Manufacturer Manual of NK105 Integrated CNC System

(Applicable to NK105G3 Handheld Box)

1 System overview

NK105 Integrated CNC System (Hereinafter referred to as **NK105**) is an integrated CNC system with three or four axes, based on embedded industrial control platform and independently developed by Weihong Company.

According to axes, **NK105** has the following types:

- General **NK105** with three axes: Including X-axis, Y-axis and Z-axis.
- General **NK105** with four axes: Including X-axis, Y-axis, Z-axis and A-axis.
- General **NK105** with double Z axes: Including X-axis, Y-axis, Z1-axis and Z2-axis.

Hardware

- NK105(E) control box with four axes / NK105 control box with three axes

  The two control boxes are almost the same, except the former supports A-axis or Z2-axis.

  This manual takes **NK105(E) control box with four axes** as an example.

- NK105G3 handheld box.

Software

About the main interface of **NK105**, see Main Software Interface.

1.1 NK105 (E) Control Box with Four Axes

This section introduces the schematic diagram and dimension drawing of **NK105 (E) control box with four axes**.

This control box, an integration of the system control card, terminal board and other parts, connects to the handheld box with 15-core connecting cable. And the up and down ends at its back are for terminals and the left side is for the USB interface and DB15 interface.

1.1.1 Schematic Diagram

This schematic diagram of **NK105 (E) control box with four axes** is as follows:
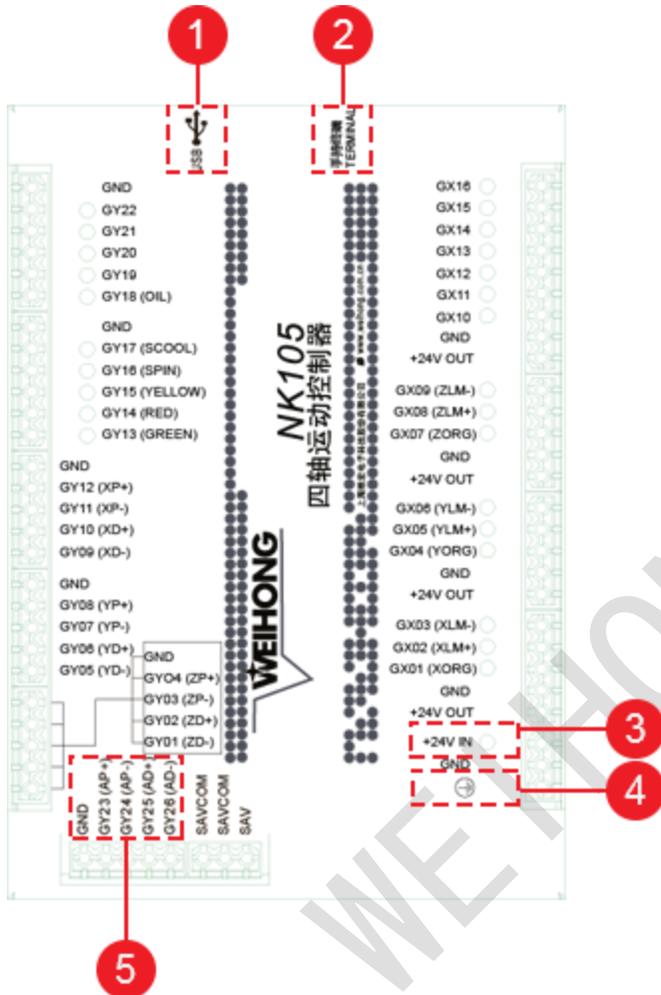
1. USB interface

   It allows **NK105** to communicate with other devices.

2. DB15 interface

   It is used to connect to a handheld box.

3. +24V IN input interface

   It is used to connect to a 24V DC power supply.

4. GND

   It is used to connect to the port GND or COM of a DC power supply.

5.　A-axis differential signals

About terminal definition of the control box, see Terminal Definition.

Note: It is only applicable to **NK105 (E) control box with four axes**.

1.1.2 Dimension Drawing (Unit: mm)

The thickness of **NK105 (E) control box with four axes** is 218.3mm.

The dimension drawing is as follows:
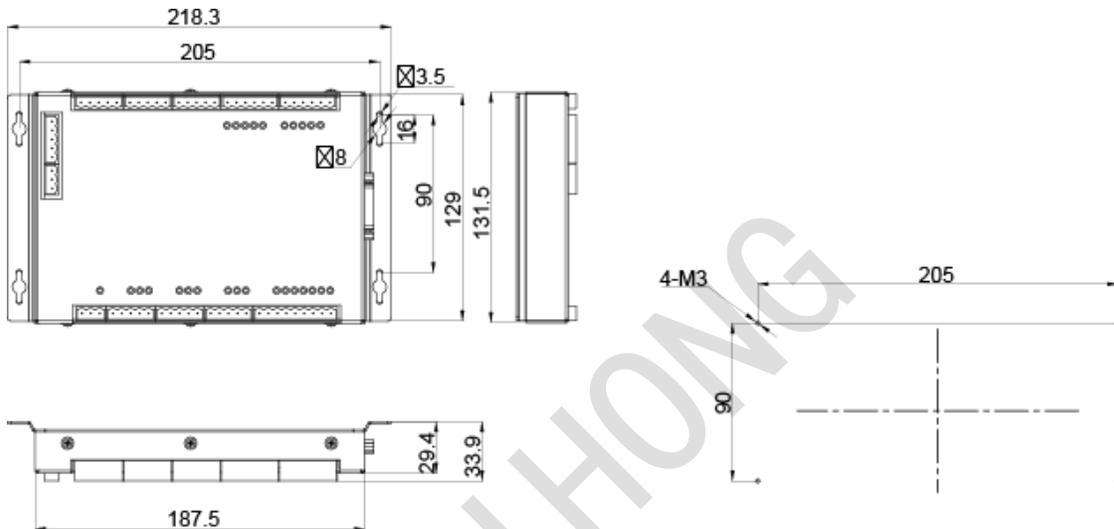
1.2 NK105G3 Handheld Box

This section introduces the button definitions on **NK105G3 handheld box** and dimension drawing of the handheld box.

The handheld box is connected to the control box with a connecting cable, jointly controlling a machine tool. So the moving distance of the handheld box and control box is limited by the connecting cable.

Note: This handheld box is updating. And you can check the newest usage of a single button and button combinations in **Help Information** interface by pressing Shift + ⌘ .
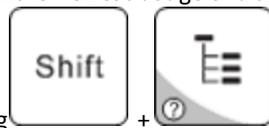
1.2.1 Button Definition

The definitions of buttons on **NK105G3 handheld box** are as follows:

• A single button

− ▶ : It is used to start machining.

− ‖ : It is used to pause machining.

− ■ : It is used to stop machining.

− ▲ : Arrow button (up).

− ▼ : Arrow button (down).

− ◀ : Arrow button (left).

− ▶ : Arrow button (right).

− ⏻ : It is used to enable / disable the spindle.

− : It is used to enter the menu interface, or enter the interface for updating the system image at the time of starting **NK105**.

- **ESC** : It is used to exit the current interface or dialog box, or return to the previous interface.

- **XY=0** : It is used to clear the workpiece coordinates of X-axis and Y-axis at the same time.

- **Z=0** :

  - General **NK105** with four axes: It is used to clear the workpiece coordinate of Z-axis or A-axis.

  - General **NK105** with double Z axes: It is used to clear the workpiece coordinate of Z1-axis.

- **Shift** : It is an auxiliary key or used to switch between **Jog** and **Step** mode in the main interface.

- **⋀⋀+** : It is used to increase the feedrate override.

- **⋀⋀−** : It is used to decrease the feedrate override.
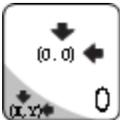
- **(0.0)** : It is used to move X-axis / Y-axis to the workpiece origin or input the figure 0.

- **X−** : It is used to move X-axis in the negative direction or input the figure 4.

− : It is used to move X-axis in the positive direction or input the figure 6.

− : It is used to move Y-axis in the positive direction or input the figure 8.

− : It is used to move Y-axis in the negative direction or input the figure 2.

− :

  • General **NK105** with four axes: It is used to move Z-axis in the positive direction or input the figure 9.

  • General **NK105** with double Z axes: It is used to move Z1-axis in the positive direction or input the figure 9.

− :

  • General **NK105** with four axes: It is used to move Z-axis in the negative direction or input the figure 3.

  • General **NK105** with double Z axes: It is used to move Z1-axis in the negative direction or input the figure 3.

− : It is used to switch between high-speed jog mode and slow jog mode or input the figure 5.

– :

- General **NK105** with four axes: It is used to move A-axis in the positive direction or input the figure 7.

- General **NK105** with double Z axes: It is used to move Z2-axis in the positive direction or input the figure 7.

– :
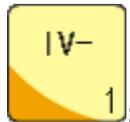
- General **NK105** with four axes: It is used to move A-axis in the negative direction or input the figure 1.

- General **NK105** with double Z axes: It is used to move Z2-axis in the negative direction or input the figure 1.

- Button combination
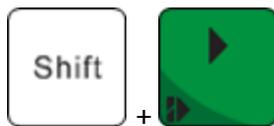
– + : Breakpoint resume.

– + : It is used to show help information.

– + : It is used to clear the workpiece coordinate of X-axis.

– + :
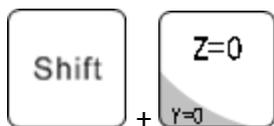
- General **NK105** with four axes: It is used to clear the workpiece coordinate of Y-axis.

- General **NK105** with double Z axes: It is used to clear the workpiece coordinate of Z2-axis.

- **Shift** + **⋁⋁+** : It is used to increase the spindle override.

- **Shift** + **⋁⋁−** : It is used to decrease the spindle override.

