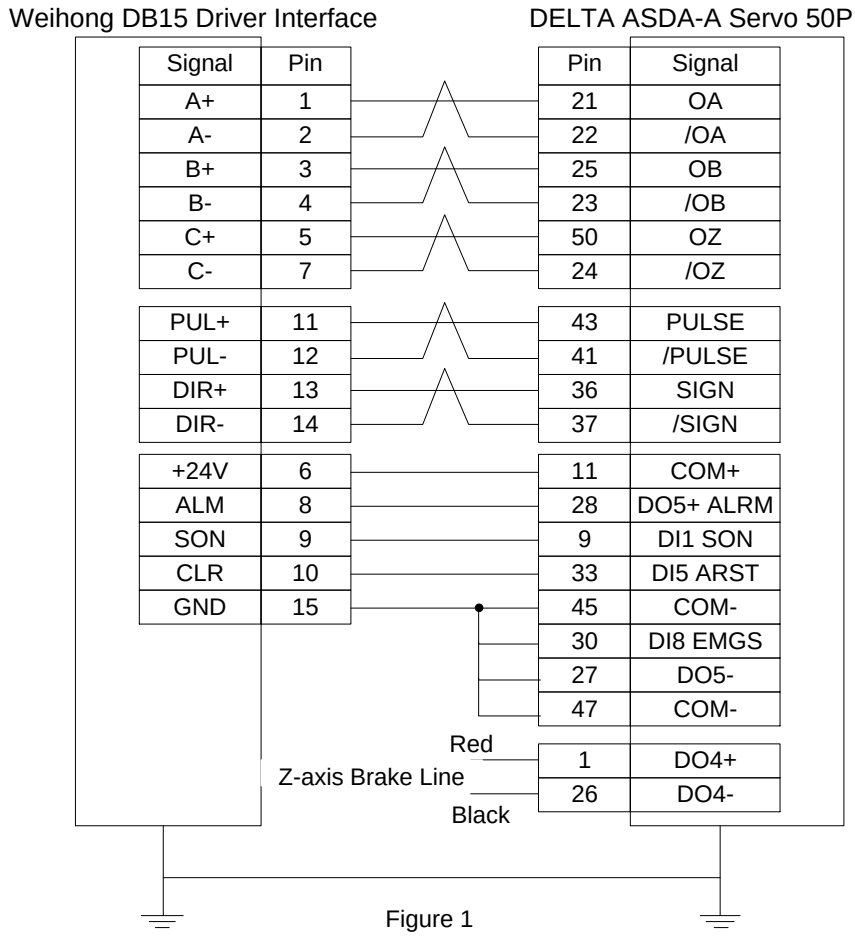


Figure 1

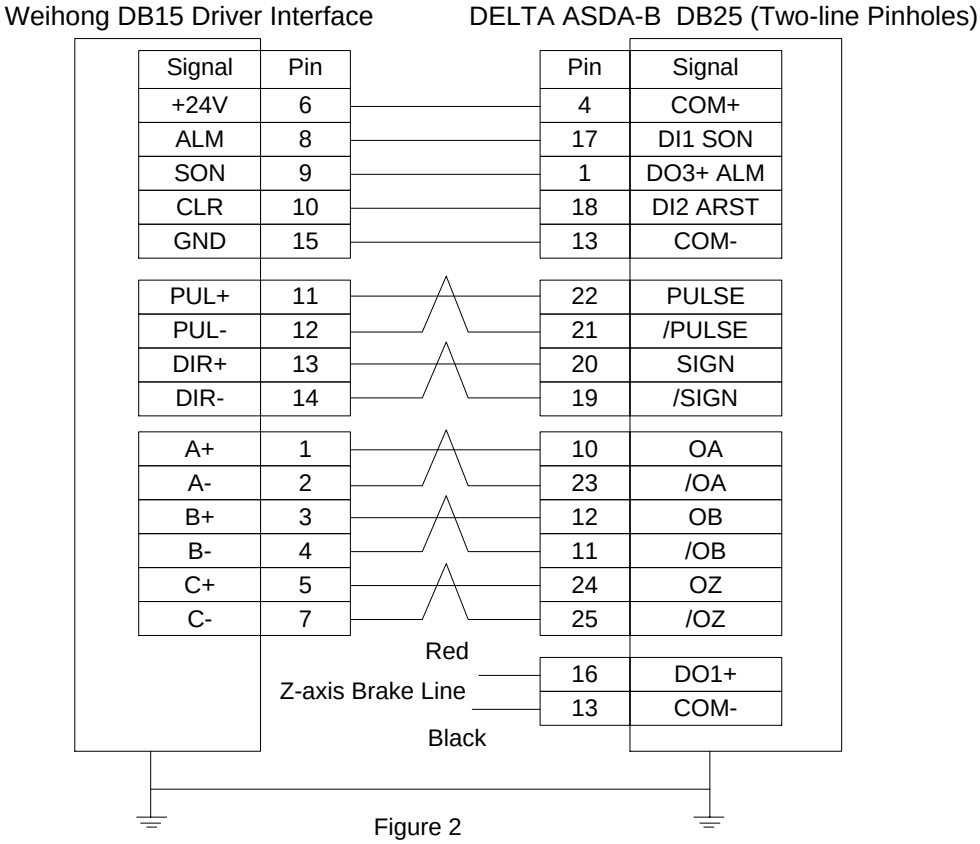


Figure 2

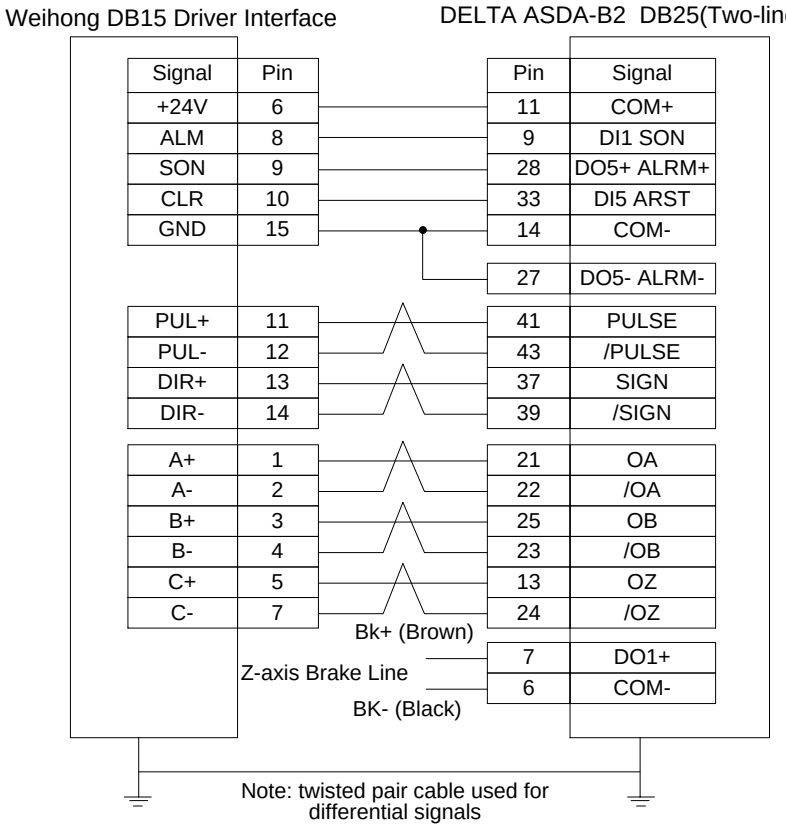


