NK280B Integrated CNC System

Manufacturers' Manual

1st Edition

(Exclusive to 3C industry)

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.weihong.com.cn/en/to download the latest PDF edition for free.

Preface

About This manual

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

Section	Contents
Preface	It introduces applicable product model, contact information and revision history.
Get Started	It introduces the hardware components, NK280B operational panel and software.
Basic Commissioning	It introduces commissioning for the machine tool and driver before machining.
Operation	It introduces main operations involved during machining.
System Maintenance	It introduces operations about the registration and update of system and software.
NK280B Parameter Reference	It introduces parameters related to main and frequently-used operations.
Software License Agreement	It is about software license agreement.

Applicable Product Model

This manual is applicable to NK280B integrated CNC system, exclusive to 3C industry. Refer to the table below for details.

Product Model	Remarks
NK280B integrated CNC system	Abbreviated as NK280B, this system supports bus control system. Functions of NK280B integrated CNC system differ according to business line and application industry. This manufacturers' manual is applicable to customers of WEIHONG 3C business line. If there is no special explanation, all images belong to general three-axis software.

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: www.weihong.com.cn/en/

Revision History

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

Date	Edition	Revision
2018.01	R1	This manual is written according to the following versions: • BOOT version: V1.3.3. • General Three-axis software: NK280B_通用三轴_8.8.4_1709291646. • General Double Z software: NK280B_通用双 Z_9.0.0_1712201005.weihong

Contents

Getting Started

Consisting of the hardware and the software, NK280B integrated CNC system can be used for tapping and drilling, engraving and milling, etc.

Functions of NK280B integrated CNC system differ according to business line and application industry. Note that this manufacturers' manual is applicable to customers of WEIHONG 3C business line.

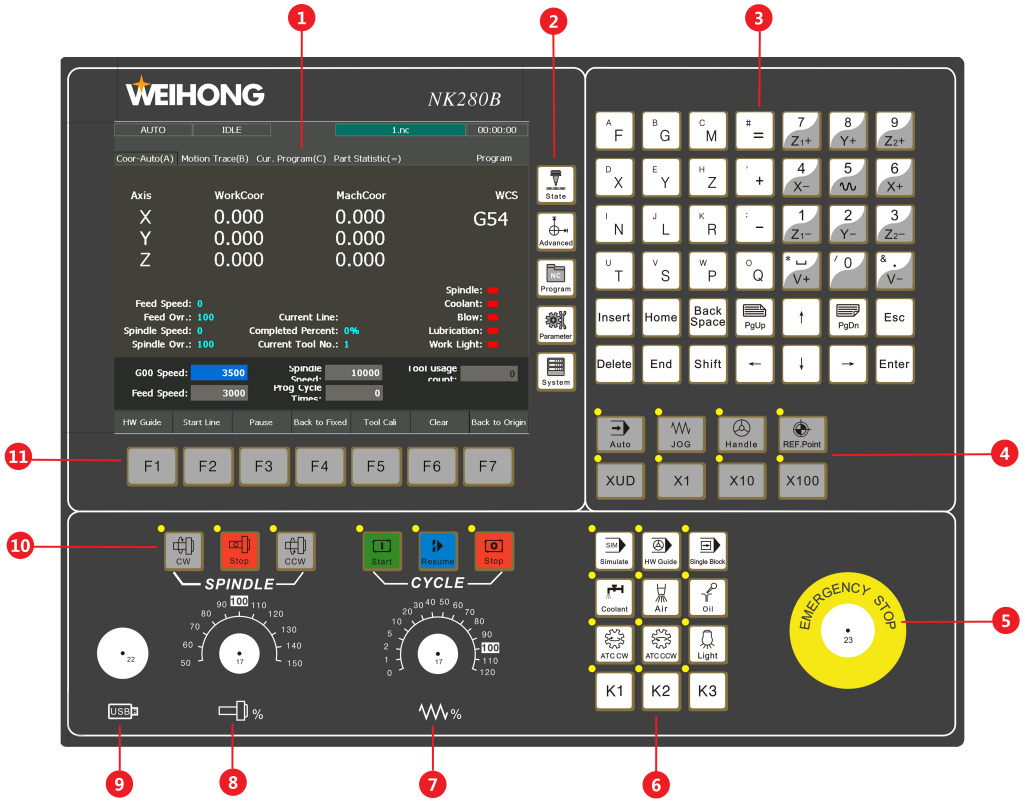
Hardware Components

- NK280B host
- Lambda 5M controller
- EX31A1 extended terminal board
- YASKAWA Σ5 / YASKAWA Σ7 / WISE bus-type servo driver
- NK-MPG-06 handwheel (optional)
- DB9M/F cable, M-Ⅱ bus-type cable

NK280B Operational Panel

Before operation, you need to know about hardware components of NK280B integrated CNC system.

See figure below for NK280B operational panel.



Knowing about NK280B front operational panel.

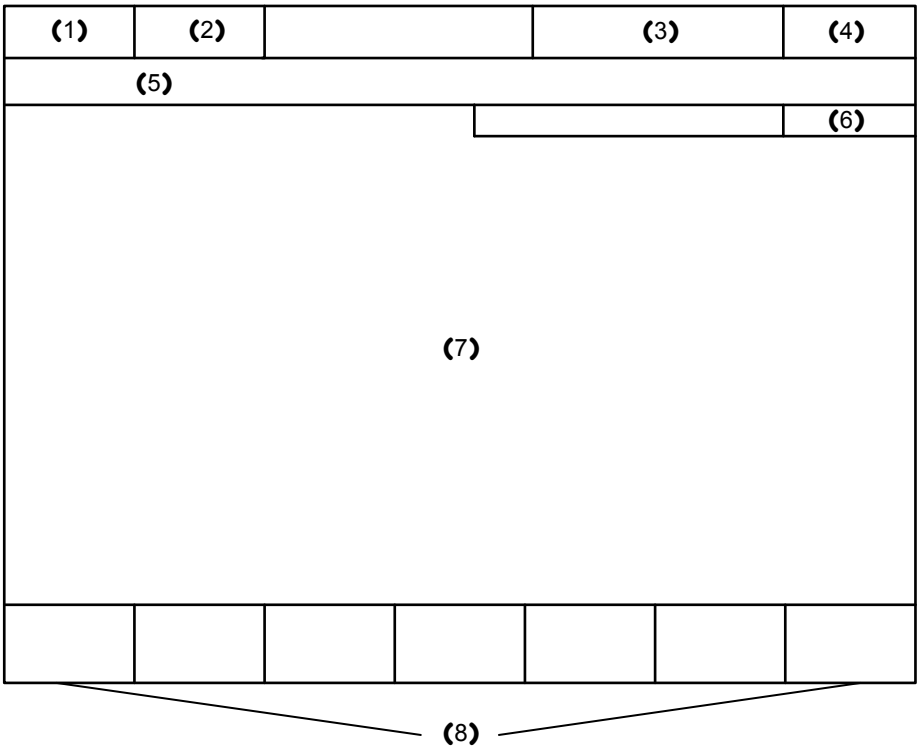
1	LED display
2	Main function access buttons

3	Alpha-numeric keypad
4	Buttons for mode switching
5	Emergency stop button (E-stop)
6	Buttons for auxiliary functions
7	Feed override switch
8	Spindle override switch
9	USB slot
10	Buttons for movement control
11	F1~F7

LED Display

LED display shows different interactive operational interfaces, namely the GUIs.

See figure below for outlines of the GUIs.



Outline

1: It shows the current operational mode, including Auto, Jog, Stepping, REF, etc.

2: It shows the current working state of the machine tool, including Idle, Running, ESTOP (Emergency Stop).

3: It shows name of current program file.

4: It shows running time.

5: It shows information such as working progress or alarms.

6: It shows menus.

7: It shows child interfaces of each main function access button.

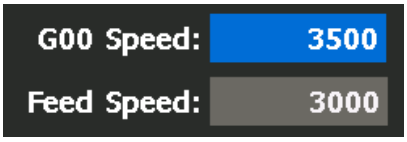
8: It shows F1~F7 buttons on different GUIs.

Shortcut Keys

You can use shortcuts to execute actions.

Press **B** to open **Motion Trace** interface.

Press arrow keys (↑, ↓, ←, →) to select setting items.

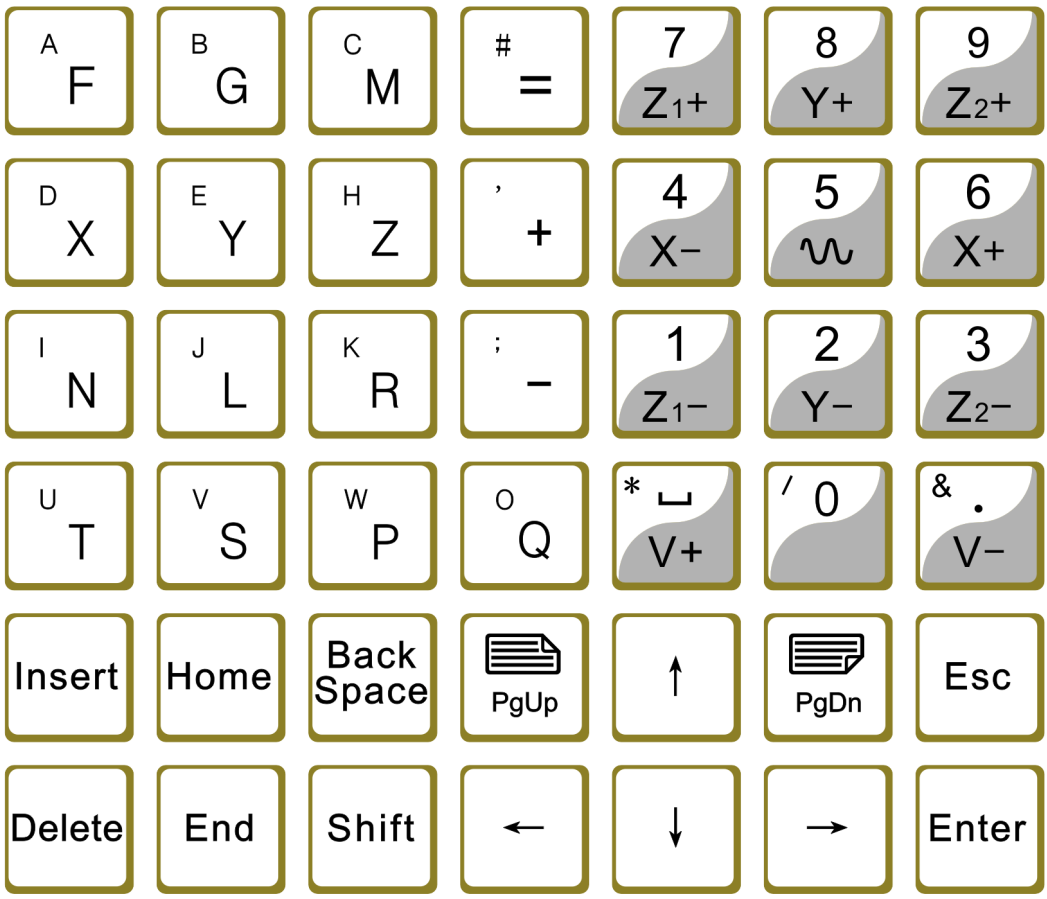


Main Function Access Buttons

Button	Remark
	It contains four child interfaces: **Coor-Manual**, **Motion Trace**, **Cur. Program** and **Part Statistic**. You can return to the machine origin, set spindle speed and feedrate, calibrate the tool, return to the fixed point, etc. F1~F7 buttons differ in different operating mode.
	It contains four child interfaces: **Coor- Manage**, **Centering**, **User Code Input** and **Technics Management**. You can set the workpiece offset and public offset, make tool compensation and screw error compensation, etc.
	It contains three child interfaces: **Local Program**, **USB File**, **Prog Wizard**. You can load the program, edit the program or set up the program wizard.
	It contains four child interfaces: **Machine Param**, **Param Backup**, **Coor Backup**, **Driver Param**. You can modify parameter settings and execute auto adjustment.
	It contains four child interfaces: **Port**, **Log**, **System Info** and **Data Collect**. You can set the IO ports, check logs and register the system, etc.

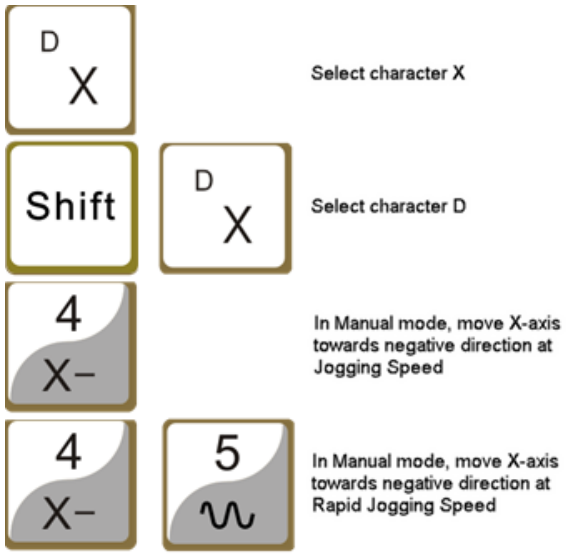
Alpha-numeric Keypad

See figure below for alpha-numeric keypad on the operational panel.