- **Shift** + **(0.0)** **0** : It is used to move X-axis or Y-axis to the fixed point.
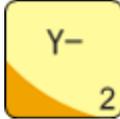
- **Shift** + **Y− 2** : It is used to select the workpiece coordinate system (WCS) from G59 to G54.

- **Shift** + **Z− 3** : It is used to select the WCS from G54 to G59.
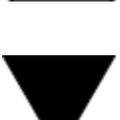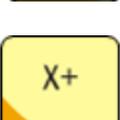
- **▼** + **X+ 6** : It is used to do centering for X-axis.
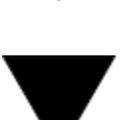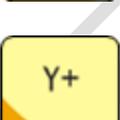
- **▼** + **Y+ 8** : It is used to do centering for Y-axis.
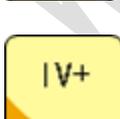
- **Shift** + **IV+ 7** : It is used to return to the machine origin (ie. the origin of the machine coordinate system (MCS)).
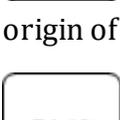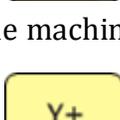
- **Shift** + **Y+ 8** : It is used to switch between workpiece coordinates and machine coordinates .
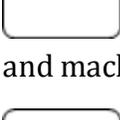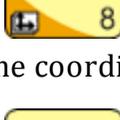
- **Shift** + **Z+ 9** : It is used to calibrate tool.

− **Shift** + **‖** : It is used to make the fine adjustment.

• Button combination (Exclusive to General **NK105** with double Z axes)

− ▼ + **Z=0** : It is used to clear the workpiece coordinate of Y-axis.

− **Shift** + **5** : It is used to activate axes.
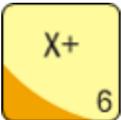
− **Shift** + **6** : It is used to switch between **Alternative** and **Linkage** mode. See Select the Aixs Mode for details.

− **Shift** + **4** : It is used to set the tool length of Z1-axis.
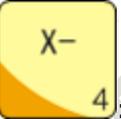
− ▼ + **4** : It is used to set the tool length of Z2-axis.

1.2.2 Dimension Drawing (Unit: mm)

The dimension drawing of **NK105G3 handheld box** is as follows:
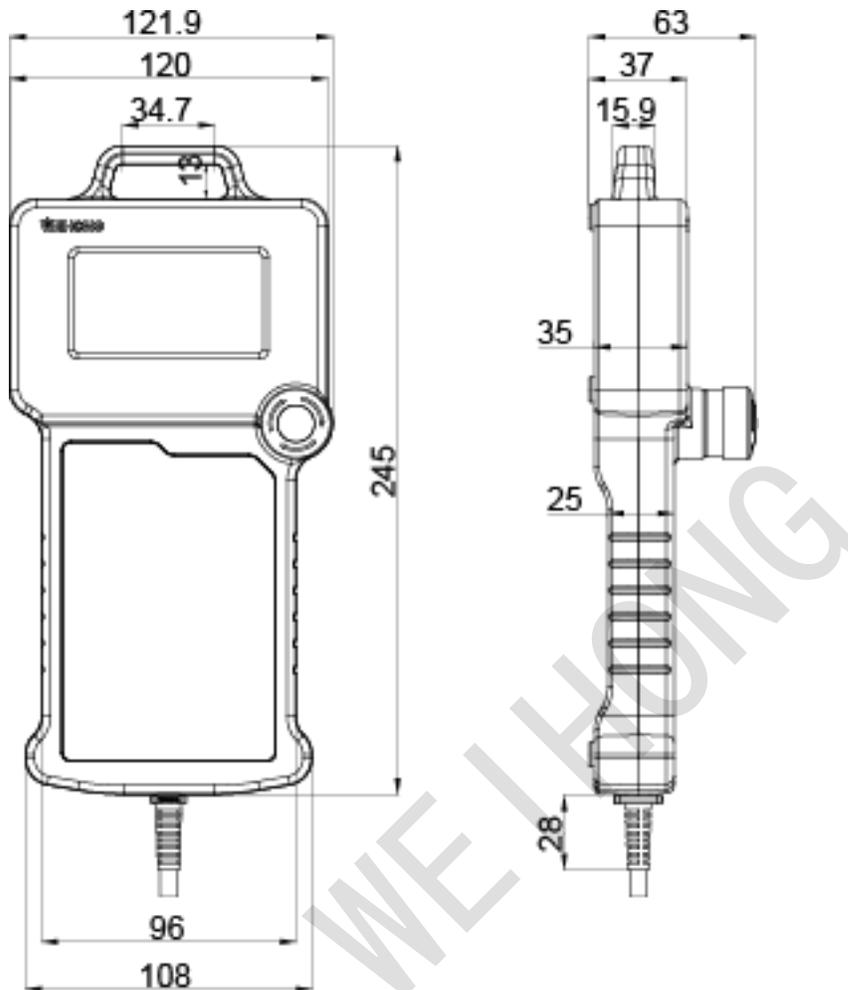
1.3 Main Software Interface

The main software interface differs in axes. General **NK105** with four axes and that with three axes are the same, except the axis number, so their main interfaces are similar. Therefore, this manual does not introduce the main interface of general **NK105** with three axes.

1.3.1 General NK105 with Four Axes

When general **NK105** with four axes starts, the system automatically enters the machining interface in the manual mode (ie. **Jog** or **Step** mode).

The main interface is as follows:

1. Activated axes

   The figures refer to the selected WCS. Specifically, **1 / 2/ 3/ 4/ 5/ 6** respectively refers to **G54/ G55/ G56/ G57/ G58/ G59**.

2. Coordinates

3. Operation status

   – Idle: Default status.
   – Running
   – Pause
   – Stop
   – Locked: **NK105** is locked when the soft limit is triggered.

4. Spindle status

   – Enabled
   – Disabled

5. Speed mode

   – High-speed jog
   – Slow jog mode

6. Motion mode

   – Jog: Finely adjust the machine coordinate.
   – Step: Coarsely adjust the machine coordinate.

1.3.2 General NK105 with Double Z Axes

The main interface is as follows:

1.   Activated axes

2.   Coordinates

3.   Operation status

   –   Idle: Default status.
   –   Running
   –   Pause
   –   Stop
   –   Locked: **NK105** is locked when the soft limit is triggered.

4.   Spindle status and coordinate system

   –   Spindle status

      •   Enabled
      •   Disabled

   –   Coordinate system

      When the coordinate system turns to show the MCS, this area shows G53; when it turns to the WCS, this area shows G54 (default one) / G55 / G56 / G57 / G58 / G59.

5.   Speed mode

   –   High-speed jog
   –   Slow jog mode

6.   Motion mode

   –   Jog: Finely adjust the machine coordinate.
   –   Step: Coarsely adjust the machine coordinate.

2 Wiring

Through this section, you can quick know the supported signal types and terminal definition of **NK105 (E) control box with four axes**, which can help you finish wiring.

2.1 Signal Type

The signal types include the following:

- Binary input signal
- Open collector output signal
- Analog output signal
- Differential output signal

2.1.1 Binary Input Signal

The wiring diagram of the binary input terminal connected to the following switches is as follows:

- Mechanical switch
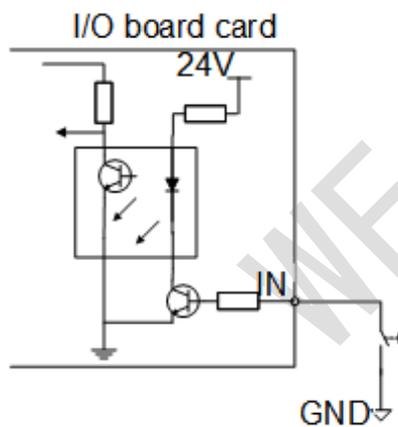
- Photoelectric switch or proximity switch
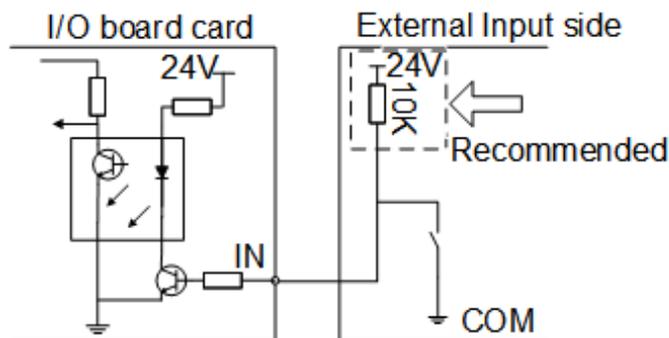
When the binary input terminal is connected to 24V, the input signal is active low:

- For a normal open switch, conducting to GND means the input signal can be received .
- For a normal close switch, disconnecting with GND means the input signal can be received.

2.1.2 Open Collector Output Signal

The output port of **NK105 (E) control box with four axes** belongs to open controller output, with the maximum output current 50mA.

The wiring diagram of the output port connected to a solid-state relay is as follows:
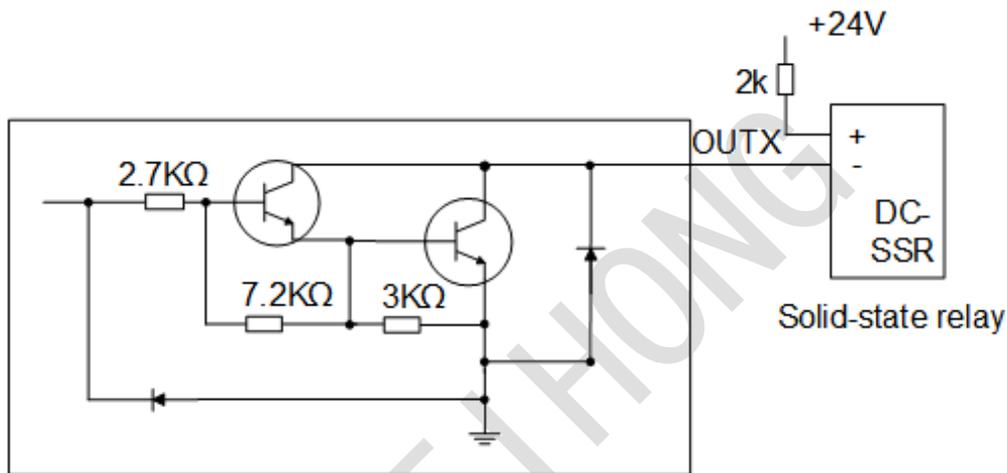
2.1.3 Analog Output Signal

SAV, an output port for 0V ~ 10V controllable voltage, is connected to the input port for analog voltage frequency of the inverter. Therefore, you can control the inverter frequency by changing the controllable voltage, so as to control the spindle speed.