Figure 3

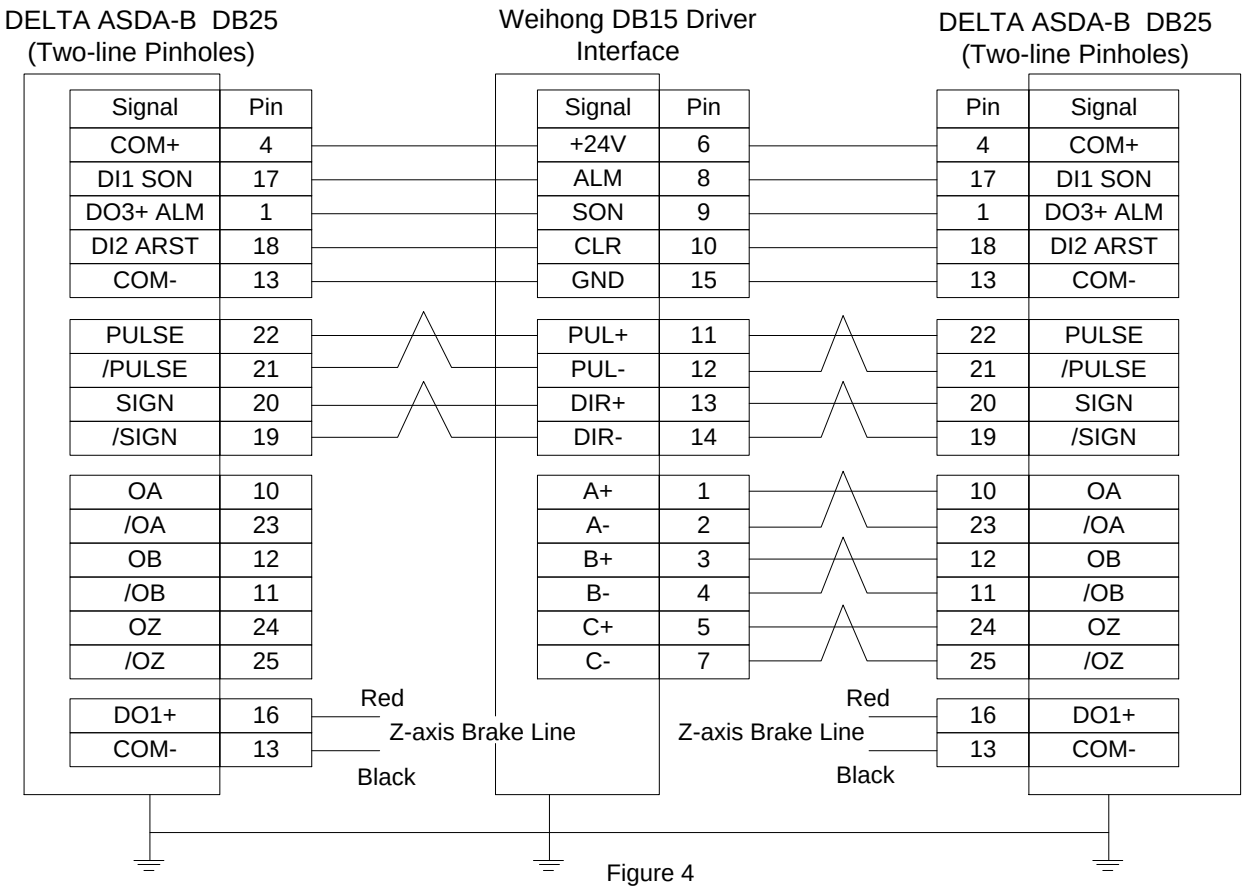
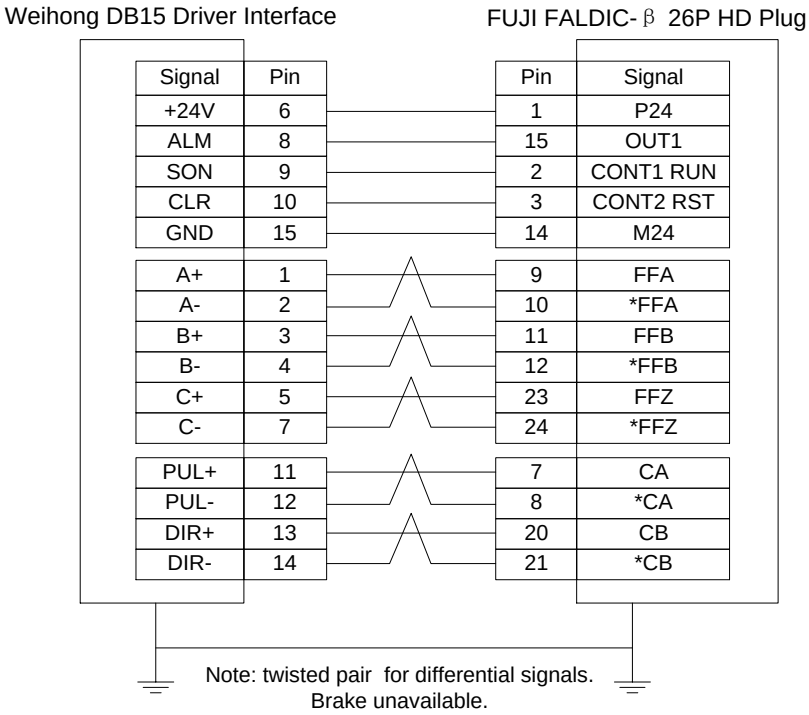
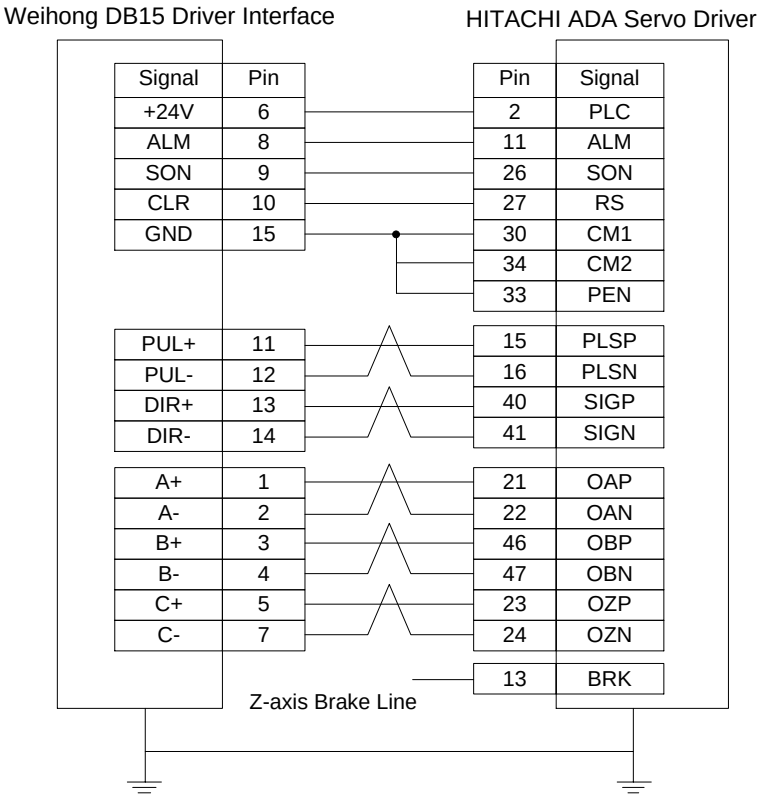