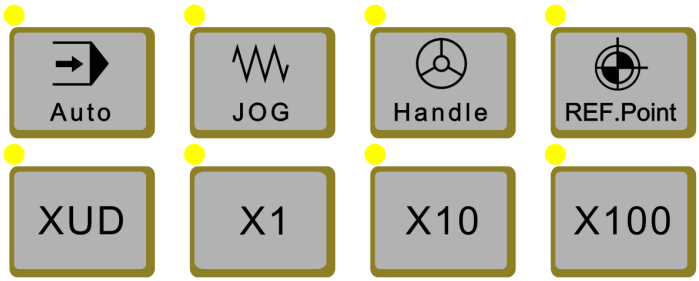
How to use:

- Directly press the button to enter the alphabet on the right-lower part.
- Press Shift + alphabet button to enter the alphabet on the left-upper part.
- In Manual mode, numeric buttons can be used as axis direction buttons. Only press the axis direction button, the axis moves at jogging speed; press the axis direction button and rapid button（“5”）together, the axis moves at rapid jogging speed.



Buttons for Mode Switch

See figure below for buttons.



NK280B supports five modes: Auto, Jog, Handle, REF.Point and Stepping mode.

When you want to control the machine movement by handwheel, select Handle mode.

Default override for Handle and Stepping mode is X100.

Select **XUD** to custom the step length.

ESTOP Switch

When the machine is in danger, press emergency stop button to completely stop the machine.

When the danger is cleared, rotate the switch in clockwise direction to remove emergency alarm.

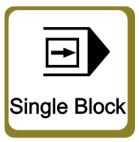
Buttons for Auxiliary Functions

Buttons are used for directly enabling some frequently-used operations or turning on or off some ports.

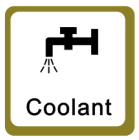
Buttons are:

Make the program run in simulation mode.

Enable handwheel guide function.



Enable the single block function. Once the button is pressed, one line of code block will be executed.



Enable or disable the coolant port to turn on or off the coolant.

Enable or disable the blowing port to start or stop blowing.

Enable or disable the lubricant port to turn on or off the lubricant.

Enable or disable the port to make the tool magazine rotate in clockwise direction or stop the tool magazine.

Enable or disable the port to make the tool magazine rotate in counter-clockwise direction or stop the tool magazine.

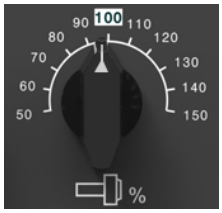
Enable or disable the port to turn on or off the light.

K1~K3 buttons are reserved for customization, whose functions can be customized.

Override Switch

It contains two types of override switch:

Adjust feed override from 0~120%.



Adjust spindle override from 50~150%.

Relationship between current speed and override is shown below.

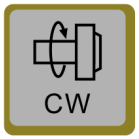
Current feedrate = feedrate × current feedrate override

Current spindle speed = spindle speed × current spindle override

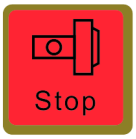
Buttons for Movement Control

It contains buttons for spindle control and program running.

Buttons are:



Start the spindle rotation in clockwise direction.



Stop the spindle rotation.

Start the spindle rotation in counter-clockwise direction.

Start programming running.

Resume programming running. In case of power interruption or emergency stop in machining, if the workpiece origin is secured, press the button to resume program running from the exact interrupted position.

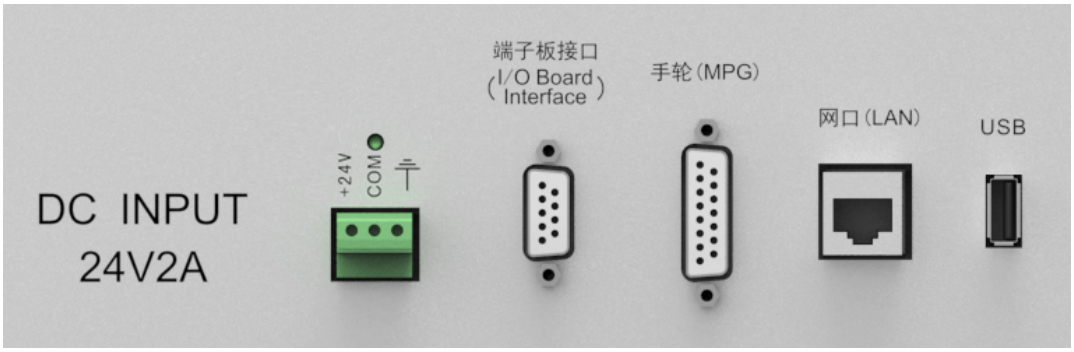
Stop program running.

Operational Buttons

F1~F7 buttons below the LED display.

Interface on the Back

Connect to correct component according to the following figure.



Software

At present, the software contains two types, according to active axes.

General Double Z Software

The software is used for machine tools with two worktables and two Z-axis. It contains two types of configuration according to movement of Z1-axis and Z2-axis.

- Linkage configuration: Z1-axis and Z2-axis can be active at the same time.
- Alternative configuration: Only Z1-axis or Z2-axis can be active at one time.

See Select Z-axis for details.

General Three-Axis Software

Use the software for normal machine tools with X-axis, Y-axis and Z-axis.

Functions such as “Axis Selection” and “Set Tool No.” are absent here.

Basic Commissioning

Before machining, you need to conduct commissioning to the tool machine and set motion control system.

After the system is correctly connected and powered on, you can start commissioning according to the following steps:

Table 1: Overview of Basic Commissioning Process Before Machining

Step	Operation	Reference
1	According to driver’s brand used, set station address and adjust servo gain for the driver.	Execute Auto Adjustment
2	Check port information, detect status and wiring of machine tool and adjust port polarity.	Check and Modify Port
3	Adjust axis direction.	Adjust Axis Direction
4	Return to machine origin.	Return to Machine Origin
5	Move machine tool and set related parameters.	Modify Parameters NK280B Parameter Reference
6	Set workpiece coordinate system (WCS), including workpiece offset and public offset.	Set Workpiece Coordinate
	Commissioning finishes. Wait for machining.	

Switch Operating Language

At present, Simplified Chinese and English are available.

To switch operating language in use:

1. Press **System** > **C** to open **System Info** interface.
2. Press **F3** to choose target language.
3. Restart the software.

   The target language takes effect.

Execute Auto Adjustment

Auto adjustment aims to automatically adjust servo gain for servo unit according to the current status of machine tool, and thus to optimize response.

In NK280B, auto adjustment for X-axis, Y-axis and Z-axis will be executed separately. Auto adjustment will not be executed until the auto adjustment for last axis is done.

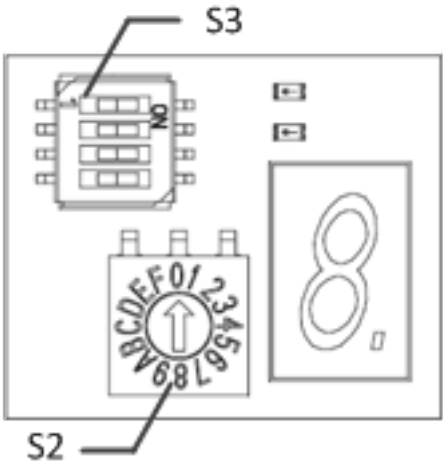
Due to the different brands of drivers, process of auto adjustment differs.

Set Station Address for Driver

Setting station address for driver is the prerequisite of data transmission among control system, lambda controller and driver.

Set Station Address for YASKAWA Driver

The setting of station address for YASKAWA bus-type driver is done through combined actions of rotary switch(S2) and toggle switch(S3).



To set station address for YASKAWA driver:

1. Optional: Modify setting of No.3 for S3.

Other numbers of S3 adopt factory settings.

Setting illustration for S3 is as shown in the following table:

No.	Illustration	Factory Setting
1	Setting for communication speed. • OFF: 4Mbps (M1) • ON: 10Mbps (M2)	ON
2	Setting for bytes transmitted. • OFF: 17 bytes • ON: 32 bytes	ON
3	Setting for station address. • OFF: actual address=40H+S2 • ON: actual address=50H+S2	OFF

No.	Illustration	Factory Setting
4	System reservation (unalterable).	OFF

2. Rotate S2 to the needed value.

Actual station addresses are as shown in the following table:

S3 No.3	S2	Station Address	S3 No.3	S2	Station Address
OFF	0	Invalid	ON	0	50H
OFF	1	41H	ON	1	51H
OFF	2	42H	ON	2	52H
OFF	3	43H	ON	3	53H
OFF	4	44H	ON	4	54H
OFF	5	45H	ON	5	55H
OFF	6	46H	ON	6	56H
OFF	7	47H	ON	7	57H
OFF	8	48H	ON	8	58H
OFF	9	49H	ON	9	59H
OFF	A	4AH	ON	A	5AH
OFF	B	4BH	ON	B	5BH
OFF	C	4CH	ON	C	5CH
OFF	D	4DH	ON	D	5DH
OFF	E	4EH	ON	E	5EH
OFF	F	4FH	ON	F	5FH

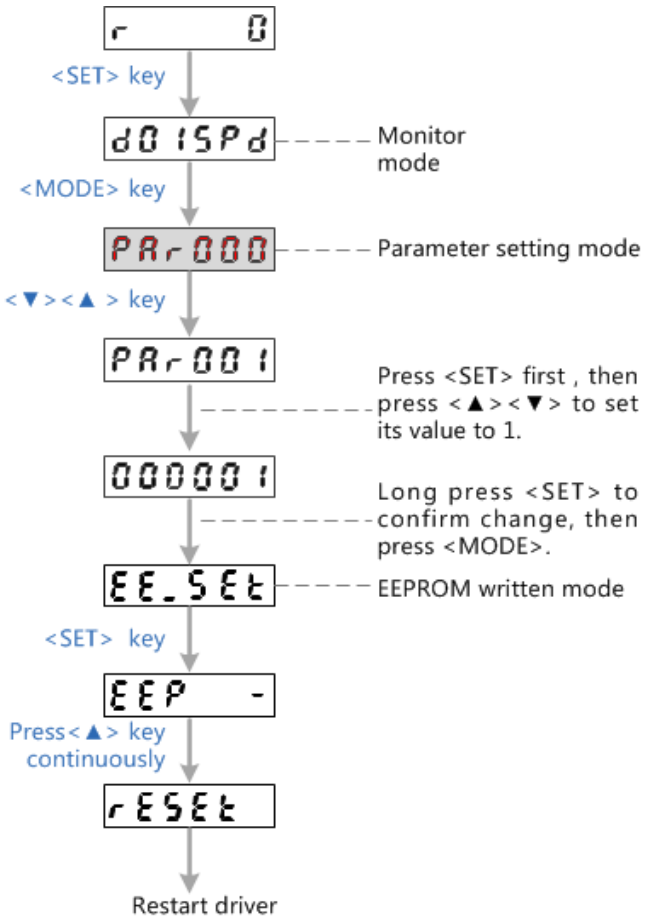
3. Restart the driver to make the settings effective.

Set Station Address for Wise Driver

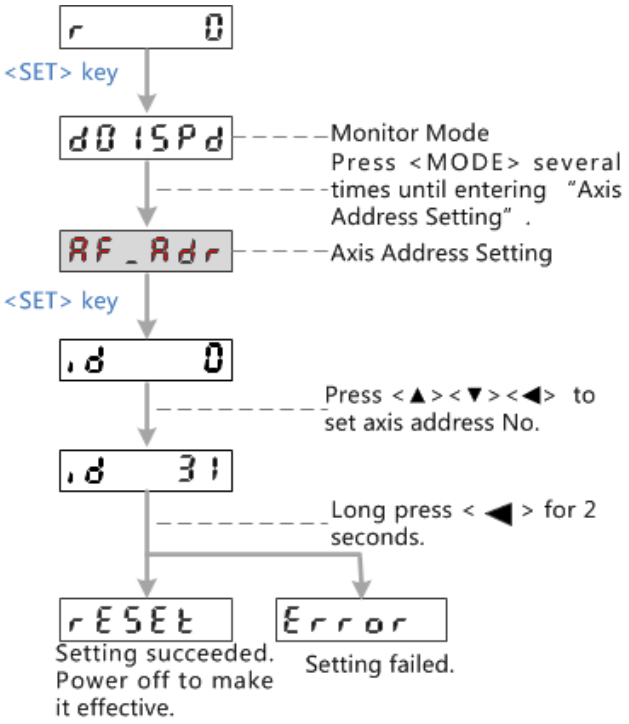
The setting of station address for Wise driver is done through the panel in front of the driver.

To set station address for Wise driver:

1. Set the value of driver parameter Pr001 Control Mode Setup to 1.



2. Set station address No.



It is suggested to set station address number in order, eg.: X-axis: 1; Y-axis: 2; Z-axis: 3.

When the No. is 0, it means communication function is disabled.

In the same control system, every station address No. is unique.