The schematic diagram is as follows:

2.1.4 Differential Output Signal

Differential output signal refers to two equivalent signals with opposite phases from the drive side. You can judge if the logic state of differential signals is **0** or **1** by the voltage difference of these two signals.

The format of pulse commands that control drive motion is pulse + direction, negative logic.

2.2 Terminal Definition

The terminal definition of **NK105 (E) control box with four axes** is as follows：

3 Basic Commissioning

Through this section, you can quickly know the basic commissioning processes of **NK105**.

To do basic commissioning, do the following:

1. Start the system.
2. Select the axis mode. (General **NK105** with double Z axes)
3. Activate the axis. (General **NK105** with double Z axes)
4. Adjust the port polarity.
5. Set pulse equivalent.
6. Adjust the axis direction.
7. Set the workbench travel.
8. Set the machine origin.
9. Set the spindle.

If the modification to parameters requires the manufacturer password, please contact us.

Note: Symbol * before a parameter means the modification requires restarting the system; No Symbol * means the modification immediately takes effect.

3.1 Start the System

Power on and start the system. The prompt *Back to REF. point?* appears:

- If you want to return to the machine origin, press   OK   .

- If not, press   ESC .

3.2 Select the Axis Mode

This operation is applicable to general **NK105** with double Z axes.

The axis mode includes the following:

• Alternative mode: You cannot activate Z1-axis and Z2-axis at the same time.

• Linkage mode: You can activate Z1-axis and Z2-axis at the same time.

To select the axis mode, do the following:

1. To enter the interface for selecting the axis mode, press [Shift] + [X+ 6] .
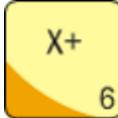
2. To select the target mode, press ▼ / ▲ → [OK] . The selection immediately takes effect.

3.3 Activate the Axis

This operation is applicable to general **NK105** with double Z axes. Z1-axis and Z2-axis are activated at the same time by default.

To activate the axis, do the following:

1. To enter the interface for activating the axis, press [Shift] + [∿ 5] .

2. To select the target axis, press ▼ / ▲ → [OK] . This screen automatically turns to the main interface and Z axes lift up (workbench travel-1):

X	0.000	Idle
Y	0.000	G53
Z1	-1.000	Slow
Z2	-1.000	Jog

The axis with a black background is activated.

3.4 Adjust the Port Polarity

The polarities of input / output ports in the software are specified in terms of the switch type:

- **P**: Normally closed switches.
- **N**: Normally open switches.

To adjust the port polarity, do the following:

1. To enter the menu interface, press [icon].

2. To enter the diagnosis interface, select **8. Diagnosis** by pressing [▼] / [▲], and press [OK].

3. To check the port polarity, select **2. Port List** by pressing [▼] / [▲], and press [OK]:

IN	GX01	○	**N**
IN	GX02	○	N
IN	GX03	○	N
IN	GX04	○	N

4. To modify the port polarity for the target port, select the target port by pressing [▼] / [▲], and press [Shift] → [OK].

3.5 Set Pulse Equivalent

During commissioning, you need to modify pulse equivalent according to the actual situation first before controlling the machine tool.

To set pulse equivalent, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of manufacturer parameters, select **5. Mfr Param** by pressing ▼ / ▲, and press **OK**.

3. To check pulse equivalent, select **3. Pulse Equiv.** by pressing ▼ / ▲, and press **OK**:

```
* P Equiv (mm/p)

X:        0.0010000
Y:        0.0010000
Z:        0.0010000
```
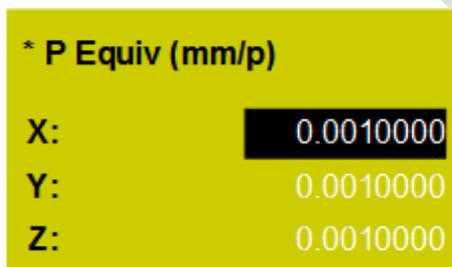
4. To modify pulse equivalent, input the value with the handheld box and press **OK**. The prompt *The parameters are modified. Restart?* appears.

5. To validate the modification, press **OK**. **NK105** automatically restarts.

3.6 Adjust the Axis Direction

This operation is used to check if the positive direction of each axis is the same with that stipulated by **Right Hand Rule**, and adjust it if not, so as to avoid damage to the machine tool due to incorrect direction.

Taking X-axis as an example, to adjust the axis direction, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of manufacturer parameters, select **5. Mfr Param** by pressing [▼] / [▲], and press [OK].

3. To check the X-axis direction, select **2. Axis Output Dir** by pressing [▼] / [▲], and press [OK]:

    *Axis output dir.

    X: Negative
    Y: Positive
    Z: Positive
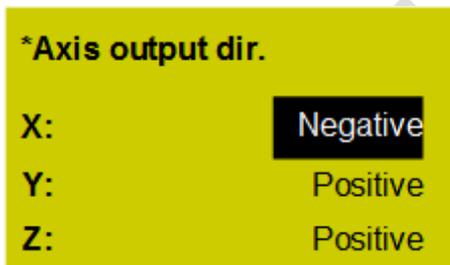
4. According to **Right Hand Rule**, judge the positive direction of X-axis.

5. To check if the positive direction of X-axis is the same with the direction stipulated by **Right Hand Rule**, move X-axis by pressing [X− 4] / [X+ 6], and observe its moving direction：
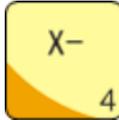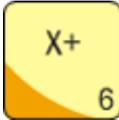
    –   Two directions are the same: The value of parameter **Axis Output Direction** is correct.

    –   Two directions are opposite: Modify the value of parameter **Axis Output Direction** to the opposite value.

3.7 Set the Workbench Travel

The workbench travel refers to the valid motion range of a machine tool, namely valid machining ranges of all axes.

NK105 sets the upper / lower limit of workbench travel as the soft limit, so the workbench travel needs to be identical with the actual dimension. Otherwise, limit overrun and axis crash may occur. If the machining range specified in the program file exceeds the workbench travel, there will be a prompt box about the soft limit. About handling the soft limit, see Handle the Soft Limit.

To set the workbench travel, do the following:

1. To enter the menu interface, press .

2. To enter the interface of manufacturer parameters, select **5. Mfr Param** by pressing / , and press .

3. To check the upper / lower limit of workbench travel, select **4. Machine Stroke** by pressing / , and press .

4. To set the upper / lower limit of workbench travel, select **Strk Upper Lmt** or **Strk Lower Lmt** by pressing / , and input the target value with the handheld box. The prompt *The parameters are modified. Restart?* appears.

5. To validate the setting, press . **NK105** automatically restarts.

WEIHONG

上海维宏电子科技股份有限公司
SHANGHAI WEIHONG ELECTRONIC TECHNOLOGY CO., LTD.

3.8 Set the Machine Origin

The machine origin, a fixed point, is defined through designing, manufacturing and debugging before the machine tool leaves factory.

To set the machine origin, do the following:

1. To enter the menu interface, press ⊡.

2. To enter the interface of manufacturer parameters, select **5. Mfr Param** by

   pressing ▼ / ▲, and press **OK**.

3. To check the parameters of the machine origin, select **7. REF. Point Set** by

   pressing ▼ / ▲, and press **OK**.

   ```
   1. REFP Speed        →
   2. REFP Dir
   3. Retreat Dist
   ```

   About parameter details, see Machine Origin.

4. To set these parameters, select **REFP Speed / REFP Dir / Retract Dist** by

   pressing ▼ / ▲, and input the value with the handheld box. The prompt *The parameters are modified. Restart?* appears.

5. To validate the setting, press **OK**. **NK105** automatically restarts.

3.9 Set the Spindle

This operation is used to set parameters **Spindle NO / OFF Delay**, **Spindle Speed** and **Maximum Spindle Speed**, so as to control the spindle motor.

To set the spindle, do the following:

1. To enter the menu interface, press ⬚.

2. To enter the interface of manufacturer parameters, select **5. Mfr Param** by pressing ▼ / ▲, and press **OK**.

3. To check the spindle parameters, select **8. Spindle Set** by pressing ▼ / ▲, and press **OK**.


```
1. ON/OFF Delay
2. Spindle Speed
3. Max Spdl Speed
```
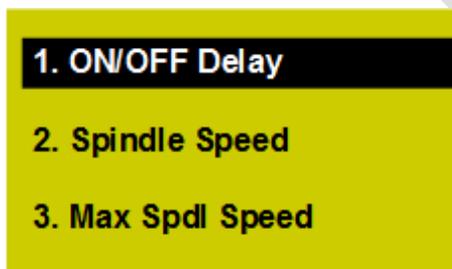
About parameter details, see Spindle.

4. To set the spindle parameters, select **1. ON / OFF Delay** / **2. Spindle Speed** / **3. Max Spdl Speed** by pressing ▼ / ▲, input the value and press **OK**. The prompt *The parameters are modified. Restart?* appears.

5. To validate the setting, press **OK**. **NK105** automatically restarts.

4 Quick Start

Through this section, you can quick know the machining procedure of **NK105**, and then start machining.