Figure 4

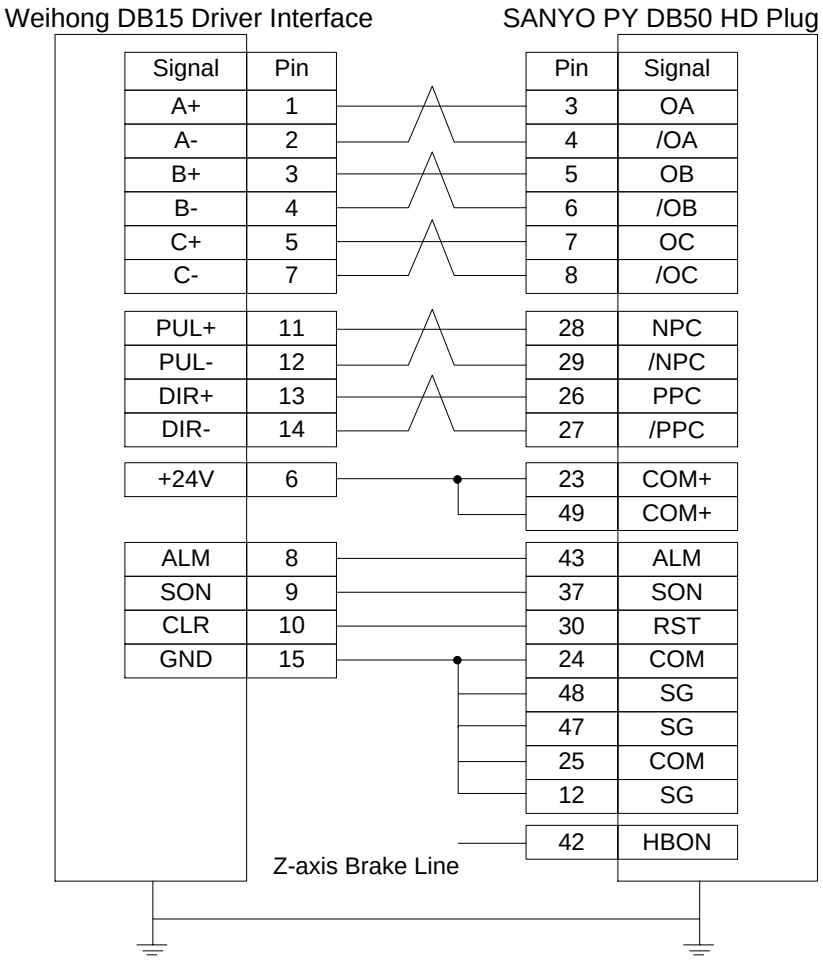
8.2.7 Wiring Diagram of FUJI Servo Driver



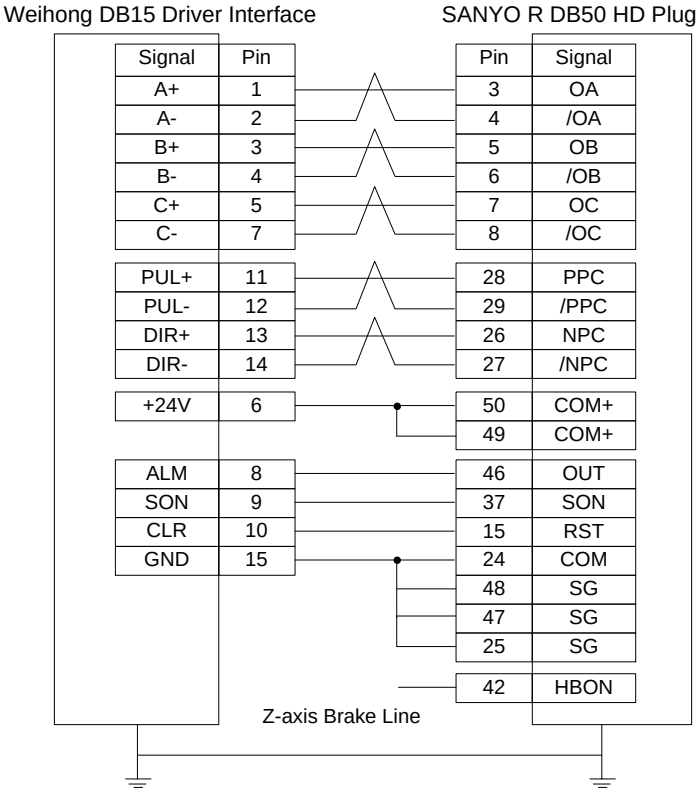
8.2.8 Wiring Diagram of HITACHI Servo Driver