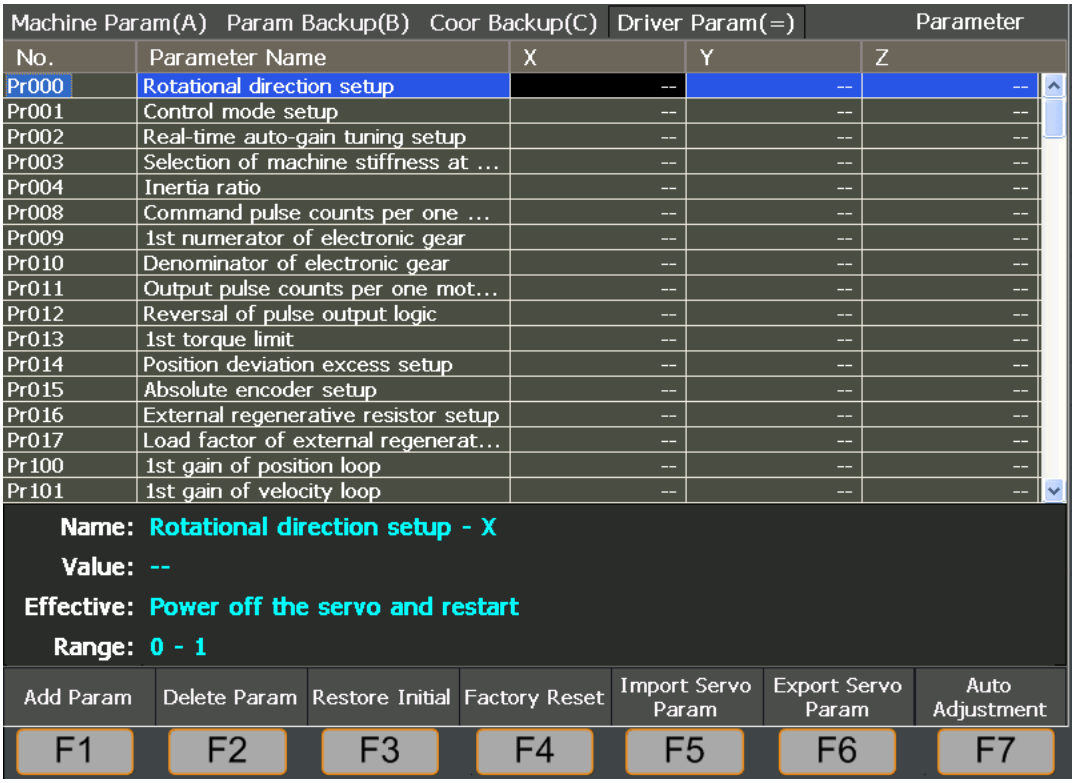
3. Restart the driver.

Set Driver Parameters

To set driver parameters:

1. Press **Parameter** > **=**.

   Driver Param interface shows.



2. To modify driver parameter, pressing direction key ↑/↓/←/→ > **Enter**.

   Due to different driver brands, setting of parameter differs. Setting value of Driver Type in software shall prevail.

3. Do one of the following:

Keys	Explanation
F1	Add other parameters that do not show. **Note:** Wise driver shows all parameters while YASKAWA driver shows only a part of them.
F2	Delete parameters unnecessary to show on the **Driver Param** interface.
F3	Restore all parameters to its initial status, that is, all the added parameters will be invalid.
F4	Restore settings of all parameters to factory settings.
F5	Directly import parameter files from mobile devices like USB flash disk to driver. **Note:**

Keys	Explanation
	You can only import files named ServoParam[number combination].dat.
F6	Export the set parameters file from driver to mobile devices like USB flash disk. **Note:** You can only name the exported files ServoParam[number combination].dat.
F7	Execute auto adjustment.

Execute Auto Adjustment

Different driver brands lead to different processes of auto adjustment.

In **Driver Param** interface, press **F7** to start auto adjustment.

Execute Auto Adjustment for YASKAWA Driver

Before executing auto adjustment, please ensure the following:

- Power of major loop is ON.
- No overtravel exists.
- No warning and alarm occurs.
- Parameter Automatic Gain Switching Selections 1 should be set to manual gain switch (parameter Pn139=n.□□□0).
- Parameter Function Selection for Test without a Motor should be invalid (parameter Pn00C=n.□□□0).
- Function of hardware base blockage (HWBB) should be invalid.
- Setting for parameter Parameter Written Prohibition should not be set to written prohibition (parameter Fn010= n.□□□0).
- Tuning-less Selection should be invalid (parameter Pn170=n.□□□0).
- Station address for driver should be consistent with the setting in control system software.

To execute auto adjustment for YASKAWA driver:

1. Set axis and moving range:
   a) Press shortcut key **T**.
   b) Select the axis for adjustment.
   c) To set the first limit, manually move the selected axis to safe position and press **F5**.
   d) To set the second limit, manually move the axis to the second safe position and press **F6**.

   When auto adjustment starts, the axis should be at the position of the second limit. As a result, please do not move machine tool after the second limit is set; otherwise, you need to reset the second limit.
2. Press shortcut key **S/P** to select mechanism and mode.

   Mechanism type is determined by the driven mechanical factors, as illustrated in the following:

Setting Value	Description
Rigid System	Suitable for mechanism with higher rigidity, like rigidity system.
Belt	Suitable for mechanism with lower rigidity, like belt.
Ball Screw	Suitable for mechanism with higher rigidity, like rigidity system, or linear servo motor.

The explanation of mode is as illustrated in the following:

Setting Value	Description
Standard	Execute standard gain adjustment. Except adjusting gain, this mode can automatically adjust notch filter and A-type vibration reduction.
For Positioning	Execute positioning private adjustment. Except adjusting gain, this mode can automatically adjust model tracking control, notch filter and A-type vibration reduction.
Overshoot Resistance	In positioning use, it emphasizes the adjustment within range. Except adjusting gain, this mode can automatically adjust notch filter and A-type vibration reduction and vibration abatement.

3. Make gain adjustment.
   a) press **F1** to check the information.
   b) Start adjustment.

      During adjustment, you can press **F2** to stop gain adjustment.

   Auto adjustment completes. And the result shows in the interface and is written into related parameters aromatically.
4. Adjust the axis again and compare the result with the first result.
   - If the result is quite similar to the first, press **F7** to exit adjustment interface for the axis.
   - If not, in **Auto Adjustment** interface, execute adjustment several times for the same axis by pressing **F1**. If all results vary largely, manually adjust driver parameter.
5. Start adjustment for other axes.
6. Restart the driver.

The following is common troubleshooting for Auto Adjustment for YASKAWA Driver.

Phenomenon	Solution
Exception in returning adjustment command.	Retry.

Phenomenon	Solution
Turning failed.	• Enlarge the range of positioning completion. (Adjust the value of parameter Pn522. • Adjust velocity loop gain. If machine tool vibrates, turn down the setting.
Bad adjustment performance.	• Ensure no alarm and warning occurs in driver. • Ensure main circuit power of driver is ON. • Ensure the driver is within the range. If not, turn down the setting of position loop gain. • Ensure function of hardware base blockage is disabled.
Interrupted duo to exception.	Retry.
Bad inertia estimation.	Double the value of parameterPn324 Moment of Inertia Calculation Starting Level.

Execute Auto Adjustment for Wise Driver

Before executing auto adjustment, please ensure the following:

- Power of major loop is ON.
- No overtravel exists.
- No warning and alarm occurs.
- The setting of driver station address should be consistent with the setting in the software.

To execute auto adjustment for Wise driver:

1. Set axis and moving range:

a) Press shortcut key **T**.

b) Select the axis for adjustment.

c) To set the first limit, manually move the selected axis to safe position and press **F5**.

d) To set the second limit, manually move the axis to the second safe position and press **F6**.

When auto adjustment starts, the axis should be at the position of the second limit. As a result, please do not move machine tool after the second limit is set; otherwise, you need to reset the second limit.

2. Press shortcut key **S/P** to select initial mode and initial rigidity.

The explanation of initial mode is as illustrated in the following:

Setting Value	Description
1：Standard	This mode emphasizes stability. And it does not compensate changeable load and friction and does not apply gain switch.

Setting Value	Description
2：Positioning	This mode emphasizes positioning. No changeable load exists in horizontal axis. It is suggested to use machine like ball screw to make friction.
3：Vertical axis	Except these mentioned in positioning mode, this mode also compensates changeable load for vertical axis, which is convenient to restrict offset caused by positioning stability time.
4：Friction Compensation	Except these mentioned in vertical axis, this mode also drive axis through belt that has large friction to shorten positioning stability time.

The explanation of initial rigidity is as illustrated in the following:

Mechanism Type	Rigidity Range
Large-scale handling/transmission equipment	0 ~ 13
Belt	5 ~ 16
Manipulator	10 ~ 20
Ball screw + Belt	13 ~ 25
Direct-coupled ball screw or mechanism with high rigidity	18 ~ 31

3. Estimate inertia.

a) press **F1** to check the information.

b) Start adjustment.

During adjustment, you can press **F2** to stop gain adjustment.

The estimation result shows in the interface and is written into related parameters aromatically.

4. Do one of the following:

- If you are satisfied with the result, go to Step 8.
- If you are not satisfied, go the next step.

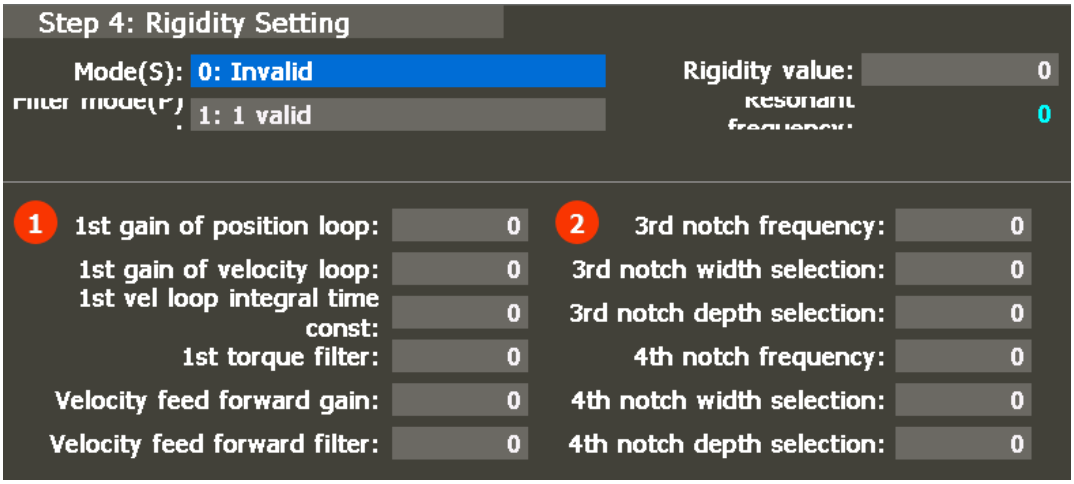
5. Set parameters of current interface.

6. Press **F2** to check running result.

- If you are satisfied, go to next step.
- If you are not satisfied, return to Step 6.

7. Set rigidity.

The interface is as shown in the following:



- When Mode is set as none-zero value, you can modify Rigidity and parameters in area ①; you cannot modify parameters in area ②. At this moment, parameters in area ① will alter, press **F1** to refresh.
- When Mode is set to 0, you cannot modify Rigidity, but you can modify parameters in area ① and ②.
- When Filter Mode is set to 1/2/3 and motor runs, you need to refresh Resonant Frequency in time.

8. Press **F4** to save.
9. Exit from current interface and start adjustment for other axes respectively.
10. Restart the driver.

The following is common troubleshooting for Auto Adjustment for YASKAWA Driver.

Phenomenon	**Solution**
Exception in returning adjustment command.	Retry.
Bad adjustment performance.	• Ensure no alarms and warnings occurs in driver. • Ensure main circuit power of driver is ON. • Ensure the driver is within the range. If not, turn down the setting of position loop gain. • Ensure station address is correct.
Interrupted duo to exception.	Retry.

Check and Modify Port

Through port information, you can monitor the status of machine tool, check electric circuit for its connection, conduct simulation test and etc. Though modifying the polarity of port, you can also clear some alarms.

To modify port:

1. Press **System** > **A**.

   Port interface shows.

2. Do one of the following:

Key	Explanation
F1 **F2**	These two keys are test-on and test-off. They are mainly for simulating hardware signal and conducting simulation test. Signal lamp in front of each port differs in test environment and actual environment. After pressing **F1**, input signal from outside will be blocked. And this is not suggested.
F3	The key is for canceling analog signal and simulation test. As a result, actual hardware signal replaces analog signal.
F4	The key is for modifying polarity of selected port. For output port, if there is no special explanation, generally, its polarity is N.
F5	If port attribute is modified as read-only, you cannot press **F1**/**F2** /**F4**.

Explanation for setting value of polarity and each icon in front of each port name is shown in the following table.

Item	Explanation
N	NC switch.
P	NO switch.
	It shows there is signal in input port.
	It shows there is no signal in input port.
	It shows there is signal in output port.
	It shows there is no signal in output port.
	It shows test is on.
	It shows test is off.

Adjust Axis Direction

Adjust the axis moving direction according to the Right Hand Principle.

To adjust the moving direction of each axis, taking X-axis as an example:

1. Check current setting value of Axis Direction (X) in parameter list.
2. Decide the positive direction of X-axis according to the Right Hand Principle.
3. Manually move X-axis to observe the moving direction.

- Press **Handle** button to activate Handwheel mode, and use handwheel to move X-axis.
- Press **Jog** button to activate Jog mode, and press **X+** or **X-** button to move X-axis.

4. (Optional) If actual moving direction is opposite with setting value, change to the opposite value.

   Current setting value for parameter Axis Direction (X) is 1, manually move X-axis but X-axis moves towards the negative direction. Change setting value to -1.

Related reference

Parameter for Axis Direction
Parameters that decide moving direction of each axis.

Operation

The Operation is designed to help users (mainly manufacturers) to understand main operations involved during machining.