To start machining, do the following:

1. Execute returning to the machine origin.
2. Load a program file.
3. Select a WCS.
4. Define the workpiece origin.
5. Set the tool number. (General **NK105** with double Z axes)
6. Execute fixed calibration. (General **NK105** with double Z axes)
7. Start machining.
8. Adjust machining.

4.1 Execute Returning to the Machine Origin

If returning to the machine origin fails due to the origin fault, set the parameter **Back REF First** to **No**. See Returning to the Machine Origin before Machining for details.

To execute returning to the machine origin, do the following:

1. To enter the menu interface, press .

2. To enter the operation interface, select **3. Operations** by pressing / , and press .

3. To enter the axis interface, select **1. Back REF Point** by pressing ▼ /

▲ , and press OK :

1. All Home
2. Z Home
3. X Home
4. Y Home
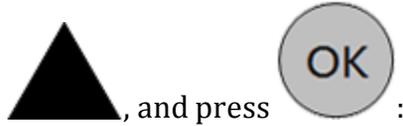
4. Select the target axis by pressing ▼ /▲ , and press OK .

Note: For safety, it is recommended to return Z-axis first, or the prompt *Dangerous operation, you'd better home Z first. Continue?* appears.

4.2 Load a Program File

This operation is used to load a program file in the host or a USB flash disk for machining. The supported file format includes NC, ENG, DXF, PLT, TAP and TXT.

Taking loading a program file in a USB flash disk as an example, to load the program file, do the following:

1. To enter the menu interface, press .

2. To enter the file interface, select **2. USB Files** by pressing ▼ / ▲ ,

    and press **OK** :

    Cycle.dxf
    FLOWER.G   √
    23.nc   √ — ①

    U :   ①LD②De③Copy
    ⑤   ④ ③ ②
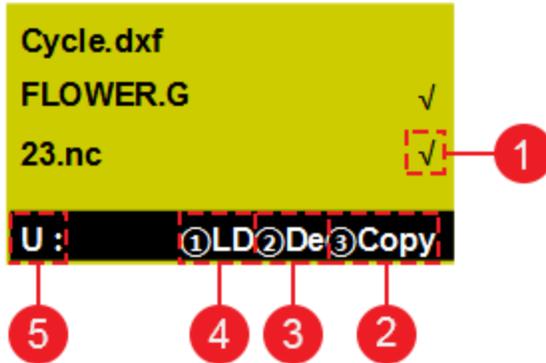
    1. It means the current file is selected.

    2. Copy the selected file.

    3. Delete the selected file.

    4. Load the selected file.

    5. The current disk is a USB flash disk.

3. Select the target program file by pressing ▼ / ▲ , and press

    OK .

4. To load the target program file, press ⌐IV– 1⌐.
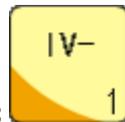
You can also do one of the following for the selected program file in the host or a USB flash disk:

- To delete the program file, press [Y- 2].

- To copy the program file to a USB flash disk or the host, press [Z- 3].

4.3 Select a WCS

In programming, programmers select a certain given point on workpiece as origin (called program origin or workpiece origin) to establish a new WCS.

In general **N105** with four axes, the figures (1 ~ 6) before each axis in the main interface indicate the corresponding WCS (G54 ~ G59).

To select a WCS, do the following:

1. To enter the menu interface, press [ ].

2. To enter the operation interface, select **3. Operations** by pressing ▼ / ▲, and press OK.

3. To enter the interface for selecting a WCS, select **6. Select WCS** by pressing ▼ / ▲, and press OK:

| G54 WCS |
| G55 WCS |
| G56 WCS |
| Select by [OK] key |
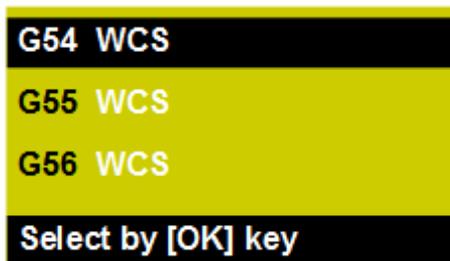
4. Select the target WCS by pressing ▼ / ▲ , and press OK .

4.4 Define the Workpiece Origin

The workpiece origin is the origin of all axes in the program file. This operation is used to define the workpiece origin according to the actual position and do clearing before machining.

This operation differs in the types of **NK105**:

- General **NK105** with four axes
- General **NK105** with Double Z Axes

4.4.1 General NK105 with Four Axes

To define the workpiece origin, do the following:

1. To specify the workpiece origins of X-axis and Y-axis, do one of the following:

    – To clear the coordinates of X-axis and Y-axis at the same time, manually move X-axis and Y-axis to the target position and press XY=0/X=0 .

    – To separately clear the coordinate of X-axis or Y-axis, do one of the following:

        • To clear the coordinate of X-axis, manually move X-axis to the target position and press Shift + XY=0/X=0 .

        • To clear the coordinate of Y-axis, manually move Y-axis to the target position and Shift + Z=0/Y=0 .

2. To specify the workpiece origin of Z-axis, do the following:

    1. To clear the coordinate of Z-axis, manually move Z-axis to the target position and press Z=0/Y=0 .
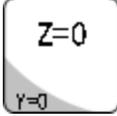
2. To execute tool calibration, press [Shift] + [Z+ / 9]. After calibration, the coordinate of Z-axis is the workpiece origin of Z-axis.

3. To specify the workpiece origin of A-axis, manually move A-axis to the target position and press [Z=0 / Y=0].

4.4.2 General NK105 with Double Z Axes

To define the workpiece origin, do the following:

1. To specify the workpiece origins of X-axis and Y-axis, do one of the following:

   – To clear the coordinates of X-axis and Y-axis at the same time, manually move X-axis and Y-axis to the target position and press [XY=0 / X=0].

   – To separately clear the coordinate of X-axis or Y-axis, do one of the following:

     • To clear the coordinate of X-axis, manually move X-axis to the target position and press [Shift] + [XY=0 / X=0].

     • To clear the coordinate of Y-axis, manually move Y-axis to the target position and press [▼] + [Z=0 / Y=0].
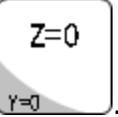
2. To specify the workpiece origin of Z1-axis, manually move Z1-axis to the target position and press [Z=0 / Y=0].
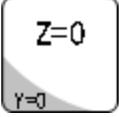
3. To specify the workpiece origin of Z2-axis, manually move Z2-axis to the target position and press [Shift] + [Z=0 / Y=0].
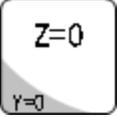
4. Execute fixed calibration for Z1-axis and Z2-axis in turn. After calibration, the coordinates of Z1-axis and Z2-axis are the workpiece origins of Z1-axis and Z2-axis.

About fixed calibration, see Execute Fixed Calibration.

4.5 Set the Tool Number

This operation is applicable to general **NK105** with double Z axes. And it is used to set the same tool number for Z1-axis and Z2-axis.

To set the tool number, do the following:

1. To enter the menu interface, press [icon].

2. To enter the operation interface, select **3. Operations** by pressing [▼] / [▲], and press [OK].

3. To enter the interface for setting tool number, select **10. ToolNo Set** by pressing [▼] / [▲], and press [OK].

4. Select the target axis by pressing [▼] / [▲], input the tool number with the handheld box and press [OK].

After setting the tool number, in the idle / running status, to check the tool number, press [▶] / [OK] in the main interface.

4.6 Execute Fixed Calibration

This operation is applicable to general **NK105** with double Z axes. And it is used to measure the tool on a certain fixed position of the machine tool to reconfirm tool offset. It helps to avoid tool length and the clamping position vary during machining due to tool damage or other causes.

About tool offset parameters, see Tool Change.

You can use the same tool sensor to execute calibration for Z1-axis and Z2-axis in turn.

Before executing fixed calibration, set the tool number.

To execute fixed calibration, do the following:

1. To respectively set the tool length for Z1-axis and Z2-axis, press Shift +
   X– 4 / ▼ + X– 4. The prompt *Set tool length of Z1-axis / Z2-axis, continue?* appears.
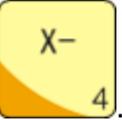
2. Press OK . The prompt *Succeeded in setting tool length of Z1-axis / Z2-axis.* appears.

3. Press OK for confirmation.

4. To reconfirm tool offset, press Shift + Z+ 9.

4.7 Start Machining

This operation is used to start to machine the program file.

During machining, to check the current loaded program file, machining time and

machining line, press in the main interface:

To start machining, press .

4.8 Adjust Machining

During machining, do one of the following according to your need:

- Adjust the feedrate override.
- Adjust the spindle speed.
- Execute the fine adjustment.
- Handle the soft limit.
- Handle the hard limit.

4.8.1 Adjust the Feedrate Override

You can adjust the actual feedrate by the feedrate override. This operation is used to adjust the feedrate override within 0% ~ 120%.

Every time you press related buttons to adjust the feedrate override, the feedrate override increases / decreases 10%.

To adjust the feedrate override, press / .
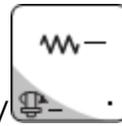
4.8.2 Adjust the Spindle Speed

The spindle speed has 11 gears with the minimum unit 0.1. This operation is used to adjust the spindle speed within 0.5S ~ 1.5S.

To adjust the spindle speed, press [Shift] + [�ᵥᵥ+ −] / [Shift] + [�ᵥᵥ− .].
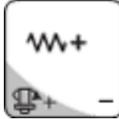
4.8.3 Execute the Fine Adjustment

This operation is used to manually make the fine adjustment in running status or pause status, if the machining is incomplete.

If you temporarily add compensations during machining, it takes effect only in **Auto** mode.

During executing the fine adjustment, the system automatically enters **Step** mode. Every time you press an axis button, the corresponding axis moves a step size.

Before executing the fine adjustment, do one of the following:

• To enter the running status, start machining.

• To enter the pause status after starting machining, press [‖].