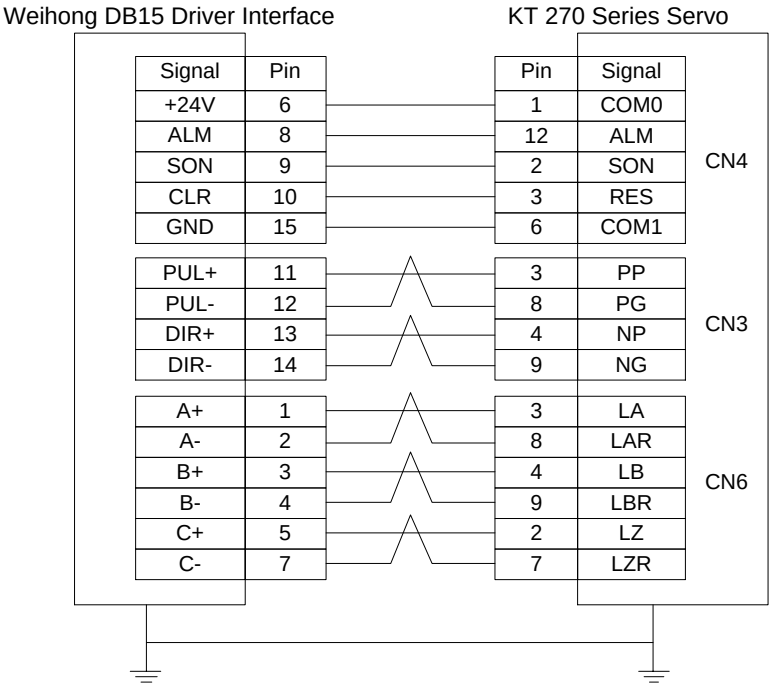
8.2.9 Wiring Diagram of SANYO PY Servo Driver



8.2.10 Wiring Diagram of SANYO R Servo Driver



8.2.11 Wiring Diagram of KT270 Servo Driver



8.2.12 Wiring Diagram of STONE GS Servo Driver

Weihong DB15 Driver Interface

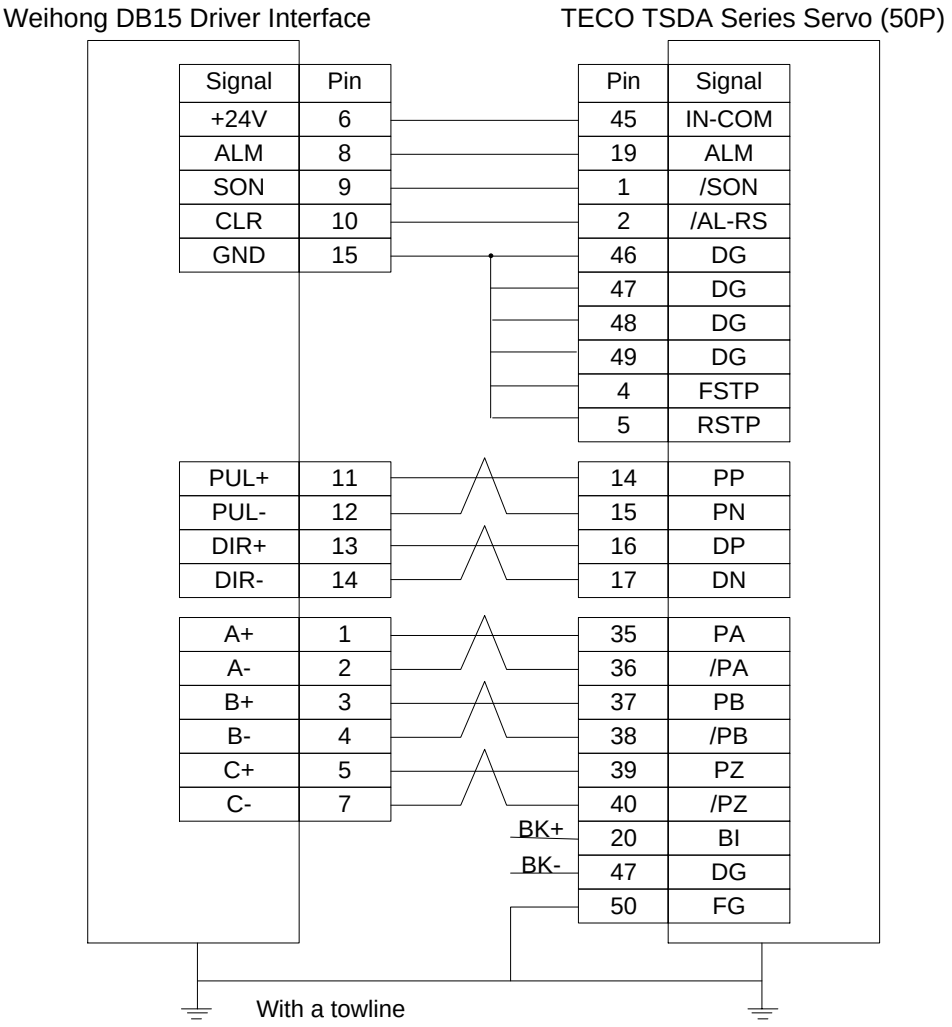
Signal	Pin
+24V	6
ALM	8
SON	9
CLR	10
GND	15
PUL+	11
PUL-	12
DIR+	13
DIR-	14
A+	1
A-	2
B+	3
B-	4
C+	5
C-	7

STONE GS Series Servo (3-line DB44 Pins)