You can start machining workpiece according to the following steps:

Table 2: Overview of Workpiece Machining Process

Step	Operation	Reference
1	Execute warm-up and trial run.	Execute Warm-up and Trial Run
2	Manually moves each axis to check its movement.	Manually Move Machine Tool
3	In general double Z software, switch software configuration.	Switch Software Configuration
4	Return to machine origin.	Return to Machine Origin
5	In general double Z software, select Z-axis.	Select Z-axis
6	Calibrate tool.	Calibrate Tool
7	Set tool No.	Set Tool Number
8	Set workpiece coordinate.	Set Workpiece Coordinate
9	Manage machining technics.	Manage Technics
10	Connect network.	Connect Network
11	Load program file.	Manage Program Files
12	Set related parameters necessary for machining.	Set Parameters
13	To have a general idea about the program and tool path, execute simulation.	Simulate
14	Run program.	Run Program

Step	Operation	Reference
15(Optional)	During machining, check current program information, adjust motion track, return to fixed point, workpiece origin and etc.	Adjust Track View Return to Workpiece Origin Return to Fixed Point

Execute Warm-up and Trial Run

After power up, it is not recommended to start machining directly without knowing actual working conditions. In this case, you can use Warm-up and Trial-run function to test the machine, including working conditions like spindle rotation, etc.

Use Warm-up and Trial-run function in following situations:

- After restarting the software
- After restarting the machine
- After all axes have been returned to the machine origin

To start warm-up and trial-run:

1. Press corresponding buttons on operational panel to turn to Jog, Handwheel or Stepping mode.

   Note that Warm-up and Trial-run is not available in REF mode.
2. Press **State** > **A** > **F5** to open **Warm-up and Trial-run** interface.
3. Set parameters.
4. Press **F1** to start warm-up and trial-run.

   Any stopping actions (including manual stop, ESTOP, stopping caused by switch limit, etc.) will stop warm-up and trial-run process.

Manually Move Machine Tool

Through related parameter settings and axis direction keys, you can control the movement of machine tool.

To manually move machine tool:

1. Set running speed and step length.
   a) Press corresponding mode button on operational panel to turn to Jog/Handwheel/Stepping mode.
   b) Press **State** > **A**.

      Coor-Manual interface shows.
   c) To set parameters Manual High Speed, Manual Low Speed, Custom Step of XY and Custom Step of Z, press direction key ↑/↓/←/→ > **Enter**.

      You can also set parameters Manual High Speed and Manual Low Speed in the **Machine Param** interface.
2. Move the machine tool:
   a) Select feed mode.

Under Jog mode, press axis direction key, machine tool will move continuously until you releasing the key.

Under Handwheel mode, you need to connects the machine to handwheel. Machine tool moves through rotating the hanwheel.

Under Stepping mode, you can select fixed step length through pressing **X1**/ **X10**/ **X100** or customize the stepping length through pressing **XUD**. Every time you press axis direction key, the axis moves selected stepping length.

b) To manually move machine tool at low speed, directly press axis direction key; to manually move it at high speed, press **F1** in **Coor-Manual** interface or press number key **5**.

Switch Software Configuration

Switching the configuration according to active axes of the machine tool. Note that generally the configuration is well set up by the manufacturer before leaving factory.

General Double Z Software provides two types of configuration: Linkage configuration and Alternative configuration. The function is absent in General Three-Axis Software.

To switch the software configuration:

1. Press **System** > **C** to open **System Info** interface.
2. Press **F6** to open a dialog box named “Switch Configuration”.
3. Select the configuration and press **Enter**.
   Target configuration takes effect immediately.

Return to Machine Origin

After system starts, generally returning machine origin is required. Only after such a operation, soft limit can be enabled and fix point can be set.

According to the type of encoder, and whether to enable encoder feedback, returning to machine origin can be divided into the following types:

- Return to machine origin incrementally.
- Set absolute datum.

Return to Machine Origin in Incremental Mode

This is suitable for incremental encoder.

Before returning to machine origin, please ensure the following:

- Parameter 11001 Encoder Type is set to 0: incremental encoder.
- Check nameplate of motor to ensure the setting is incremental encoder.

To return to machine origin:

1. Press **BACKREF** to activate reference point mode.
2. Press **Status** > **A** to enter into **Coor-Reference** interface.
3. Do one of the following

Key	Explanation
F1	X-axis returns to machine origin

Key	Explanation
F2	Y-axis returns to machine origin
F2	Z-axis returns to machine origin
F7	All axes return to machine origin

For safety, when returning to machine origin, it is suggested to firstly return Z-axis, then X-axis and Y-axis.

If you press **F7**, the system will return Z-axis first and then return X-axis and Y-axis at the same time by default.

Set Absolute Datum

This is suitable for absolute encoder.

Before setting datum, please ensure the following:

- Parameter 11001 Encoder Type is set to 1: absolute encoder.
- Parameter 11001 Encoder Feedback is set to Yes: absolute encoder.
- Check the nameplate of motor to ensure the setting is absolute encoder.

To set absolute datum:

1. Press **REF.Point** to activate Reference Point mode.
2. Press **State** > **A** to enter into **Coor-Reference** interface.
3. Press **F5** and input developer password to enter into **Datum Setting** interface.
4. Do one of the following:

Key	Explanation
F1; **F2**; **F3**	This is for setting datum for each axis. Take X-axis as an example: **1.** Switch to manual mode. **2.** Move X-axis to a fixed position. **3.** Return to Reference Point mode. **4.** Press **F1**. **5.** To make the setting effective, restart software.
F5	To import datum data: **1.** After installing new software, press **F1** to import datum data you saved before from USB flash disk for direct use. **2.** To make the setting effective, restart software.
F6	After updating software, the set datum information will lose. To avoid set machine origin again, after setting datum, press **F6** to USB flash disk for later use.

Select Z-axis

This operation is suitable for double Z software, and it is used for selecting Z1-axis and Z2-axis.

To select Z-axis:

1. Enter into **Select Z-axis** interface:
   - Under Auto mode, press **State** > **A** > **F6**.
   - Under Jog/Handwheel/Stepping mode, press **State** > **A** > **F4**.
2. Select Z-axis:
   - Press **F1** to select Z1-axis.
   - Press **F2** to select Z2-axis.
   - Press **F3** to select both Z1-axis and Z2-axis.

   Operations of returning to fixed point, returning to workpiece origin and calibrating tool are only available for the corresponding axis you choose.

   Under alternative configuration, key **F3** does not exist.

Calibrate Tool

Through calibrating tool, you can set tool offset and then define the specific location for workpiece coordinate system.

NK280B offers two methods of calibrating tool.

Execute Fixed Calibration

During calibrating tool, the system will record the machine coordinate when tool nose touches the surface of tool sensor. (Tool offset at this time is equal to the value of machine coordinate.)

When you need to calibrate several tools, switch tool through T command and then proceed the following steps.

To execute fixed calibration:

1. Select a tool for tool calibration.
2. Do one of the following:
   - Under Auto mode, press **State** > **A** > **F5**.
   - Under Jog/Handwheel/Stepping mode, press **State** > **A** > **F3**.
3. Calibrate tool:

Condition	Step
With a tool sensor	• In General Double Z Software, press **F1** /**F2**/**F3** to select corresponding axis. • In General Three-Axis Software, press **F1**.
Without a tool sensor	• In General Double Z Software, press **F4** /**F5**/ to select corresponding axis and manually set tool offset for Z1Z2-axis. • In General Three-Axis Software, press **F4**.

   The system records tool offset automatically into **Tool Table** interface.
4. Move any tool to the surface of workpiece.
5. Execute **Clear**.

Execute First and Exchange Calibration

In essence, this mode is to compensate tool offset to workpiece offset.

In General Double Z Software, you can choose to execute tool calibration for Z1-axis and Z2-axis separately or together.The setting in the operation of selecting Z-axis shall prevail.

In General Three-Axis Software, it directly execute tool calibration for Z-axis.

To execute first and exchange calibration:

1. Manually move Z-axis to the surface of workpiece.
2. To define workpiece origin, press **Clear**.
3. Enter into tool calibration interface:
   - Under Auto mode, press **State** > **A** > **F5**.
   - Under Jog/Handwheel/Stepping mode, press **State** > **A** > **F3**.
4. Execute first calibration:
   - In General Double Z Software, press **F1**/ **F3**/**F5**.
   - In General Three-Axis Software, press **F1**.

   The system automatically records the machine coordinate of Z-axis at this moment and machine tool can machine workpiece.
5. After tool change or tool breakage, execute second calibration:
   - In General Double Z Software, press **F2**/ **F4**/**F6**.
   - In General Three-Axis Software, press **F2**.

   This operation is not available until first calibration is done.

   The system recovers Z-axis workpiece coordinate of current point and machine tool can continue to machine workpiece.

Set Tool Number

This operation is to set tool number for spindle.

In General Three-Axis Software, to select tool No. for spindle, modify parameter Current Tool No.

In General Double Z Software, to specify the tool for Z1 and Z2-axis by setting tool No.:

1. Press **Auto** button on operational panel to activate Auto mode.
2. Press **State** > **A** > **F4** to open **Set Tool No.** dialog box.
3. Choose the Z-axis.
4. Enter the tool No. for the selected Z-axis.
5. Press **Enter** to confirm it.

Set Workpiece Coordinate

Workpiece Coordinate meets the following formula:

Workpiece Coordinate = Machine Coordinate – Workpiece Offset – Public Offset – Tool Offset

Set WCS Origin by Centering

For regular-shaped workpiece, you can use Centering function to get workpiece origins of X-axis and Y-axis.

It includes two types of centering:

- Line Centering: Used to find the center point by two points on the workpiece. It is applicable to regular rectangular workpiece.
- Circle Centering: Used to find the center point by three points on the workpiece. It is applicable to circular workpiece.

For details, see NcStudio Function Manual.

To start centering (taking X-axis and line centering as an example):

1. Press corresponding mode button on operational panel to activate manual mode, including Jog mode, Handwheel mode and Stepping mode.

   Handwheel mode is suggested.
2. Press **Advanced** > **B** > **T** to open **Center** interface.
3. (Optional) Press **F5** to turn on the edge finder.

   In centering process, use the edge finder to position precisely. At this time, value of parameter Spindle Speed in Centering will be used.

   If the edge finder is not enabled, parameter Spindle Speed in Centering is inactive. Press **CW** or **CCW** on operational panel to turn on the spindle. Spindle rotational speed will be the setting value for the spindle in the software or the rotational speed specified by S command in the program file.
4. Manually move the tool to one side of the workpiece, and press **F1**.
   The software records the X and Y machine coordinates of the current position.
5. Manually move the tool to the other side of the workpiece, and press **F2**.

   When X-axis is centered, the other axes should keep still.

   The software calculates the middle point according to the second position and the last recorded values, and sets it as the workpiece origin in X and Y axes.

Set WCS Origin by Clearing

When demand for accuracy is not strict enough, you can set workpiece origin by clearing.

Use Clear (also called Zeroing) to clear current workpiece coordinate of each axis, and set work offset value as corresponding machine coordinate of each axis. Machine coordinate of each axis remains unchanged.

To clear the workpiece coordinate and set workpiece origin:

1. Press corresponding mode button on operational panel to activate available modes.

   Clear function is only used in Jog, Handwheel and Stepping modes.
2. Press **State** > **A** > **F2**.
   Clear interface opens.
3. Do clearing on each axis.

   - **F1**: Clear current workpiece coordinate of X-axis to zero.
   - **F2**: Clear current workpiece coordinate of Y-axis to zero.

- **F3**: Clear current workpiece coordinates of both X-axis and Y-axis to zero.
- **F4**: Clear current workpiece coordinate of Z1-axis to zero.
- **F5**: Clear current workpiece coordinate of Z2-axis to zero.
- **F6**: Clear current workpiece coordinate of all axes to zero.
- **F7**: Return to previous operational button bar.

The system will automatically set current machine coordinates as corresponding work offsets.

Current workpiece coordinates and machine coordinates of each axes are as shown in following figure.

Do Clear to both X-axis and Y-axis.

Set Workpiece Offset

Workpiece offset is the offset relative to machine origin.

To set workpiece offset:

1. Press **Advanced** > **A** to enter into **Coor-Manage** interface.
2. Select coordinate system through direction key **PgUp**/**PgDn**.
3. Press **F1** to make setting effective.

   The system offers 26 coordinate systems. They are G54~G59, G54P0~G54P19.

   The setting takes effect and shows directly in the interface.
4. Set offset for each axis under the set coordinate system.

   The set value shows directly in **WP OFT** setting area.

Set Public Offset

In **Coor-Manage** interface, press ↑/↓ to enter into **Public OFT(External)** setting area.

To set public offset:

- Manually input value.

  This is suitable for all axes.

  a) Select a setting box.
  b) Press **Enter** and input the offset value in the popup dialog.
- Customize the lift distance or deepen distance for Z-axis.
  a) To set adjustment value for Z-axis, press **F5**.

     The system offers the following adjustment value: 0.01mm, 0.10mm, 0.50mm, 1.00mm, 5.00mm.
  b) Press **F6 Deepen**/ **F7 Lift**.

     The set adjustment value displays in **Public OFT (External)** setting area.

Back up Coordinate

Coordinate backup is to save the current workpiece offset into the system for later use.

To save or recover coordinate:

1. Press **Parameter** > **C** to open **Coor Backup** interface.
2. Save the current workpiece offset into the system for later use.
   a) Press direction key ↑/↓ to choose line No.
   b) Press **F1**.

   The system will save the offset into the selected line No. If you do not select new line No., the existing offset in the line will be overwritten.
3. To avoid set workpiece offset repeatedly, recover the saved offset into the current workpiece coordinate.
   a) Press direction key ↑/↓ to choose the offset you would like to recover.
   b) Press **F2**.