To execute the fine adjustment, do the following:

1. To enter the interface of fine adjustment, press [Shift] + [‖].

2. To set the step size, press [Shift] + [‖].

   Step size: 0.01, 0.02, 0.05, 0.10, 0.20, 0.50, 1.00.

3. To make the fine adjustment for the corresponding axis, press [X− 4] / [X+ 6] / [Y− 2] / [Y+ 8] / [Z− 3] / [Z+ 9] / [IV− 1] / [IV+ 7].
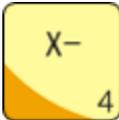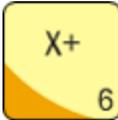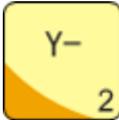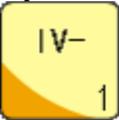
4. To return to the main interface, press ESC .

5. According to the operation status, do one of the following:

   – Running status: Keep machining.

   – Pause status: To continue machining from the pause position, press
    .

If machining normally ends or if you start machining after pausing it, the fine adjustment is always valid.

If one of the following occurs, the file adjustment is invalid:

• If the system enters the idle status.

• If you execute breakpoint resume or start machining whether or not the action that you power on after powering off occurs, after machining stops.

• If you execute returning to the machine origin again and then start machining, after E-stop or limit alarm occurs.

4.8.4 Handle the Soft Limit

If an axis exceeds the upper / lower limit of workbench travel during machining, the soft limit will be triggered.

To handle the soft limit, do the following:

1. To exit the alarm dialog box and return to the main interface, press OK /
   ESC .

2. To release the soft limit, manually move the limit axis in the reverse direction.

   After releasing the soft limit, the system forbids the limit axis to move towards the limit direction.

4.8.5 Handle the Hard Limit

The system regularly checks the hard limit, a limit switch (travel switch). And when the hard limit is triggered, the prompt *Limit. Exit manually.* appears.

To handle the hard limit, do the following:

1. To release the limit, press [OK]. The system automatically enters **Jog** mode, and character **Limit Rls** shows in the lower right corner of main interface:

1X	15.617	Idle
1Y	51.107	Stop
1Z	9.900	Slow
1A	9.900	LimitRls

2. Move the machine tool away from the limit position. Character **Limit Rls** turns to **Jog**.

5 Common Operations

Through this section, you can quick know the common operations of **NK105**.

5.1 Set the Machining Wizard

NK105 supports to machine the rectangular milling face or circle milling face.

To set the machining wizard, do the following:

1. To enter the menu interface, press [menu icon].

2. To enter the operation interface, select **3. Operations** by pressing [▼] / [▲], and press [OK].

3.　To enter the interface of machining wizard, select **2. Prog Wizard** by pressing

▼ / ▲, and press OK :

1. Rect Machining
2. Circle Pocket
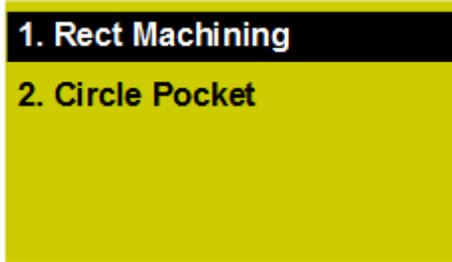
4.　To select a machining method, press ▼ / ▲ → OK :

1. Params Setting
2. Load The Last

–　If you have set related parameters before, to use previous parameter

settings, select **2. Load The Last** by pressing ▼ / ▲, and

press OK .

− If it is your first time to set the machining wizard, do the following:

1. Select **1. Params Setting** by pressing ▼ / ▲ , press **OK** , and set parameters.

   Note: If parameter **Engraving depth** is too large, when a program file is loaded, the warning *Too many file layers generated. Continue?* appears. At this time, it is prohibited to press any button.

   It is default to show the first four parameters, and you can check and set the others by pressing arrow buttons.

   About parameter details, see Machining Wizard.

2. To load the program file, select **Load Now** by pressing ▼ / ▲ , and press **OK** .

5.2 Execute Selective Machining

This operation is used to customize the start line and end line for machining.

To execute selective machining, do the following:

1. To enter the menu interface, press 📋 .

2. To enter the operation interface, select **3. Operations** by pressing ▼ / ▲ , and press **OK** .

3. To enter the interface of selective machining, select **3. Select Line No** by

pressing ▼ / ▲ , and press OK :

> **Total:** 22
> **StartLi** 1
> **EndLine** 22
> —————— **Execute Now** ———
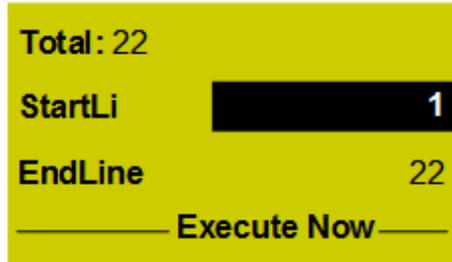
The start line is the breakpoint position of the current file and the end line is the last line by default.

4. Modify the start line and end line with the handheld box and click OK .

5. To immediately start machining, select **Execute Now** by pressing ▼ / ▲ , and press OK .

5.3 Parse the Simulation Range

This operation is used to parse the loaded file for simulation. The system automatically calculates the machining duration and machining range of each axis.

To parse the simulation range, do the following:

1. To enter the menu interface, press .

2. To enter the operation interface, select **3. Operations** by pressing ▼ / ▲ , and press OK .

3. To parse machining ranges, select **4. Machining Info** by pressing ▼ /

▲, and press OK :
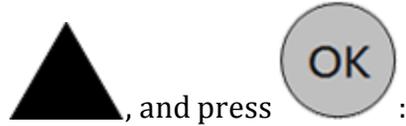
```
Time: 0 : 1 : 42

X :        108      205
Y :         20      117
Z :          0        5
```

5.4 Set the Stop Position

This operation is used to set the stop position for the spindle after machining.

To set the stop position, do the following:

1. To enter the menu interface, press .

2. To enter the operation interface, select **3. Operations** by pressing ▼ /

▲, and press OK .

3. To enter the interface for setting the stop position, select **5. Park MCS Site** by

pressing ▼ / ▲, and press OK .

4. To select the stop mode, select **1. Park Mode** by pressing ▼ / ▲ ,

press **OK** , and do one of the following:

Not Move
To Park Site
To WCS Origin
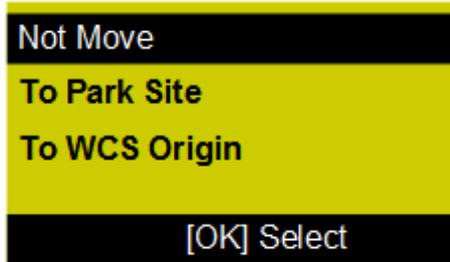
[OK] Select

− Select **Not Move** or **To MCS Origin** by pressing ▼ / ▲ , and

press **OK** .

− Select **To Park Site** by pressing ▼ / ▲ , press **OK** ,
select **2. Park Site**, and set the stop position by inputting coordinates of
each axis or specifying the current position to stop.

5.5 Use Array Machining

This operation is used to set the row number, column number, row space and
column space for array machining. After setting, a program file will be automatically
generated and loaded.

To use array machining, do the following:

1. To enter the menu interface, press 🗎 .

2. To enter the operation interface, select **3. Operations** by pressing ▼ /

▲ , and press **OK** .

3. To enter the interface of array machining, select **7. Array Process** by pressing

▼ / ▲ , and press OK :

On Array	0
Rows	2
Columns	2
Row Space	40.000

4. To enable array machining, select **On Array** by pressing ▼ / ▲ ,

input **1** and press OK :

  – 0: Enable array machining. (Default setting)
  – 1: Disable array machining.

5. To set the row number, column number, row space and column space, select

Rows, **Columns**, **RowSpace**, **Colspace** by pressing ▼ / ▲ , input

corresponding values and press OK .

5.6 Save the Workpiece Origin

This operation is used to save the current position as a workpiece origin.

To save the workpiece origin, do the following:

1. To enter the menu interface, press .

2. To enter the operation interface, select **3. Operations** by pressing ▼ /
▲ , and press OK .
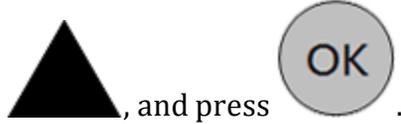
3. To check the list of workpiece origins, select **8. Origin List** by pressing ▼
/ ▲ , and press OK .
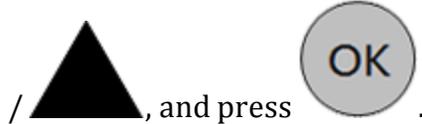
4. Select the target coordinates by pressing ▼ / ▲ , and press
OK → [IV−1] .
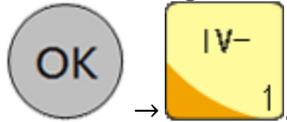
After saving the workpiece origin, according to your need, do one of the following:

- To load the saved workpiece origin, select the target coordinates by pressing
▼ ▲ / , and press OK → [Y−2] .

- To delete the saved workpiece origin, select the target coordinates by pressing
▼ ▲ / , and press OK → [Z−3] .

5.7 Select an Adjacent Point for Machining

This operation is used to select the machining point that is the closest to the current spindle position, and then start machining from this point.

To select an adjacent point for machining, do the following:

1. Manually move the spindle to an adjacent point for machining.

2. To enter the menu interface, press [☰] .

3. To enter the operation interface, select **3. Operations** by pressing ▼ /

▲ , and press OK .

4. To start machining from the adjacent point of spindle, select **9. Nearby Process**

by pressing ▼ / ▲ , and press OK .

6 Parameter Maintenance

6.1 Back up Parameters

This operation is used to back up parameters as a file.

To back up parameters, do the following:

1. To enter the menu interface, press ▤ .

2. To enter the interface of parameter maintenance, select **6. Param Upkeep** by

pressing ▼ / ▲ , and press OK .

3. To back up parameters, select **1. Backup Params** by pressing ▼ /

▲ , and press OK .