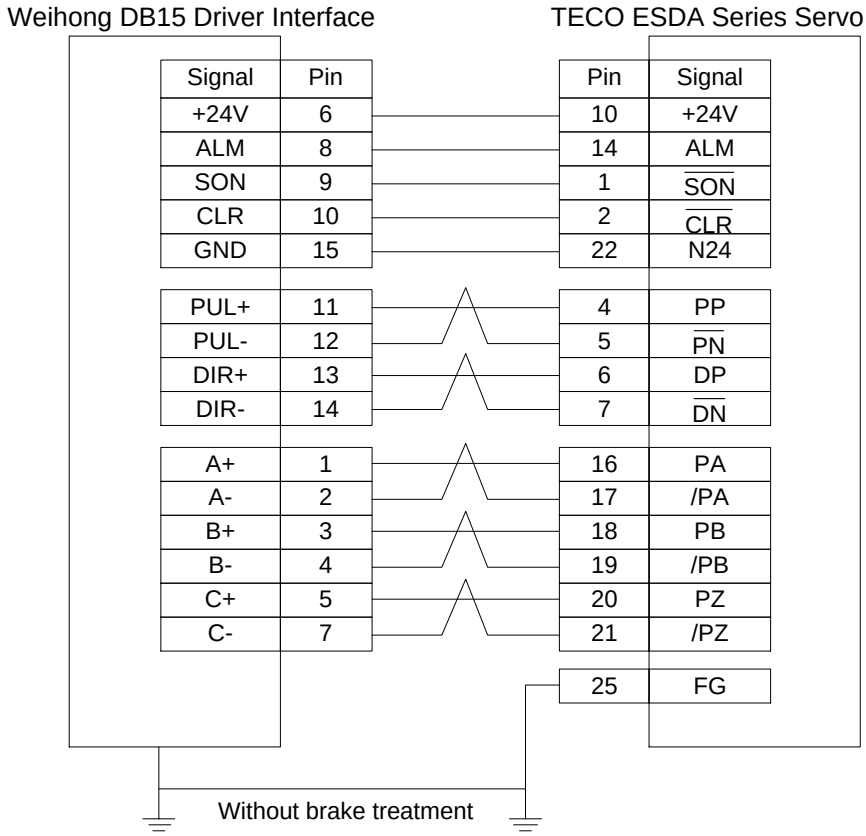
Pin	Signal
7	Input signal anode common port
22	Fault signal output+
23	Servo enable input
8	Alarm clear signal input
6	Fault signal output-
12	Pulse command signal input+
27	Pulse command signal input-
13	Direction/pulse command input +
28	Direction/pulse command input-
33	Signal differential output +
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

8.2.13 Wiring Diagram of TECO TSDA Servo Driver



8.2.14 Wiring Diagram of TECO ESDA Servo Driver



9 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	3.2.1
N10010	Pulse Equivalent (X/Y/Z)	-0.0000009~999 (mm/p)	0.001	Restart	3.2.2
N10020	TravelLimits-Negative(X/Y/Z)	-99999~99999 (mm)	X: 0 Y: 0 Z: -100	Restart	3.2.3
N10030	TravelLimits-Positive (X/Y/Z)	-99999~99999 (mm)	X: 800 Y: 600 Z: 0	Restart	3.2.3
N10040	Enable Travel Limits (X/Y/Z)	YES: enabled; NO: disabled	YES	Restart	3.2.3
N10050	Positive ToolMeas. Travel limits (X/Y/Z, or X/Y/Z1/Z2)	-99999~99999 (mm)	9999	Restart	3.7.2
N10055	Negative ToolMeas. Travel limits (X/Y/Z, or X/Y/Z1/Z2)	-99999~99999 (mm)	-9999	Restart	3.7.2
N10060	Enable ToolMeas. Travel Limits	YES: enabled; NO: disabled	No	Restart	3.7.2
1.1 Encoders					
N11110	Axis Encoder Dir	1: Increasing encoder value; -1: Decreasing encoder value	1	Restart	3.3.2
N11130	Check Encoder Error	YES: enabled; NO: disabled	YES	Restart	3.3.2
N11140	Static Tolerance	1~999999	500	Restart	3.3.2
N11150	Dynamic Tolerance	1~999999	500	Restart	3.3.2
N11160	Frequency Division Pulses of PG (X4)	1~999999	10000	Restart	3.3.2
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: enabled;	YES	Restart	3.3.2

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
		NO: disabled			
N11309	Delay in Setting REF Sign	0.5~5	0.5	Immediate	-
	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	3.12.1
N12001	Enable Backlash Compensation	YES: enabled; NO: disabled	YES	Restart	3.12.1
N12020	Turn On AQE Compensation	YES: enabled; NO: disabled	NO	Immediate	3.12.3
N12130	Time(Group 0 of 6 groups)	0~10(sec)	0	Immediate	3.12.3
N12131	Distance(Group 0 of 6 groups)	0~10(mm)	0	Immediate	3.12.3
N12132	Delay(Group 0 of 6 groups)	0~10(sec)	0	Immediate	3.12.3
N12133	Intensity(Group 0 of 6 groups)	0.01~0.99	0.75	Immediate	3.12.3
1.3 Velo/Acc limits					
N13000	Max. Axis F(X/Y/Z)	0.001~100000 (mm/min)	48000	Immediate	3.12.3
1.4					
N14001	Programming Units	0: Angle; 1: surface distance of the rotary workpiece	1	Immediate	3.16.1
N14002	Workpiece Diameter	1~3000 (mm)	10	Immediate	3.16.1
N14003	Rotary Y Max Velocity	0.001~9999(rpm)	600	Restart	3.16.1
N14004	Angular Acceleration	(deg/s^2)	500	Restart	3.16.1
N14005	Coordinate Units	0: Angle; 1: Millimeter	1	Immediate	3.16.1
2.0 Spindle					
N20001	Max. S	0~999999 (rpm)	24000	Restart	3.5
N20005	Spindle Cool Off Delay Time	0~600 (sec)	5	Immediate	3.5
N20006	Spindle Speed when Centering	0~100000(rpm)	500	Immediate	3.9
N20010	Spindle On Delay Time	0~60(sec)	5	Immediate	3.5
N20011	Spindle Off Delay Time	0~60(sec)	5	Immediate	3.5
4.1 Lubricate					
N41000	Auto Lubricate	YES: Auto on;	NO	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
		NO: Not auto on			
	It sets whether the system automatically opens lubrication pump periodically and fills lube.				
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 keyboard.				
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	-
N42007	Select Tool Mode	0; 1	0	Immediate	-
N42021	Set delay of prepare workpiece position	-	0	Restart	-
5.2 Handwheel					
N52001	Precise Pulse Counting	YES: Adopt; NO: Not adopt	NO	Restart	3.15.1
N52002	Handwheel Direction	1; -1	1	Restart	3.15.1
	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	3.15.1
N52004	Multiple At X10	0.001~10 (mm)	0.01	Restart	3.15.1
N52005	Multiple At X100	0.001~10 (mm)	0.1	Restart	3.15.1
N52006	HW Lead Gear (Numerator)	1~1000	1	Restart	3.15.2
N52007	HW Lead Gear (Denominator)	1~1000	1	Restart	3.15.2
N52010	Handwheel Acceleration	1~1000 (mm/s^2)	200	Restart	3.15.1
N52012	Deceleration when Switching Axis	YES; NO	YES	Restart	3.15.1
N52013	Forbid HW Reverse Guide	YES; NO	NO	Restart	3.15.1
5.3 Operation panel					