   A dialog containing following information pops up.

   Yes: recover offset coordinate for X-axis, Y-axis and Z-axis at the same time.

   No: recover offset coordinate for X-axis and Y-axis.

Manage Technics

During machining, this operation is helpful to improve machining performance. NK280B motion control system offers several machining technics, you can set it in **Technics Management** interface.

Execute Arraying

For batch machining, you can use arraying function to improve efficiency.

For details, see NcStudio Function Manual.

To enable arraying function:

1. Press **Advanced** > **=** > **F1**.
   It opens **Array** interface.
2. Set parameters.

   For details about parameters, see NcStudio Function Manual.
3. Press **Start** button on operational panel.
   The system executes arraying automatically according to parameter settings.

Compensate Workpiece

You can compensate workpiece to correct the workpiece dimension offset due to tool wear or other causes.

Before stetting parameters for workpiece compensation, set parameters for arraying first.

To compensate workpiece:

1. Press **Advanced** > **=** > **F2**.
   Workpiece Compensation interface opens.
2. Set compensation parameters.

   If the desired workpiece dimension is: X=120mm, Y=80mm

   The the actual piecework dimension is: X=120.001mm, Y=79.999mm

Make workpiece compensation as follows:

- **Dimension(J)**: 120.000
- **Dimension(K)**: 80.000
- **Direction**, **Order**, **Comp Mode**: custom

You can get workpiece whose dimension is 120mm*80mm.

Compensate Screw Error

It includes screw pitch error compensation and backlash compensation.

To compensate screw error:

1. Press **Advanced** > **=** > **F7**.

   Screw Error Compensation interface shows.

2. Do one of the following:

Operation	Step
Manually set compensation data.	1. Press shortcut key **T**/**S**/**P** to select an axis. 2. Do one of the following: • Press **F1** to insert number for compensation data and set related data by direction key ↑/↓/←/→/ > **Enter**. • Press **F3** and press **T**/**S**/**P** to enter setting values. The system automatically generates a group of compensation data into the compensation table. 3. Press **X**/**Y**/**Z** to set backlash for each axis. 4. Press **F6** to make setting effective.
Import compensation data.	1. Press **F4**. 2. Do one of the following: • Press **F1** to use imported compensation data directly. • Press **F2** to add the new compensation value to the old value. 3. Press **F6** to make setting effective.
Export compensation data.	Press **F5** to export current compensation data to USB flash disk.

Except no need to set backlash for bidirectional compensation, other settings for bidirectional compensation is the same with settings for unidirectional compensation.

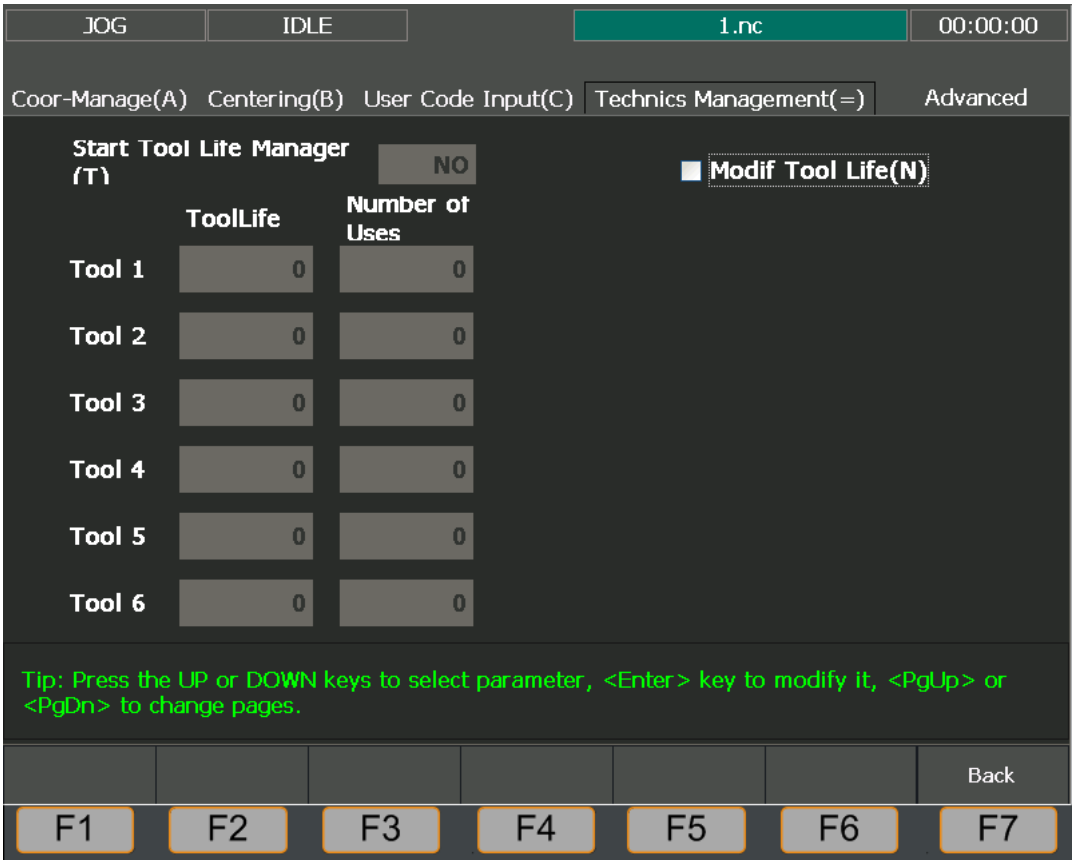
Manage Tool Life

Note that the interface is different in General Three-axis Software and Double Z Software.

Tool life decides how many times a tool can be used for machining task.

To set life of a tool:

1. Press **Advanced** > **=** > **F3** to open **Technics Management** interface.



2. Press **T** to select "Yes" to enable tool life management function.
3. Press **N** to check in the checkbox in front of "Modify Tool Life(N)".
4. Set used times for each tool.

Compensate Tool

If the tool nose radius is altered due to tool wear, tool sharpening or tool change, you can input the new tool parameter values in the **Tool Compensation** interface, saving trouble of modifying the programmed machining file.

To compensate tool:

1. Press **Advanced** > **=** > **F6**.

   Tool compensation interface shows.

2. To set corresponding values for tool compensation, press direction key ↑/↓/←/→ > **Enter**.

Connect Network

To realize mutual access between NK280B and the computer, you need to set network connection first.

Make sure the computer and NK280B are within the same LAN.

To set network connection between the computer and NK280B:

1. Set IP address.
   - Automatically set IP address.
   - Manually set IP address.
2. Modify device information.

3. Verify connection.

Manually Set IP Address

If DHCP function is not supported, you need to manually set IP address of both NK280B and the computer.

When the computer and NK280B is directly connected by net cable, or one computer is connected with several NK280B via router. DHCP function is not supported.

Manually set IP address of the computer and NK280B:

1. Set IP address of the computer.
2. Turn to NK280B. Press **System** > **C** > **F7** > **F5**.
   IP Setup dialog box opens.
3. Press **S**, and enter address information in the dialog box.
   You need to set up both the IP address and submask.
4. After setup, you can turn to **Net Info** interface to check the updated net information.

   An example of setup of IP address and submask of both the computer and NK280B:

   IP address: 192.168.1.188 (IP address of the first group of NK280B should be the same as those of the computer).

   Subnet mask: 255.255.255.0 (Subnet mask of NK280B should be the same as that of the computer).

   Default gateway: 192.168.1.1 (Gateway of NK280B should be the same as that of the computer).

Automatically Set IP Address

If the devices support DHCP function, the system can obtain IP address automatically.

Make sure all devices support DHCP function and all devices have been well connected.

To automatically set IP address of the computer and NK280B:

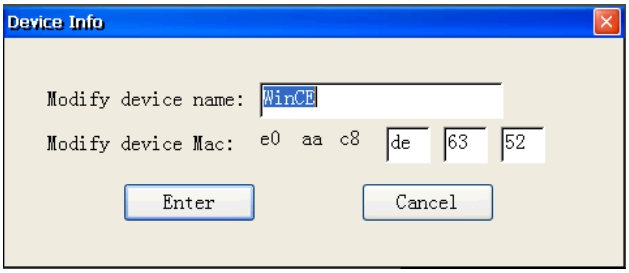
1. Obtain IP address of the computer.
2. Obtain IP address of NK280B.
   a) Press **System** > **C** > **F7** > **F5** to open **IP Setting** dialog box.
   b) Press **L** to select automatic way, and press **Enter**.

Modify Device Information

If there are more than one NK280B, IP address and MAC address of each of them should not be the same. If the same, manually reset IP address and MAC address.

To manually set IP address and MAC address of NK280B:

1. Press **System** > > **F6** to open **Device Info** dialog box.



2. Modify the name of device and MAC address.

   Modify the device name to WinCE01, WinCE02, WinCE03...

   MAC address can be any hexadecimal data.

Verify Network Connection

After IP address setup of both NK280B and the computer, use ping command to verify the connection.

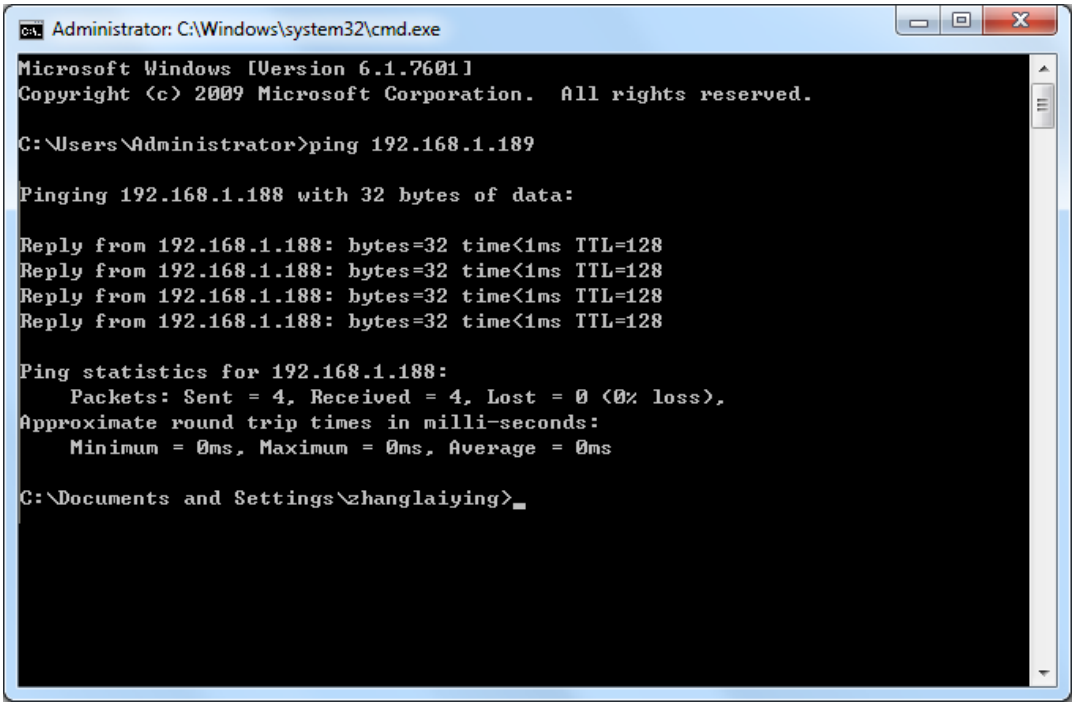
To verify the connection between the host and the computer:

1. Click on the computer: **Start** > **Run**, enter "cmd" and click **Enter**.
2. On the pop-out console, type in ping command and click **Enter** to check the result.

   ping command: ping IP address

   Enter ping command: ping 192.168.1.189

   Successful ping result goes like:



If ping failed, check connection of devices and their IP setups.

Manage Program Files

NK280B can access program files of four types:

- Local file: program files stored in NK280B host.

- USB file: program files stored in a USB flash disk.
- Net file: program files stored in computer or other NK280B host.
- Wizard file: program files generated by NK280B program wizard.

NK280B supports four types of file formats:

- .nc
- .plt
- .eng
- .dxf

Access Local Files

You can access program files that are stored in `D:\` directory of NK280B.

To access the local files and edit the files:

1. Press **Program** > **A** to open **Local** interface.
2. Operate on program files.
   - **F1**: Load the selected program file into the system for machining.
   - **F2**: Unload the current program file from the system.
   - **F3**: Delete the selected program file.
   - **F4**: Edit the selected program file. You can insert block, modify code or find and replace characters.
   - **F5**: Rename the selected program file.
   - **F6**: Copy the selected program file to USB flash disk.
   - **F7**: Create a new program file.

   Only program file that is formatted with ".nc" is editable.

Access USB Program Files

You can access and edit program files that are stored in USB flash disk.

To access and edit program files from USB flash disk:

1. Press **Program** > **B** to open **USB** interface.
2. Operate on program files.

   Operations are the same with that of editing local program files. See Access Local Files.

Access NK280B Program Files from Computer

You can access and manage program files in control system hosts from the computer.