The prompt *Successfully saved.* appears.

If the prompt *Failed to save.* appears, try again or contact us.

6.2 Restore Back-up Parameters

Before restoring back-up parameters, ensure you have backed up parameters. Otherwise, the prompt *Cannot find the backup file.* appears.

To restore back-up parameters, do the following:

1. To enter the menu interface, press .

2. To enter the interface of parameter maintenance, select **6. Param Upkeep** by pressing / , and press .

3. To restore back-up parameters, select **2. Backup Params** by pressing / , and press . The prompt *Sure to restore backup* appears.

4. Press for confirmation. The prompt *Reboot Immediately?* appears.

5. Press for confirmation. **NK105** automatically restarts.

6.3 Restore Parameters to Factory Settings

This operation is used to clear all the data and internal parameters stored in the system memory, when internal files are garbled or the system is upgraded.

To restore parameters to factory settings, do the following:

1. To enter the menu interface, press .

2. To enter the interface of parameter maintenance, select **6. Param Upkeep** by pressing / , and press .

3. Select **3. Factory Params** by pressing ▼ / ▲ , and press OK .
The prompt *Restore factory parameters?* appears.

Note: This operation does not delete the backup file. If you delete all internal parameters due to misoperations, you can restore backup parameters.

4. Press OK for confirmation. The prompt *Reboot Immediately?* appears.

5. Press OK for confirmation. **NK105** automatically restarts.

6.4 Export Parameters

This operation is used to export parameters to a USB flash disk, in case of the software or hardware failure.

To export parameters, do the following:

1. To enter the menu interface, press ⊟ .

2. To enter the interface of parameter maintenance, select **6. Param Upkeep** by pressing ▼ / ▲ , and press OK .

3. Select **4. Export Params** by pressing ▼ / ▲ , and press OK .

6.5 Import Parameters

This operation is used to import parameters in the USB flash disk to the system, avoiding setting parameters again.

To import parameters, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of parameter maintenance, select **6. Param Upkeep** by pressing [down] / [up], and press [OK].

3. Select **5. Import Params** by pressing [down] / [up], and press [OK].

4. Press [OK] for confirmation. **NK105** automatically restarts.

6.6 Import the Compensation File

This operation is used to import the file of screw error compensation in th e USB flash disk to the system. During machining, the system automatically executes compensation according to this file.

To import the compensation file, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of parameter maintenance, select **6. Param Upkeep** by pressing [down] / [up], and press [OK].

3. Select **6. Import ErrData** by pressing [down] / [up], and press [OK].

7 System Maintenance

7.1 Switch the Language

NK105 supports Chinese, English and Polish.

To switch the language, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of system maintenance, select **7. System Upkeep** by pressing [icon] / [icon], and press [OK].

3. To check the supported language, select **1. Language** by pressing [icon] / [icon], and press [OK].

4. Select the target language by pressing [icon] / [icon], and press [OK]. **NK105** automatically restarts.

7.2 Export Logs

This operation is used to export the log file `Log.txt` to a USB flash disk.

To export logs, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of system maintenance, select **7. System Upkeep** by pressing [icon] / [icon], and press [OK].

3. To export the log file, select **2. Export Log** by pressing ▼ / ▲ , and

press OK .

The prompt *Log is exported successfully.* appears.

7.3 Update the System

This operation is used to update the system after deleting the parameter file.

To update the system, do the following:

1. To enter the menu interface, press .

2. To enter the interface of system maintenance, select **7. System Upkeep** by

pressing ▼ / ▲ , and press OK .

3. Select **3. System Update** by pressing ▼ / ▲ , and press OK .
   The prompt *Update the system?* appears.

4. To enter the interface for updating the system, press OK :

> **Update Public**
> Delete Parameter
> Start System
> Update System
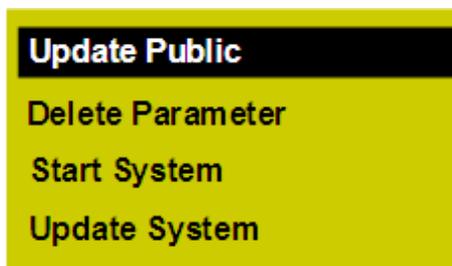
5. To update the system, select **Update System** by pressing ▼ / ▲ ,

and press OK .

In the interface for upgrading the system, according to your need, do one of the

following by pressing ▼ / ▲ , and press OK :

• To update the public file `Public.dat`, select **Update Public**.

• To delete the parameter file, select **Delete Parameter**.

• To start the original system and not to update the system, select **Start System**.

• To back up the system to a USB flash disk as a file folder, select **Export Backup**.

   The default name of the file folder is `backup`.

• To import the parameter file in the root directory of the USB flash disk, select **Import Parameter**.

   If there is no such a file in the root directory, you can search the file `setting.dat` in the file folder `backup`.

• To check the version number of BOOT, select **Version Number**.

7.4 Update the System Application

This operation is used to update the system application, if a breakdown occurs when you get **NK105**.

To update the system application, do the following:

1. Store the target system application to a USB flash disk.

2. Power on and repeatedly press Shift until the prompt *U-disk is available now.* appears.

3. Press OK .

7.5 Update the System Image

This operation is used to update the operation system by the file of system image if a breakdown occurs when you get **NK105**, and its updating process lasts about 3 minutes.

Before updating the system image, ensure the following:

• The format of the USB flash disk is FAT32.

• The file of system image has been copied to the root directory of the USB flash disk.

• The system parameter has been backed up.

To update the system image, do the following:

1. Power on and long press until entering the interface of update selection:

2. To enter the updating menu, press :
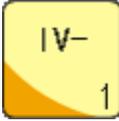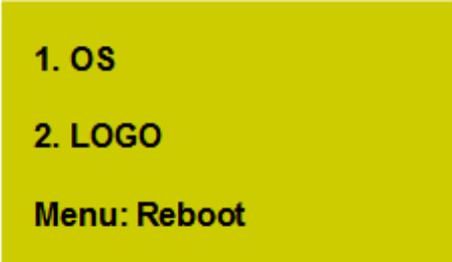
3. To start updating the system image, press .
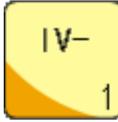
7.6 Use Registration

Before using registration, ensure the following:

• The software can normally start.

• You have generated a registration code with a registration management tool.

    About the registration management tool, contact your developer.

To use registration, do the following:

1. To enter the menu interface, press [icon].

2. To enter the interface of system maintenance, select **7. System Upkeep** by

    pressing [▼] / [▲], and press [OK].

3. To enter the registration interface, select **4. Register** by pressing [▼] /

    [▲] :
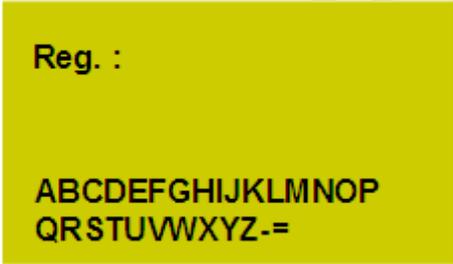
Reg. :

ABCDEFGHIJKLMNOP
QRSTUVWXYZ-=

4. To input the registration code, select letters by arrow buttons and press

    [OK] , and input figures by number buttons.

5. Press [OK] for confirmation.

7.7 Use Writing Numbers

This operation is used to write numbers again if the system type does not match with the hardware.

Before using writing numbers, prepare a USB flash disk with the updated system in its root directory.

To use writing numbers, do the following:

1. Power on and repeatedly press [Shift] until the prompt *USB is available now.* appears.

2. Press [OK] for confirmation. The display enters the updating interface:

System type does not match with hardware platform.
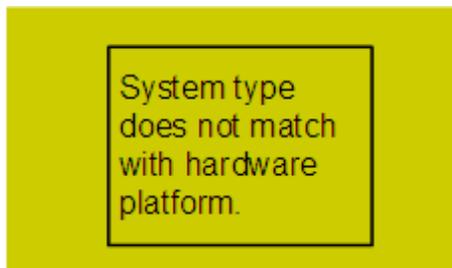
3. To get the registration code, press [OK], check and send the number of board card and self-checking information to us.

Note: During this operation, keep the power on. Otherwise, you need to do the operation again.

4. Press **OK**, input the registration code and wait for the result:

Registering,
Please wait......

– The result is bad: Press **OK** and, input the registration code again.

– The result is good: **NK105** automatically restarts.

7.8 Set Help Interface

This operation is used to set the delay time of **Help** interface popping up, and this delay time is an integer within the range -1 & [1 ~ 999999]. If you set it to **-1**, **Help** interface does not automatically pop up.

To set **Help** interface, do the following:

1. To enter the menu interface, press [menu button].

2. To enter the interface of system maintenance, select **7. System Upkeep** by pressing ▼ / ▲, and press **OK**.

3. To enter the interface for help settings, select **5. Help** by pressing ▼ / ▲.

4. Input the delay time with number buttons, and press **OK**.

7.9 Restart the System

To restart the system, do the following:

1. To enter the menu interface, press ⬚.

2. To enter the interface of system maintenance, select **7. System Upkeep** by

   pressing ▼ / ▲, and press **OK**.

3. Select **6. Reboot** by pressing ▼ / ▲, and press **OK**. **NK105** restarts.

7.10 Exit the System

This operation is used to exit the system. At that time, the screen goes blank. To restart the system after exiting the system, you need to power off first and power on.

To exit the system, do the following:

1. To enter the menu interface, press ⬚.

2. To enter the interface of system maintenance, select **7. System Upkeep** by

   pressing ▼ / ▲, and press **OK**.

3. To exit the system, select **7. Exit** by pressing ▼ / ▲, and press

   OK.

7.11 Delete Logs

This operation is used to delete logs when the free space of the disk is not adequate.