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N53004	Enable Jog Override	YES; NO	NO	Restart	-
6.2 G code options					
N62000	Deceleration Distance	0~999 (mm)	2	Immediate	3.14.3
N62001	Approach F	0.001~99999 (mm/min)	300	Immediate	3.14.3
N62020	Enable Arc IJK Programming	YES: enabled; NO: disabled	YES	Immediate	3.14.3
N62021	Arc Radius Tolerance	0~9999 (mm)	1	Immediate	3.14.3
N62022	Enable Tool Selection by G-code File	YES; NO	NO	Reload program	-
N62090	Exact Stop Tolerance	0~99	0.001	Immediate	-
	The tolerance for G09 code. The parameter can be used together with pulse equivalent and other parameters.				
N62410	Enable Tool Compensation	YES: enabled; NO: disabled	NO	Immediate	3.12.2
N62411	Tool Compensation Type	1: Normal type; 2: Intersect type; 3: Insert type	1	Immediate	3.12.2
N62412	Tool Compensation Direction	0: Null; 1: Left; 2: Right	1	Immediate	3.12.2
N62413	Interferometry Path Segments	1~5	3	Immediate	3.12.2
N62414	Enable Evade Interferometry	YES: enabled NO: disabled	NO	Immediate	-
N62730	G73_G83 Lifting Distance	-99999~99999 (mm)	0	Immediate	5.3
N62760	G76_G87 Stop Orientation	0: G17 +X; 1: G17 –X; 2: G17 +Y; 3: G17 -Y	0	Immediate	-
6.3 Trajectory					
N63000	Look Ahead Distance	0~999	0.5	Immediate	-
N63001	Connect Speed LEP Lookahead Distance	1~1000	0	Immediate	-
N63002	Delay for Exact Stop	0~999 (s)	0	Immediate	3.10.3
N62003	Max COON angle Num	0~0.064	100	Immediate	-
N63006	Path Smoothing Time	0~0.064 (s)	0	Immediate	3.10.3
N63007	Trace Pretreatment Options	0: not disposed; 1: tolerate; 3: smooth	0	Immediate	-
N63008	Trace Pretreatment Precision	0~0.1	0	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N63009	Max Angle of Trace Pretreatment	0~180	180	Immediate	-
6.4 Velocity/Acc					
N64000	Startup Speed	0~600(mm/min)	0	Immediate	3.10.3
N64060	Max Feedrate	0~100000 (mm/min)	48000	Immediate	-
N64101	Rapid Motion Axial Acceleration	0.001~100000 (mm/s^2)	800	Immediate	3.10.3
N64102	Z-axis Acceleration	0.001~100000 (mm/s^2)	800	Immediate	3.10.3
N64103	Speed Up Acceleration	0.001~100000 (mm/s^2)	800	Immediate	-
N64104	Speed Down Deceleration	0.001~100000 (mm/s^2)	800	Immediate	-
N64120	Acceleration for Corners	0.001~100000 (mm/s^2)	3800	Immediate	3.10.3
N64150	Axial Jerk	0.001~1e+011 (mm/s^3)	150000	Immediate	3.10.3
N64200	Smoothing The Path Velocity	YES: enabled; NO: disabled	YES	Immediate	-
N64201	MAX Angle Smooth Velocity	0~180	90	Immediate	-
	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.				
N64203	Path Interpolation Algorithm	1; 2; 3	3	Immediate	3.10.3
	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 Velocity in LEP	0~100000	60	Immediate	-
N64207	Arc Velocity Limit	YES: enabled NO: disabled	YES	Immediate	-
N64208	Max Velocity of Reference Circle	0.001~100000 (mm/min)	3600	Immediate	3.10.3
N64209	Min Velocity of Arc	0.001~100000 (mm/min)	180	Immediate	3.10.3
N64240	Smoothing Time Factor	0.01~10	1	Immediate	3.10.3
N64241	Connect Speed	YES: enabled;	YES	Immediate	3.10.3

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
	decreased at Max Connect Angle	NO: disabled			
N64245	Pretreatment Path Number	1~2000	300	Immediate	3.10.3
N64246	Slide speed for small lines	YES: enabled; NO: disabled	NO	Immediate	3.10.3
N64247	Reference length of slide speed for small lines	0.001~10	1	Immediate	3.10.3
N64248	Enable slide optimization	YES: enabled; NO: disabled	YES	Immediate	3.10.3
N64249	Velocity Smooth for Single Axis	YES: enabled; NO: disabled	YES	Restart	-
6.5 File translation					
N65000	Retract (PLT)	0~99999 (mm)	5	Reload program	3.14.3
N65001	PLT Units	0.001~99999	40	Reload program	3.14.3
N65002	Tool Offset	0.0001~99999 (mm)	0.025	Reload program	3.14.3
N65003	Cutting Depth(PLT)	-99999~0 (mm)	0	Reload program	3.14.3
N65100	Retract (DXF)	0~99999 (mm)	5	Reload program	3.14.3
N65101	Cutting Depth(DXF)	-99999~0 (mm)	0	Reload program	3.14.3
N65102	Layer Depth	-99999~0 (mm)	0	Reload program	3.14.3
N65103	First Point As Origin	YES: use; NO: do not use	YES	Reload program	3.14.3
N65104	By Contour	YES: enabled; NO: disabled	NO	Reload program	3.14.3
N65105	Enable Bottom Cutting	YES: enabled; NO: disabled	NO	Reload program	3.14.3
N65106	Use Metric	YES: Forcibly use; NO: Not forcibly use	NO	Reload program	3.14.3
N65200	Retract (ENG)	0~99999 (mm)	5	Reload program	-
N65201	Prompt For Tool Change	YES ; NO	YES	Reload program	3.14.3
N65203	Cutting By Tool Number	YES: use;	NO	Reload	3.14.3