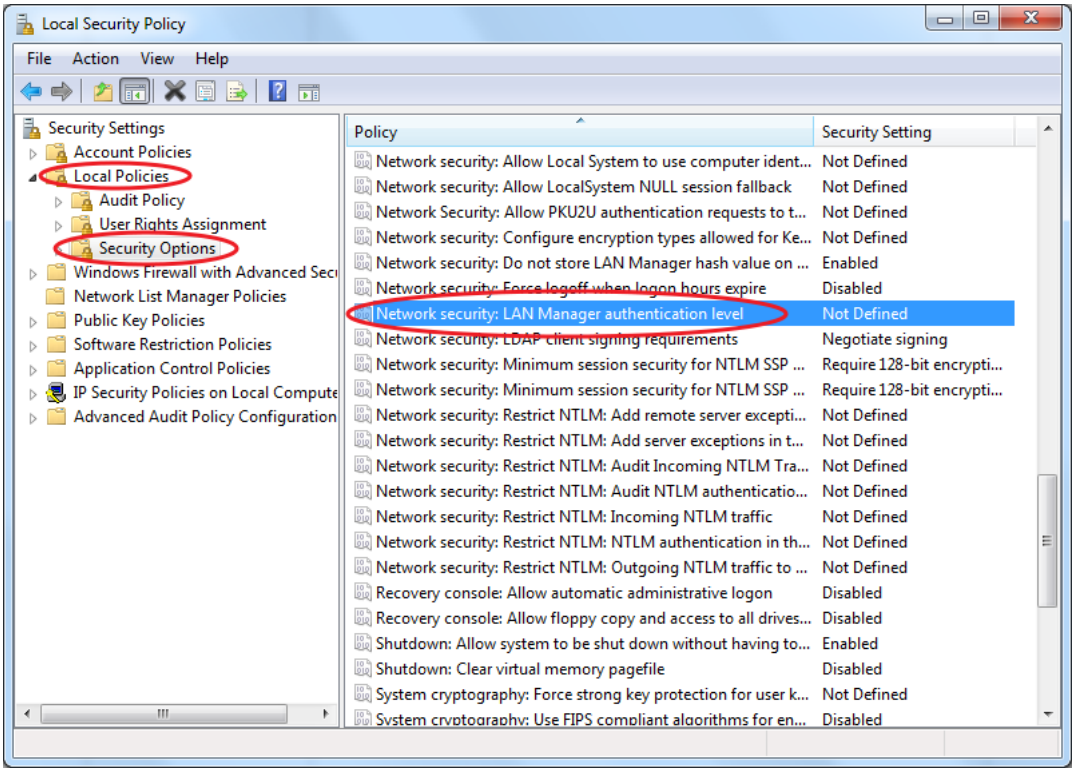
Two ways to access network files:

- Sharedoc
- FTP

Sharedoc

1. Open **Sharedocs** interface.
   - Windows XP operating system: Click **Start** > **Run**, enter IP address (e.g. “\\192.168.1.188”).

- Win7 operating system: Click **Start** > **Control Panel** > **Administrative Tools** > **Local Security Policy**, set attribute of network safety, and restart the computer.



2. Double click on file folder "Sharedocs".
3. Upload files to nk280b download files from NK280B to the computer, or edit and delete the files.

FTP: Access share docs in the host via the server. Make sure FTP server has been established in the computer.

4. Enter address (eg. "ftp://192.168.1.188") in the explorer to open net files of NK280B.

Access Network Files

You can access net files on **Local** interface.

Make sure connection between control system hosts and the computer has been well done. For details, see Connect Network.

To access net files:

1. Press **Program** > **A** to open **Local** interface.
2. Press **Shift**+**Bkspc** to refresh the screen.

   Net files are tagged "Net".

Run Program Wizard

It helps users to generate program files by setting some parameters.

There are five types of wizards:

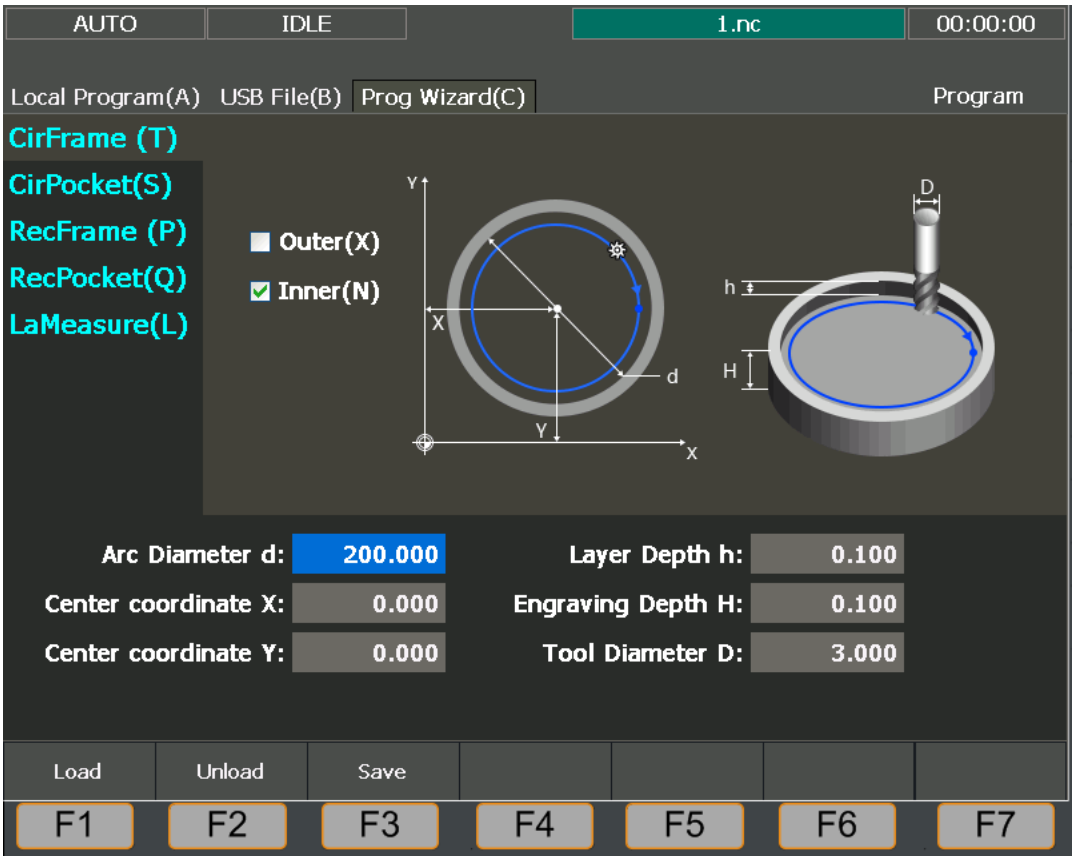
- CirFrame(T)
- CirPocket(S)
- RecFrame(P)
- RecPocket(Q)

- LaMeasure(L)

For details, see NcStudio Function Manual.

To run program wizard:

1. Press **Program** > **C** to open **Prog Wizard** interface.



2. Use shortcuts to choose wizard type as shown on the left.
3. Set parameters.
4. Press **F3** to generate and save the program file.
5. Press **F1** to load the program file into the system.

   The wizard program file can be used for machining directly.

   For LaMeasure(L) wizard, parameter settings should conform to following rules.

   - Start position and end position should be within travel range.
   - One repeat refers to the process of "Start position-->End position-->Start position". Every repeat records one group of data. When generating screw error compensation file, average value is used.
   - Measuring points should be an integer.

Set Parameters

This operation includes checking, modifying and backing up parameters.

Modify Parameters

Check parameter settings for current working conditions of the system or modify parameter settings for enabling and disabling certain functions.

To check current parameter settings or modify parameters:

1. Press **Parameter** > **A** to open **Machine Parameter** interface.
2. (Optional) Choose your role.
   - **F1**: You are an operator or end-user of the machine tool.
   - **F2**: You are a manufacturer and need to access manufacturer's parameters.
   - **F3**: You are a control system developer and need to access developer's parameters.

   According to user roles and privileges, all parameters are divided into three classifications. To access manufacturer's parameters and developer's parameters, passwords are required.
3. Press arrow keys to select the parameter, and press **Enter** to open user input box.
4. Change the parameter value and press **OK**.

   According to time to take effect, all parameters are divided into three classifications.

Effective Time	Description
Immediately	Modification to the parameter takes effect immediately.
After Restart	Modification to the parameter takes effect after restarting the system.
After Reload	Modification to the parameter takes effect after reloading the program file.

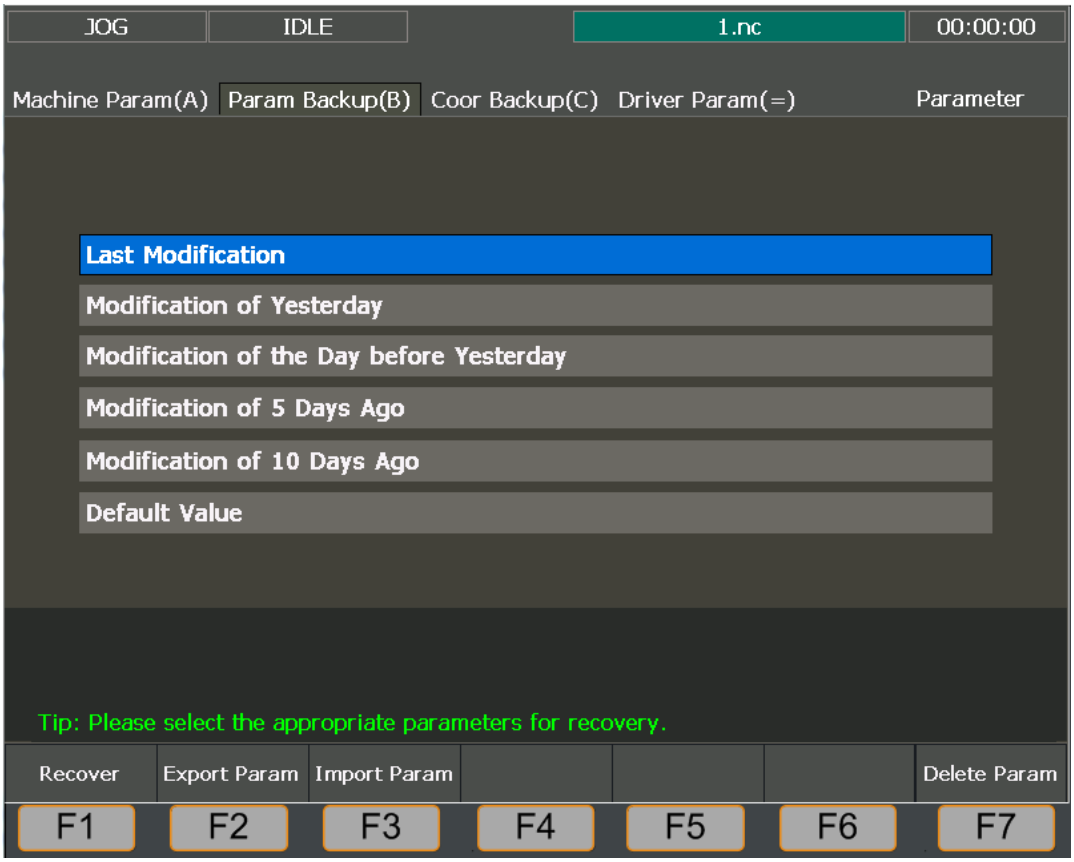
5. (Optional) Press **F7** to modify privilege passwords.

Back Up Parameters

Use parameter backup function to save parameter settings and restore the settings in the future or for other CNC device.

You can save parameter settings, restore the settings, export the settings or import another parameter settings.

1. Press **Parameter** > **B** to open **Param Backup** interface.



2. Operate on parameter settings.

   - **F1**: Restore the selected parameter settings.
   - **F2**: Export the selected parameter settings to USB flash disk. The file is formatted as by default.
   - **F3**: Import an external parameter settings file () into the current system. You need to restart the system to validate the parameter settings.
   - **F7**: Delete the selected parameter settings.

Simulate

Simulation provides a fast but lifelike simulated processing environment for users.

Make sure you have load the program file into the system before simulation.

Running in the mode of simulation, 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 **Motion Trace** interface. By simulating, users see moving form of the machine tool in advance, avoiding machine tool damage due to programming mistakes in processing procedure.

To execute simulation of a program:

1. Start simulation.

   - Press **State** > **B** > **F1** to start simulation.
   - Press **Simulate** button on operational panel to start simulation.

2. Press **F7** on **Motion Trace** interface to stop simulation.

Run Program

There are four ways to run program.

Execute Auto Run

To start auto run:

In movement control button area of NK280B operational panel, press **Start**.

The system automatically execute the program until end or being interrupted by man.

Execute Single Block

Use single block function for better error diagnosis and troubleshooting.

Once Single Block function is enabled, the program running will stop the moment axial feedrate is 0.

To enable Single Block function:

1. Press **Auto** button on operational panel to activate Auto mode.
2. Press **Single Block** button on operational panel to enable the function.
   The indicator light turns on.
3. Press **Start** button on operational panel to start program running.
   After one block of the program has been executed, the system stops and enters into Pause state.
4. Press **Start** button to continue program running till whole program has been executed.

Use Handwheel Guide

Handwheel guide is a way of operation that the automatic execution speed is manually controlled during auto processing to guard against such problem as “tool damage” and dangers caused by wrongly loaded program or inappropriate tool path.

With handwheel guide function, the system will execute the processing program once the handwheel is turned in clockwise direction and stop processing within 300ms after the handwheel turning stops. Processing speed changes according to the handwheel turning speed.

To enable Handwheel Guide function:

1. Press **Auto** button on the operational panel to activate Auto mode.
2. Enable Handwheel Guide function by following two ways:
   - Press **HW Guide** button on the operational panel.
   - Press **State** > **A** > **F1**.