To delete logs, do the following:

1. To enter the menu interface, press ⬚.

2. To enter the interface of system maintenance, select **7. System Upkeep** by
    pressing ▼ / ▲, and press **OK**.

3. Select **8. Delete Log** by pressing ▼ / ▲, and press **OK**.

7.12 Check Disk Space

This operation is used to check total space and used space of the disk, so as to ensure the free space is adequate.

To check disk space, do the following:

1. To enter the menu interface, press ⬚.

2. To enter the interface of system maintenance, select **7. System Upkeep** by
    pressing ▼ / ▲, and press **OK**.

3. To check total disk space and used space, select **9. Disk Space** by pressing
    ▼ / ▲, and press **OK**.

7.13 Modify the Manufacturer Password

If you need the manufacturer password, please contact us.

To modify the manufacturer password, do the following:

1. To enter the menu interface, press .

2. To enter the interface of system maintenance, select **7. System Upkeep** by pressing / , and press .

3. To enter the interface for modifying the manufacturer password, select **10. Modify Code** by pressing / , and press .

4. Input the previous and new manufacturer password, and press .

5. To validate the modification, select **Modify** by pressing / , and press .

8 System Diagnosis

8.1 Check System Information

This operation is used to check system information, including the software version, board card number, remaining time and registered times.

To check system information, do the following:

1. To enter the menu interface, press .

2. To enter the diagnosis interface, select **8. Diagnosis** by pressing / , and press OK .

3. Select **1. System Info** by pressing / , and press OK .

    If one piece of system information cannot be read, the prompt *Failed to read registration information.* appears. At this time, you cannot check all system information. Please contact your developer.

8.2 Check Buttons

This operation is used to check if the buttons on the handheld box are normal.

Taking as an example, to check buttons, do the following:

1. To enter the menu interface, press .

2. To enter the diagnosis interface, select **8. Diagnosis** by pressing / , and press OK .

3. To enter the interface of button diagnosis, select **3. Keypress Diag** by pressing

▼ / ▲ , and press OK .

4. To check the button, press ▲ :

Current Key

Up
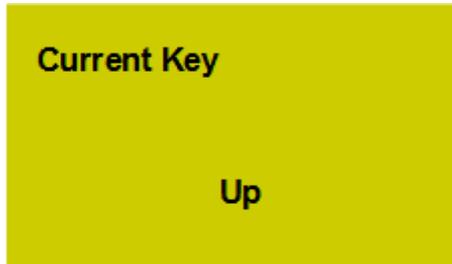
If the button is damaged, nothing or a wrong button name shows in the display.

8.3 Check Input Ports

This operation is used to check the polarity of input ports.

To check input ports, do the following:

1. To enter the menu interface, press ⊟ .

2. To enter the diagnosis interface, select **8. Diagnosis** by pressing ▼ /

▲ , and press OK .
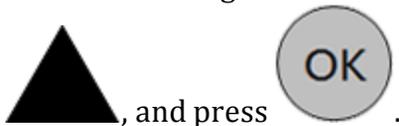
3. Select **4. Inport Diag** by pressing ▼ / ▲, and press OK :

0	1	2	3	4	5	6	7
o	o	o	o	o	o	o	o
8	9	A	B	C	D	E	F
o	o	o	o	o	o	o	o

- **0 ~ 9** and **A ~ F** respectively represents input ports **GX01 ~ GX16**.

- ○ represents the polarity is **P**.

- ● represents the polarity is **N**.

8.4 Check Output Ports

This operation is used to check if output ports are normal by running lights.

To check output ports, do the following:

1. To enter the menu interface, press ⊙.

2. To enter the diagnosis interface, select **8. Diagnosis** by pressing ▼ / ▲, and press OK .
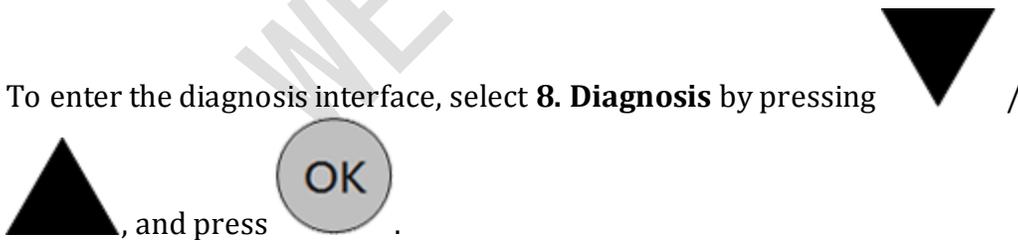
3. Select **5. Outport Diag** by pressing ▼ / ▲, press OK , and check if output ports are normal:

- Yes: The running lights under output ports flash in turn, and the corresponding indication light on the control box flashes.

- No: Contact your developer.

8.5 Check LED Lights

This operation is used to check if LED lights on the handheld box are normal.

To check LED lights, do the following:

1. To enter the menu interface, press [icon].

2. To enter the diagnosis interface, select **8. Diagnosis** by pressing [▼] / [▲], and press [OK].

3. To enter the interface for diagnosing LED lights, select **6. LED Diag** by pressing [▼] / [▲], and press [OK].

4. Press [K1], and check if LED lights are normal:

    – Yes: All LED lights on the handheld box turn on.

    – No: Contact your developer.

9 System Parameters

9.1 Operator Parameters

Operator parameters include the following aspects:

- Machining wizard
- Velocity
- Returning to the machine origin before machining
- Z position at pause
- Offset
- Cycle machining
- Command
- Stopping the Spindle
- Override
- File
- Tool change
- Red light
- Calibration height
- Tool compensation

9.1.1 Machining Wizard

Engr Dpth

- Unit: -
- Range: [Each Dpth, -]
- Default value: -
- Effective: Immediately
- Description: The engraving depth of tools.

Each Dpth

- Unit: -
- Range: [-, Engr Dpth]
- Default value: -
- Effective: Immediately
- Description: Each down depth of tools.

Nose Gap

- Unit: -
- Range: [-, Tool Diameter]
- Default value: 0.5
- Effective: Immediately
- Description: The distance between two adjacent toolpaths.

9.1.2 Velocity

G00 Speed

- Unit: mm/min
- Range: [Feedrate, Max Speed]
- Default value: 3600
- Effective: Immediately
- Description: The default speed of the machine tool during positioning.

Gxx Speed

- Unit: mm/min
- Range: [0, the smaller one of **G00 Speed** and **Max Feedrate**]
- Default value: 3600
- Effective: Immediately
- Description: The default speed of the machine tool during machining.

9.1.3 Returning to the Machine Origin before Machining

Back REF First

- Unit: -
- Range: Yes; No
- Default value: Yes
- Effective: Immediately
- Description: Whether returning to the machine origin before machining is required.

  It is recommended to return to the machine origin before machining, so as to ensure the position accuracy and avoid deviation during machining.

  If the position accuracy can be guaranteed, or you cannot return the axis to the machine origin due to the origin switch error, set this parameter to **No**.

9.1.4 Z Position at Pause

For general **NK105** with double Z axes, the following parameters of Z1-axis should be set to the same value with that of Z2-axis .

Position Option

- Unit: -
- Range: 0; 1; 2
- Default value: 0
- Effective: Immediately
- Description: The positioning mode of Z-axis at pause.

  – 0: to the specified distance.
  – 1: to specified workpiece coordinate.
  – 2: to specified machine coordinate.

Lifting Height

- Unit: mm
- Range: [0, 10000]
- Default value: 10
- Effective: Immediately
- Description: The lifting height of Z-axis at pause when the parameter **Position Option** is set to **1** or **2**.

Z Work Coor

- Unit: mm
- Range: [-10000, 10000]
- Default value: 10
- Effective: Immediately
- Description: The workpiece coordinate of Z-axis at pause when the parameter **Position Option** is set to **1**.

Z Mach Coor

- Unit: mm
- Range: [-10000, 10000]
- Default value: 0
- Effective: Immediately
- Description: The machine coordinate of Z-axis at pause when the parameter **Position Option** is set to **2**.

9.1.5 Offset

Public Offset

- Unit: mm
- Range: [-10000, 10000]
- Default value: 0
- Effective: Immediately
- Description: The public offset of X-axis, Y-axis, Z-axis and A-axis.

Work Offset

- Unit: -
- Range: [-10000, 10000]
- Default value: 0
- Effective: Immediately
- Description: The difference value between workpiece origin and machine origin.

9.1.6 Cycle Machining

Cycle Process

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: Whether to enable cycle machining.

Cycle Times

- Unit: -
- Range: [1, 9999]
- Default value: 2
- Effective: Immediately
- Description: The times for cycle machining.

Cycle Interval

- Unit: ms
- Range: [0, 3600000]
- Default value: 0
- Effective: Immediately
- Description: The time interval between two adjacent cycles.

SOff in Intevl

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: Whether to stop spindle in the cycle interval.

9.1.7 Command

G73_G83 Retract

- Unit: mm
- Range: [0, 1000000]
- Default value: 0
- Effective: Immediately
- Description: The retract or spacing amount of G73_G83 command.

    - G73: The retract amount after each feed.
    - G83: The distance between the feed plane where the tool changes from G00 to Gxx and the previous peck depth.

Ignore F code

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to use the feedrate command in the program file.

Ignore S code

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to use the spindle command in the program file.

9.1.8 Stopping the Spindle

SOFF at Pause

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to stop the spindle when machining pauses.

SOff at Stop

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to stop the spindle when machining stops.

SOff at End

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to stop the spindle when machining ends.

9.1.9 Override

Ration ON MANU

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: Whether the feedrate of manual operations is adjusted by the override.

Ration ON Jigg

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether the feedrate of fine adjustment is adjusted by the override.

9.1.10 File

Including the following:

- DXF File
- ENG File
- PLT File

9.1.10.1 DXF File

Lifting Height

- Unit: mm
- Range: [0, 99999]
- Default value: 5
- Effective: Immediately
- Description: The lifting height of Z-axis during rapid traverse when a DXF file is being processed.