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
		NO: do not use		program	
N65204	Deep Hole Cutting Type	0: Reciprocating chip removal 1: High-speed reciprocating chip removal; 2: Up to a safe altitude	0	Reload program	3.14.3
N65205	Lifting Distance	0~99999	1	Reload program	3.14.3
N65206	Force To Use Tool Compensation	YES: force; NO: do not force	YES	Reload program	3.14.2
N65207	Modify by Tool Number	YES: enabled; NO: disabled	NO	Reload program	-
N65208	Tool Deepen Type	0; 1	1	Reload program	-
	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	-
N65210	Ignore Coordination System Instruction	YES: enabled; NO: disabled	NO	Immediate	-
N65211	Z UpType after Drilling	0; 1	0	Immediate	-
	0: Up to R Plane; 1: Up to specified work coor position, just ENG file allowed				
N65212	Z Posafter Drill	-1000~1000 (mm)	10	Immediate	-
N65213	Depth Tool Speed Way	0: Feedrate in machining 1: Feedrate in rapid traversing	0	Reload program	-
N65215	Force G00 Retract to Lift Height	YES: force; NO: do not force	NO	Reload program	-
6.6 Change tool					
N66000	Prompt for Tool Change	YES: enabled; NO: disabled	NO	Immediate	3.18.3
N66002	Pause in Toolchange for Same Active and Target Tool No.	YES: pause; NO: do not pause	NO	Immediate	3.18.3
	On condition that N66000 is enabled and this parameter set to YES, machine will pause when reach the same T number command in change; when set to NO, machine will not pause if the active and target T No. are the same.				

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N66005	Upper Position	-99999~99999 (mm)	0	Immediate	3.18.3
N66006	Lower Position	-99999~99999 (mm)	0	Immediate	3.18.3
N66007 ~N66009	Spindle Position (X/Y/Z)	-99999~99999 (mm)	9999	Immediate	3.18.3
N66010 ~N66012	Deceleration Position (X/Y/Z)	-99999~100000 (mm)	0	Immediate	3.18.3
N66015	Tool Chang Speed	0~100000 (mm/min)	3000	Immediate	3.18.3
N66016	Z-axis Speed	0~13740 (mm/min)	1800	Immediate	3.18.3
N66017	Automatic Tool Measure	YES: enable Automatic tool measure NO: disable Automatic tool measure	YES	Immediate	3.18.3
N66020	Tool Magazine Type	0: No Tool Magazine 1: Disk Tool Magazine 2: Linear Tool Magazine	0	Restart	3.18.3
N66021	Tool Magazine Capacity	1~255	8	Immediate	3.18.3
N66022	Check ChangeToolNo	YES: check; No: do not check	YES	Immediate	3.18.3
N66030	Tool Count Port	Port Address	NA	Immediate	-
N66031	Tool Mag. Back to Origin Port	Port Address	NA	Immediate	
N66032	Tool Mag. CW port	Port Address	NA	Immediate	-
N66033	Tool Mag. CCW port	Port Address	NA	Immediate	-
N66034	Mag. CW to Origin Delay	0~5000 (ms)	0	Immediate	-
N66035	Mag. CW to Origin Delay	0~5000 (ms)	0	Immediate	-
N66036	Tool Count to CW Delay	0~5000 (ms)	0	Immediate	-
N66037	Tool Count to CCW Delay	0~5000 (ms)	0	Immediate	-
N66074	Tool 1 Coordinate X (there are 21 tools in total.)	(mm)	0	Immediate	3.18.1
N66075	Tool 1 Coordinate Y (there are 21 tools in total)	(mm)	0	Immediate	3.18.1
N66076	Tool 1 Coordinate Z (there	(mm)	0	Immediate	3.18.1

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
	are 21 tools in total)				
6.7					
N67000 ~N67002	Negative Change Tool Travel Limits(X/Y/Z)	(mm)	-10000	Immediate	3.18.3
N67010 ~N67012	Positive Change Tool Travel Limits(X/Y/Z)	(mm)	10000	Immediate	3.18.3
N67020	Enable Change Tool Travel Limits(MCS)	YES: check; NO: not check	NO	Restart	-
7.1 Manu					
N71000	Slow Jog Speed	0~N71001 (mm/min)	1200	Immediate	3.10.1
N71001	Rapid Jog Speed	0~N13000 (mm/min)	3000	Immediate	3.10.1
N71002	Max Jog Speed Before Back to REF Point	0~ “Rapid Jog Speed”	1200	Immediate	-
7.2 Auto					
N72004	Spindle Off when Cycle Stop	YES: on; NO: off	YES	Immediate	3.5
N72008	Spindle On when Cycle Start	YES: on; NO: off	YES	Immediate	3.5
N72009	Cycle Machining Interval	0~1000	10	Immediate	-
N72010	Enable Work Coordinate Limits	YES: enable; NO: disable	YES	Immediate	3.8.1
N72020	Negative Work Coordinate Limits	(mm)	-99999	Immediate	3.8.1
N72030	Positive Work Coordinate Limits	(mm)	99999	Immediate	3.8.1
7.3 Pause					
N73000	Z-axis Feedrate after Pause	0~100000 (mm/min)	600	Immediate	-
N73001	Z-axis Lifting Feedrate on Pause	0~100000 (mm/min)	600	Immediate	-
N73002	Z-axis Lifting Mode on Pause	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: do not stop	YES	Immediate	3.5