   Indicator light on the upper left of the button turns on or F1 button is highlighted.

Enable Selective Machining

Selective machining is a form of optional skip.

Use selective machining to select desired program blocks for execution.

To enable selective machining function:

1. Press **Auto** button on operational panel to activate Auto mode.
2. Open **Start Line** dialog box.

- Press **State** > **C** > **F1**.
- Press **State** > **A** > **F2**.

Start Line dialog box pops up.

3. Set parameters for selective machining, including "Start" position and "End" position.

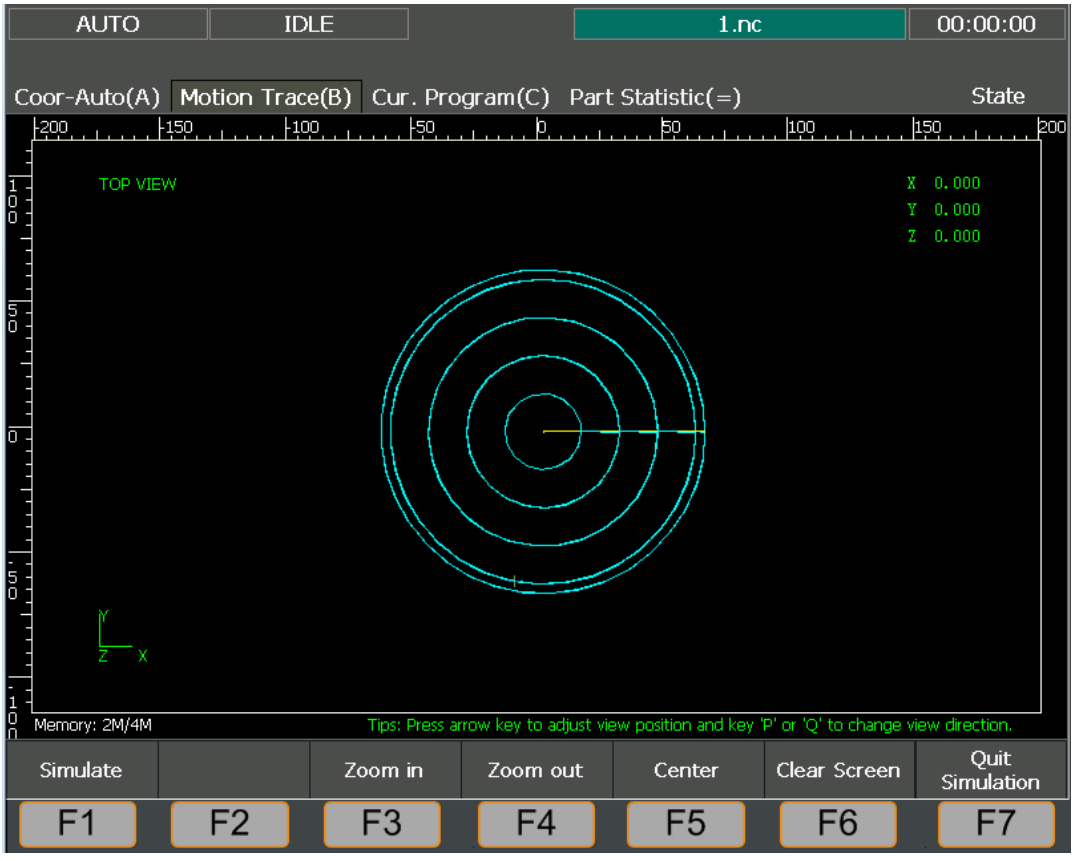
   Shortcut keys: **T** for "File header", **S** for block No. of start position, **P** for "File End" and **Q** for block No. of end position.

Adjust Track View

Track gives a three-dimensional display of the tool path in real time. Users can view the tool path more institutionally.

To adjust track of the tool path:

1. Press **State** > **B** to open **Motion Trace** interface.



2. Scale track on the interface.

   - Zoom Out: Press **F3** to enlarge the track to check some details.
   - Zoom In: Press **F4** to shrink the track.

3. Adjust position of track on the interface.

   - Center: Press **F5** to move the track to the center.
   - Move: Press arrow keys (← → ↑ ↓) to move the track to the left, right, up and down.

4. Press **P** or **Q** to change observation view of track.

   Press **P** to switch to next view and press **Q** to switch to previous view.

Return to Workpiece Origin

At any time during machining, you can return to the workpiece origin(also called WCS Zero).

When machine tool returns to the workpiece origin, Z-axis returns to the safety height first, then X-axis and Y-axis return to the origin. The safety height for Z-axis is to aviod potential crash onto the workpiece surface during returning.

To return to the workpiece origin:

Press **State** > **A** > **F7**.
X, Y and Z-axis return to the workpiece origin automatically in order of Z-XY.

Return to Fixed Point

The fixed point is a position on the machine, which is specified by parameters.

Returning to the fixed point facilitates following operations.

- Change the workpiece.
- Change the tool.

For details, see NcStudio Function Manual.

To return to the fixed point:

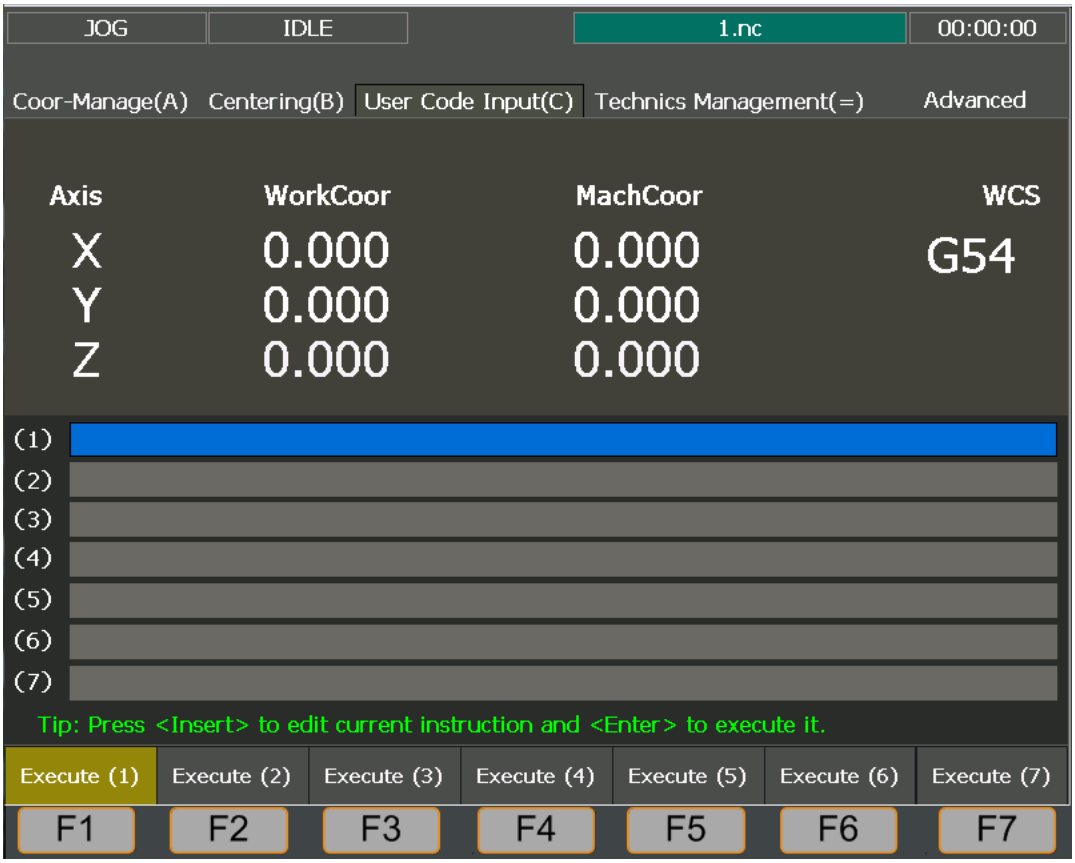
1. Activate Jog, Handwheel or Stepping mode.
2. Press **State** > **A** > **F6**.
   Three axes return to the fixed point.

Execute MDI

Manual Data Input (MDI) is a way to directly execute user input NC code.

To enter NC code and execute it:

1. Press **Advanced** > **C** to turn to **User Code Input** interface.



2. Use arrow keys to select line, and press **Insert** button to open edit box.
3. Enter your NC code in edit box.
4. Press **Auto** button on operational panel to activate Auto mode.
5. Press **F1**~**F7** or **Enter** to execute corresponding code.

   Execution of user-defined NC code must be in Auto mode.

Check Statistics Information

Users can check machining statistics information of both current program and history programs, including completed workpiece number, completed length, running time, etc.

To check the machining statistics information:

1. Press **State** > **=**.

   It turns to **Part Statistic** interface.
2. (Optional) Press **F1** to clear all statistics in the program list.
3. (Optional) Press **F2** to export all statistics in the program list to external storage device (e.g. a USB flash disk). The statistics will be saved as a .txt document.

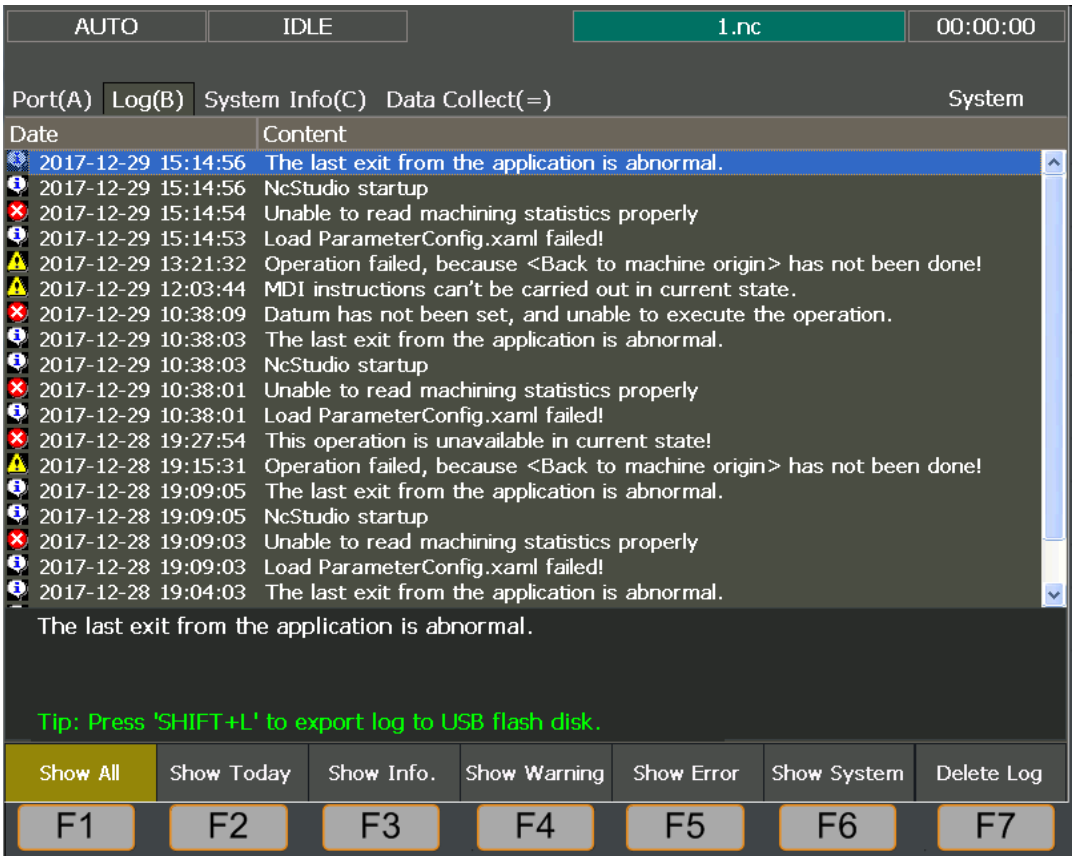
Check Logs

History operations, warnings, errors can be logged.

To open **Log** interface:

1. Press **System** > **B** to turn to **Log** interface.
   - Show All: Show all logs.
   - Show Today: Show today's logs only.

- Show Info: Show normal information only.
- Show Warning: Show warnings only.
- Show Error: Show errors only.
- Show System: Show logs related with the system only.
- Delete: Delete all logs on the interface.



2. (Optional) Press **Shift+L** to export all logs to USB flash disk.

System Maintenance

This part is designed to help users to register the software and the driver, update the software or FPGA, BOOT.

Register Software and Driver

Register usage time for the software and the driver.

Time registration limits usage time of the software of the driver. At present, time registration function can only be available to WISE driver. Usage time of the WISE driver is registered together with the software.

You can turn to **System Info** interface to check remaining usage time.

To register the system and the driver:

1. Send the "Device No." on the **System Info** interface to the supplier or developer to get back a registration code.
2. Press **System** > **C** > **F1** to open the dialog box named "Register".
3. Enter the registration code.

Update FPGA

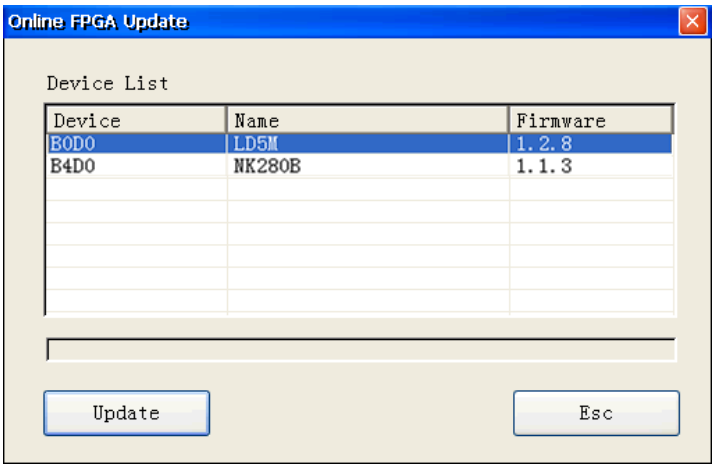
It is designed to help users to update the FPGA programs.

Make sure all hardware components have been well connected.

You can update the FPGA programs of NK280B host, Lambda controller and extended terminal board.

To update the FPGA program:

1. Press **System** > **C** > **F5** to open **Online FPGA Update** dialog box.



2. Press arrow keys to select the device, and press **Enter** to start updating.

Delete Temporary Files

Too much temporary files slow down system running.

Deleting temporary files from time to time helps to release memory of NK280B host, and improve the performance of the system.

To delete temporary files:

Press **System** > **F4** > **F1**.

Update Software to Latest Version

When current version of the software breaks down or there is a new version, you need to update the software.

Prepare a flash disk with the latest version of the software (formatted.weihong) in.

To update the software:

1. Insert the prepared flash disk into USB slot of NK280B host.
2. Access BOOT screen.
   - Press **System** > **C** > **F2**, pass through the authentication and restart NK280B.
   - At the same time of power up, press **G** for several times.
3. Press **F6**, and press arrow keys to select the target software in the pop-up list.
4. Press **Enter** to start updating process.

New version of the software will be opened.

Update Mirror Image

If the operating system of NK280B cannot be loaded, update the mirror image.

Prepare a USB flash disk with the mirror image file named “NK280B_NK_Rx.x.x.nb0”.

To update the mirror image:

1. Insert the USB flash disk with mirror image file into NK280B host.
2. Power on NK280B and simultaneously press **M** key several times .

   It turns to an interface with several system updating options.
3. Press **T** key to choose “Update System” command.

   The system loads the mirror image file from USB flash disk.
4. After successfully loading the mirror image, press **Y** key to confirm and go on.

After updating the mirror image, you need to install the CNC software, otherwise, it is only an empty system.

Install Software

It helps users to install the software after mirror image has been upgraded.

Prepare a flash disk with software (formatted in.weihong) in.

To install the software for the first time:

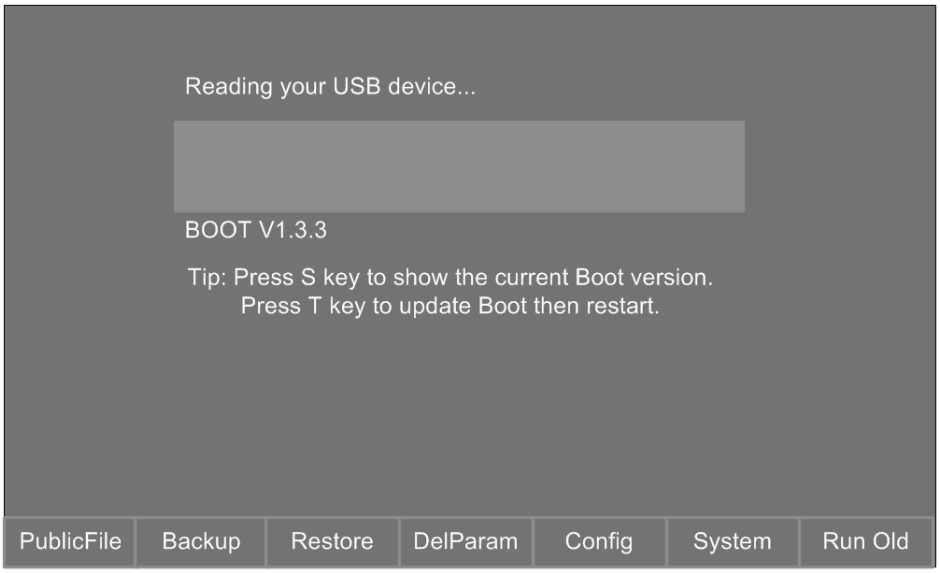
1. Insert the flash disk into USB slot of NK280B host.
2. After mirror image updating has been finished, press **Z** to enter **BOOT** interface.
3. Press **F6** to open the list, and use arrow keys to select the target software.
4. Press **Enter** to start installation.

After installation finishes, new software will be opened.

Update BOOT Interface

BOOT update interface is involved in both process of mirror image updating and software updating.

See figure below for the BOOT interface.



Public Files	Use PUBLIC file in the USB flash disk. PUBLIC files are stored in `CHN\files` and `ENG\files`.

Backup	Export the software which has been backed up to `\Backup` file folder in root directory of USB flash disk.
Restore	Restore the system with software in `\Backup` file folder.
Delete Parameters	When updating the software to the latest version, use the button to delete parameter settings of the last version. Skip it if you want to reserve previous parameter settings.
Configuration	Use new system configuration file, i.e. Config file.
System	It refers to updating the software in nature, upgrading the software to the latest one or installing a new one. Only software formatted in.weihong can be identified.
Run Old	Start the previous operating system.

NK280B Parameter Reference

The Parameter Reference is designed to help users (mainly manufacturers) to locate information easily and quickly. It includes frequently-used parameters for main functions and operations.

Note that only parameters related to main and frequently-used functions are listed in a brief way. For complete parameter information, please refer to Parameter Reference Manual.

Parameter for Axis Direction

Parameters that decide moving direction of each axis.

Table 3: Parameters Related with Axis Direction

Parameter Name	Descritpion
Axis Direction	It sets moving direction of each axis. 1: Positive direction; -1: Negative direction.

Driver Parameters for Auto Adjustment

Driver parameters that are related to auto adjustment. In terms of driver brands, setting may differs.

Table 4: YASKAWA Driver Parameters Related with Auto Adjustment

Parameter Name	Description
Pn000 Basic Function Selections 0	It is for selecting rotation direction. • 0000: take CCW as positive direction. • 0001: take CW as positive direction.
Pn20E Electronic Gear Ratio (Numerator)	It is for ensuring control system parameters Electronic Gear Ratio (Numerator) and Electronic Gear Ratio (Denominator) are consistent with the same driver parameters.
Pn210 Electronic Gear Ratio (Denominator)	

Parameter Name	Description
Pn212 Number of Encoder Output Pulses	-

Table 5: Wise Driver Parameters Related with Auto Adjustment

Parameter Name	Description
Pr000 Rotational Direction Setup	It is for setting rotation direction for motor. • 0000: for forward commands, the motor rotates towards CW. • 0001: for forward commands, the motor rotates towards CCW.
Pr008 Command Pulse Counts per One Motor revolution	Its value is equal to divide screw-pitch by pulse equivalent.
Pr009 1st Numerator of Electronic Gear	It is for ensuring control system parameters Electronic Gear Ratio (Numerator) and Electronic Gear Ratio (Denominator) are consistent with the same driver parameters.
Pr010 1st Denominator of Electronic Gear	
Pr011 Output Pulse Counts per One Motor Revolution	Its value is equal to divide PG frequency dividing ratio(X4) by 4.
Pr015 Absolute Encoder Setup	It is for ensuring control system parameter 11001 Encoder Type is consistent with the driver parameter. • 0: used as absolute encoder. • 1: used as incremental encoder. • 2: used as absolute encoder and ignoring count overflow of multiple rotations. • 3: set by manufacturer. • 4: set by manufacturer.

Parameters for Encoder

If you enable the encoder function, you need to set the parameters.

Use the encoder feedback function to detect or feed back linear movement distance of screw or angular displacement of the servo motor.

Process of returning to the machine origin and tool calibration are diffferent acccording to the usage of encoder feedback function.

You need to check nameplate of the encoder used for the servo motor for type information or digit information.

Table 6: Parameters Related with Encoder Function

Parameter Name	Description
Encoder Feedback	Whether to use the encoder feedback function or not.
Encoder Type	0: Incremental encoder; 1: Absolute encoder.
Encoder Digit	The digit number of the encoder.

Parameters for Handwheel Operation

It includes parameters that will influence the usage of handwheel.

Table 7: Parameters Related with Handwheel Operation

Parameter Name	Description
Precise Pulse Counting	Yes: Machine moves the exact pulses generated by handwheel; No: Machine moves only when the handwheel is turning.
Handwheel Direction	Relations between the directions of handwheel turning and axes motion. 1: Same direction; -1: Opposite direction.
Handwheel Acceleration	Smaller value gets smoother handwheel movements.
Handwheel Connection	0: Connect the handwheel with Lambda controller; 1: Connect the handwheel with NK280B host.

Parameters for Workbench Range

Workbench Range refers to valid movement ranges or the travelling range in X, Y and Z direction. It serves as soft limits to protect the machine.

Setting values for the upper limits and the lower limits are decided by specification of the machine. When you set the range for the first time, please carefully observe actual moving distance of the axes.

Table 8: Parameters Related with Worktable Stroke

Parameter Name	Description
Negative Travel Limit (MCS)	The positive travel limit in each axis.
Positive Travel Limit (MCS)	The negative travel limit in each axis.
Enable Workbench Range Check	Whether to check the workbench range.

Parameters for Prompt

Parameters that decides prompt method for certain functions or actions.

Table 9: Parameters Related with Prompt

Parameter Name	Description
Enable Workpiece Compensation	If the parameter is set to Yes , the system will ask you to confirm whether to continue machining with workpiece compensation enabled when you start machining.
Prompt for Public Offset Saving	Whether to give prompt when clearing coordinate to zero. If it sets to "True", prompt the user to save the public offset or not; else directly save the public offset without prompt.

Parameters for QEC

Parameters that will influence the use of QEC (Quadrant Error Compensation) function.

When machining an arc, use QEC function to make compensation for the acute angle to get smooth result.

For details about QEC function, see NcStudio Function Manual.

Table 10: Parameters Related with QEC

Parameter Name	Description
Enable Quadrant Compensation	Whether to enable cross quadrant compensation function for the acute angle.
Quadrant Compensation Time	The larger value is, the larger the compensation-affected area will be. Recommended value: 0.02s.
Quadrant Compensation Value	The larger its value is, the more obvious the compensation result will be. However, note that too large value will make the arc concaved, while too small value will not suppress the spike effectively. It is suggested to measure the actual height of the spike with a laser interferometer or other measuring device in debugging, and then set this parameter as 0.3 to 3 times of spike height. Compensation result is also related with compensation time and intensity.

Parameter Name	Description
Quadrant Compensation Delay	The spikes may not appear exactly at the four quadrant positions due to mechanical properties of machine tool, but a little distance away from the quadrant points. Estimate the time to travel this distance and then set the time as the value of this parameter.
Quadrant Compensation Intensity	The larger the value is, the more obvious the compensation result will be.

Parameters for Tool Compensation

Parameters that will influence the tool compensation function.

For details about the tool compensation function, see NcStudio Function Manual.

Table 11: Parameters Related with Tool Compensation

Parameter Name	Description
Enable Cutter Compensation	Yes. Enable G41,G42 cutter radius compensation and G43,G44 cutter length compensation.
Cutter Compensation Option	Specify the Cutter Compensation Type. 1: Normal type; 2: Intersect type; 3: Insert type.
Cutter Compensation Interfere Type	0: None; 1: Three; 2: More
Cutter Compensation Lookahead Segno	Specify the segment number when tool compensation interfere avoidance is enabled, with range [3~30].

Parameters for Tool Calibration

Parameters that will influence tool calibration function.

Table 12: Parameters Related with Tool Calibration

Parameter Name	Description
Cali Mode	Tool calibration mode. 1: Auto measurement mode (namely the fixed tool calibration); 2: First and exchange calibration mode.
Fine Tool Cali Times	Fine calibrating times when the tool approaches to the surface of tool sensor.
Rough Speed of Cali.	The speed when the tool approaches the surface of tool sensor for the first time.

Parameter Name	Description
Max Cali Tolerance	During several times of tool calibration, the maximum tolerance of calibration results.
Tool Sensor Thickness	The distance from Tool Sensor surface to WCS Z0. It can be used to automatically calculate WCS Z0 when cutter touches the Tool Sensor.
Tool Sensor Fixed Position (MCS)	It includes the X, Y and Z-axis machine coordinate of the fixed tool sensor.

Parameters for Actions after Program End

Parameters that will influence the system actions after machining stops, including prompt method and spindle actions.

Table 13: Parameters Related with Actions after Program End

Parameter Name	Description
Notification Options when Cycle Completed	Inform types when cycle ends. 0: No light on; 1: Red light on for 3 sec; 2: Red light on until any input from mouse or keyboard.
Spindle Action Options when Cycle Completed	Spindle actions when cycle ends. 0: Stay; 1: Move to fixed point; 2: Move to WCS zero.
Spindle Off when Cycle Completed	Whether the spindle stops when cycle completed.
Spindle Off when Cycle Stop	Spindle Off automatically when cycle stops, with or without related command codes in NC file.

Software License Agreement

Important—Read Carefully before Using This Product:

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

Description of Further Rights and Restrictions:

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

Intellectual Property Rights Notice:

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

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

After-sales Guarantee:

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

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

Limitation of Liability:

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

Termination:

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

Applicable Law:

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

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

Weihong Electronic Technology Co., Ltd.