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N73006	Z-axis Lifting Pos in MCS	-100~0 (mm)	0	Immediate	-
N73007 ~N73009	Return to Fixture on Pause (X/Y/Z)	-99999~99999	0	Immediate	-
7.4 Return Machine Home					
N74000	Cancel REF Sign when reset	YES: cancel; NO: do not cancel	YES	Immediate	3.4.3
N74001	Back to REF Required	YES: required; NO: not required	YES	Immediate	3.4.3
N74002	Cancel REF Sign when E-stop	YES: cancel; NO: do not cacel	YES	Immediate	3.4.3
N74010	Machine Zero Position(MCS)	0~N10030 (mm)	0	Restart	3.4.3
N74020	Home Search Dir. (X/Y/Z)	1: Positive direction -1: Negative direction	X: -1; Y: -1; Z: 1	Immediate	3.4.3
N74030	Home Search Velocity(X/Y/Z)	0.001~10000 (mm/min)	1800	Immediate	3.4.3
N74040	Home Switch Inport Adddress	X: 00117; Y: 00120; Z: 00123	X: 00117; Y: 00120; Z: 00123	Immediate	-
	The input port of PLC address of coarse positioning switch of each axis.				
N74050	Home Latch Dir. (X/Y/Z)	1: Positive direction -1: Negative direction	X: 1 Y: 1 Z: -1	Immediate	3.4.3
N74060	Home Latch Velocity (X/Y/Z)	0.001~10000 (mm/min)	60	Immediate	3.4.3
N74070	Index Pulse Inport Address	X: 00000; Y: 00001; Z:00002	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	3.4.3
N74090	Home Latch Count	1~100	1	Immediate	3.4.3
N74100	Leadscrew Pitch	0~100 (mm)	5	Immediate	3.4.3
N74110	Min Distance of Coarse/Fine Switches	0~2.5 (mm)	1	Immediate	3.4.3
N74120	Coarse/Fine Pos Distance Tolerance	0~100 (%)	10	Immediate	3.4.3
N74130	Max Distance during REF Positioning	5~99999	50	Immediate	3.4.3

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
7.5 Tool Measurement					
N75000	Probe Input Port Addr	00031	00031	Immediate	-
	The PLC address of the input port Tool Presetter Signal.				
N75001	ToolMea Fine Speed	(mm/min)	60	Immediate	3.7.2
N75002	ToolMea Fine Time	1~99999	1	Immediate	3.7.2
N75020	ToolMea Result Tolerance	0~10	0.1	Immediate	3.7.2
N75023	Disconnect Probe Input Port	-1; 1	-1	Immediate	-
	The PLC address of output on I/O board, which system output a switch signal to Open/Close probe wring to disable/enable probe. 1: Close probe wring to disable/enable probe; -1: Open probe wring to disable/enable probe.				
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	YES: enable; NO: disable	YES	Restart	3.7.3
N75100	Mobile Probe Surface to WCS Z0	-1000~1000 (mm)	0	Immediate	3.7.2
N75201	Fixed Probe Surface to WCS Z0	(mm)	10	Immediate	3.7.3
N75203	Fixed Preset Speed	(mm/min)	300	Immediate	3.7.3
N75210	Fixed Tool Sensor Position (X/Y/Z)	-99999~99999(mm)	X: 0; Y: 0; Z: -1	Immediate	3.7.3
7.9 Operation others					
N79000	Z Down Feedrate Limitation Mode	0; 1; 2	0	Immediate	-
	0: No Limitation; 1: Limit when only Z axis down; 2: Limit when including Z axis down motionn				
N79001	Z Down Federate Limitation	0~100000 (mm/min)	480	Immediate	-
N79003	Safe Height	0~1000 (mm)	10	Immediate	-
N79004	Forced to Use the Z Feed	YES; NO	YES	Immediate	-
N79100	Stop Mode when Cycle Completed	0: Stay where it is; 1: Cutter moves to fixed point; 2: Cutter moves to zero in WCS.	0	Immediate	-

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
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	3.8.4
N80003	Support Program Task	YES: Support; NO: Not support	NO	Restart	3.14.3.3
N80004	Print Info	YES; NO	NO	Immediate	3.3.2
N80007	Page Jump After Load File	0: Jump to “Coor-Auto” Page; 1: Jump to :Coor-Program” Page	0	Immediate	-
N80018	Use New Frp Algorithm	YES: Use; NO: not use	NO	Immediate	-
N80032	Calibration Type	0: Mobile calibration; 1: Fixed calibration; 2: First/Exchanged calibration	1	Immediate	3.7
8.1 Position view					
N81000	Auto Load Graph	YES; NO	NO	Immediate	3.11.3
N81001	Max File Size	(KB)	1000	Immediate	3.12.3
N81004	Allow Spindle-On when centring	YES; NO	YES	Restart	3.11.3
N81005	Show Remain Time	YES; NO	YES	Immediate	3.11.3
N81006	Keep Register Type Remain	YES; NO	NO	Restart	3.11.3
N81007	Auto Restart After Register	YES; NO	YES	Immediate	3.11.3
N81010	Gradient Fill	YES; NO	YES	Immediate	3.11.3
N81011	Draw Workbench	YES; NO	NO	Immediate	3.11.3
N81012	Draw Grid	YES; NO	NO	Immediate	3.11.3
N81013	2D Mode	YES; NO	NO	Immediate	3.11.3
N81015	Clear On Loading	YES; NO	YES	Immediate	3.11.3
N81016	Draw WC Origin	YES; NO	NO	Immediate	3.11.3

Para. No.	Name	Setting Range	Default	Effective	Refer-ence
N81017	Draw MC Origin	YES, NO	NO	Immediate	3.11.3
N81018	Bkground Color 1	Select a color	0×00000000	Immediate	3.11.3
N81019	Bkground Color 2	Select a color	0×00000000	Immediate	3.11.3
N81020	G00 Color (running)	Select a color	0×0000FFFF	Immediate	3.11.3
N81021	G01 Color (running)	Select a color	0×00FFFF00	Immediate	3.11.3
N81022	G02 Color (running)	Select a color	0×00FFFF00	Immediate	3.11.3
N81023	G03 Color (running)	Select a color	0×00FFFF00	Immediate	3.11.3
N81032	G00 Color (loading)	Select a color	0×04000000	Immediate	3.11.3
N81033	G01 Color (loading)	Select a color	0×00600000	Immediate	3.11.3
N81034	G02 Color (loading)	Select a color	0×00600000	Immediate	3.11.3
N81035	G03 Color (loading)	Select a color	0×00600000	Immediate	3.11.3
N81045	Grid Color	Select a color	0×00800080	Immediate	3.11.3
N81046	Coordinate Color	Select a color	0×0000FF00	Immediate	3.11.3
N81049	WC Origin Color	Select a color	0×0000FFFF	Immediate	3.11.3
N81050	MC Origin Color	Select a color	0×0000FFFF	Immediate	3.11.3

10 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 mal-operation. 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.