Process Depth

- Unit: mm
- Range: [-99999, 0]
- Default value: 0
- Effective: Immediately
- Description: The machining depth for two-dimensional files.

First Point as 0

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: The first point as workpiece origin when a DXF file is processed.

Shape Process

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: The next shape is not processed until the current shape is finished.

Bottom Process

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: Whether to enable bottom process.

Metric Size

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: Whether to use metric size.

9.1.10.2 ENG File

Lifting Height

- Unit: mm
- Range: [0, 99999]
- Default value: 5
- Effective: Immediately
- Description: The lifting height of Z-axis during rapid traverse when an ENG file is being processed.

Tool Change Tip

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether the prompt about tool change appears.

Deep Hole Mode

- Unit: -
- Range: 0 ; 1
- Default value: 0
- Effective: Immediately
- Description: The mode of machining deep holes.

  - 0: Reciprocating chip removal.
  - 1: High-speed reciprocating chip removal

Retract Amount

- Unit: mm
- Range: [0, 99999999]
- Default value: 1
- Effective: Immediately
- Description: The retract amount after each feed in high-speed reciprocating chip removal mode.

9.1.10.3 PLT File

Lifting Height

- Unit: mm
- Range: [0, 99999]
- Default value: 5
- Effective: Immediately
- Description: The lifting height of Z-axis during rapid traverse when a PLT file is being processed.

Plt Unit

- Unit: -
- Range: [0.001, 99999]
- Default value: 40
- Effective: Immediately
- Description: Normally, 1plt=40.195mm. It can be enlarged or diminished by setting this parameter.

Tool Step

- Unit: mm
- Range: [0.001, 99999]
- Default value: 0.025
- Effective: After restart
- Description: The space of two adjacent tools during machining PLT area.

Processing Depth

- Unit: mm
- Range: [-99999, 0]
- Default value: 0
- Effective: Immediately
- Description: The machining depth for two-dimensional files.

9.1.11 Tool Change

ATC Capacity

- Unit: -
- Range: [1, 20]
- Default value: 10
- Effective: Immediately
- Description: The capacity of the tool magazine.

Current Tool No.

- Unit: -
- Range: [1, ATC Capacity]
- Default value: 1
- Effective: Immediately
- Description: The current tool number.

Tool Offset

- Unit: mm
- Range: [-10000, 10000]
- Default value: 0
- Effective: Immediately
- Description: The tool offset of each axis.

Tool Change Tip

- Unit: -
- Range: Yes ; No
- Default value: No
- Effective: Immediately
- Description: Whether the prompt appears when there is a tool change command in the file.

Cali Coor

- Unit: mm
- Range: -
- Default value: 0
- Effective: Immediately
- Description: The machine coordinate (X/Y/Z) of the tool presetter.

9.1.12 Red Light

Process End Tip

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: Whether the red light turns on for prompt at process ends.

9.1.13 Calibration Height

Cali. Height

- Unit: mm
- Range: [0.001,9999]
- Default value: 1
- Effective: Immediately
- Description: The lifting height after calibration.

9.1.14 Tool Compensation

Tool Comp Enable

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: Whether to enable tool compensation.

Tool Comp Type

- Unit: -
- Range: 1; 2; 3
- Default value: 1
- Effective: Immediately
- Description: The mode of tool compensation.

    - 1: General mode.
    - 2: Intersection mode.
    - 3: Insertion mode.

9.2 Manufacturer Parameters

Manufacturer parameters include the following aspects:

- Velocity
- Axis output direction
- Pulse equivalence
- Workbench travel
- Tool change travel
- Travel limit
- Machine origin
- Spindle
- Rotary axis (General **NK105** with double Z Axes)
- A-axis (General **NK105** with four axes)
- Compensation
- Presetter thickness
- Interpolation algorithm
- Arc increment
- Arc radius tolerance
- Look-ahead segment
- Origin sign
- Safety height
- Lubrication
- G00 feed 100 Percent
- Smoothing time
- Corner option
- Corner tolerance
- Spindle space (General **NK105** with double Z-axes)
- Decreasing time to soft limit
- Smoothing time for override
- Override acceleration
- Deceleration time at pause

9.2.1 Velocity

Decel. Dist.

- Unit: mm
- Range: [0, 999]
- Default value: 10
- Effective: Immediately
- Description: The distance from the decelerating position to the target position.

Approach Speed

- Unit: mm/min
- Range: [Jump Speed, Machining Speed]
- Default value: 600
- Effective: Immediately
- Description: The feedrate when the tool approaches workpiece during positioning (the distance between the tool and workpiece is smaller than deceleration distance).

Run Acc.

- Unit: mm/s^2
- Range: [0.001, 100000]
- Default value: 500
- Effective: Immediately
- Description: The maximum acceleration of each feed axis during machining.

Dry Run Acc.

- Unit: mm/s^2
- Range: [0.001, 100000]
- Default value: 500
- Effective: Immediately
- Description: The maximum acceleration of each feed axis during positioning.

Max. Turn Acc.

- Unit: mm/s^2
- Range: [0.001, 100000]
- Default value: 1000
- Effective: Immediately
- Description: The maximum acceleration of feed motion on adjacent axes.

Jerk

- Unit: mm/s^3
- Range: [0.001, 1000000]
- Default value: 150000
- Effective: Immediately
- Description: The change rate of acceleration of a single axis.

Max Speed

- Unit: mm/min
- Range: [0, max speed]
- Default value: 10000
- Effective: Immediately
- Description: The maximum speed of X-axis, Y-axis and Z-axis.

Short Seg Spd Lmt

- Unit: -
- Range: Yes ; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to enable speed limit for short segments.

Spd Lmt Length

- Unit: mm
- Range: [0, 100000]
- Default value: 0.5
- Effective: Immediately
- Description: When the length of a short segment is smaller than this parameter, the speed to machine this short segment will be limited for machining effect.

Z Down Option

- Unit: -
- Range: 0; 1; 2
- Default value: 0
- Effective: Immediately
- Description: The downward cutting mode of Z-axis.

    - 0: Not disposed.
    - 1: Only Z axis.
    - 2: X-axis, Y-axis and Z-axis synchronize.

Z Plunge Cut Spd

- Unit: mm/min
- Range: [0, Z Max Speed]
- Default value: 300
- Effective: Immediately
- Description: The downward cutting speed of Z-axis under G01 downward cut.

Ref Cir Radius

- Unit: mm
- Range: [0, 100000]
- Default value: 5
- Effective: Immediately
- Description: The reference for machining circular workpieces.

Ref Cir Speed

- Unit: mm/min
- Range: [Min Speed of Arc Machining, Machining Speed]
- Default value: 3000
- Effective: Immediately
- Description: The maximum speed of machining the reference circle without obvious vibration.
- Steps:

  1. Debug the machine tool and machine an arc.

     During machining, the machine tool may vibrate caused by centrifugal force. The higher the machining speed is, the harder the machine tool vibrates.

  2. Increase the feedrate and observe if the machine tool vibrates until you get the maximum speed of machining an arc without obvious vibration. This arc is the reference circle machined with the maximum speed of machine tool without obvious vibration.

  3. Calculate the maximum acceleration of centripetal force (a) based on the radius and maximum speed of the reference circle. The formula is as follows:

$$a = \frac{V_0^2}{R_0} = \frac{V_x^2}{R_x}$$

     - V0: Reference circle speed
     - R0: Reference circle radius
     - Vx: Speed to machine arcs
     - Rx: Arc radius

     Known Rx, when the speed to machine the arc is larger than Vx, the system will automatically limit the speed, avoiding vibration.

Jump Speed

- Unit: mm/min
- Range: [Approach Speed, Machining Speed]
- Default value: 0
- Effective: Immediately
- Description: The maximum speed for the stepper motor at start-up without acceleration.

Look Ahead Dis

- Unit: mm
- Range: [0, 20]
- Default value: 0.5
- Effective: Immediately
- Description: The look-ahead distance for velocity.

The system makes velocity plan based on the analysis of the current point.

Interp Dis

- Unit: mm
- Range: [0, 20]
- Default value: 0.5
- Effective: Immediately
- Description: The look-ahead distance.

The system makes interpolation plan based on the analysis of the current point.

Max Angle

- Unit: deg
- Range: [0, 180]
- Default value: 120
- Effective: Immediately
- Description: The maximum angle for high-speed connection.

  - When connecting angel is larger than this parameters, the system automatically starts machining with jump speed without adjusting angle.
  - When connecting angel is smaller than this parameters, the system calculates proper connecting speed according to connecting angle.

F before Ref

- Unit: mm/min
- Range: [0.06, max speed]
- Default value: 1500
- Effective: Immediately
- Description: The maximum manual speed before returning to machine origin.

9.2.2 Axis Output Direction

Axis Outp Dir

- Unit: -
- Range: Positive; Negative
- Default value: Positive
- Effective: Immediately
- Description: The moving direction of each axis.

9.2.3 Pulse Equivalence

Pulse Equiv.

- Unit: mm/p
- Range: [0.00009, 999]
- Default value: 0.001
- Effective: Immediately
- Description: The workbench travel per pulse sent by the CNC device or the rotary degree of a rotary axis, i.e. the least distance the CNC system can control.

9.2.4 Workbench Travel

Machine Stroke

- Unit: mm
- Range: -
- Default value: 0
- Effective: Immediately
- Description: The valid machining range of each axis.

9.2.5 Tool Change Travel

Strk Upper Lmt

- Unit: mm
- Range: [Lower Limit of Tool Change, 67108.864]
- Default value: 400
- Effective: Immediately
- Description: The machine coordinate of upper limit of workbench travel during tool change.

Strk Lower Lmt

- Unit: mm
- Range: [-67108.864, 67108.864]
- Default value: -100
- Effective: Immediately
- Description: The machine coordinate of lower limit of workbench travel during tool change.

9.2.6 Travel Limit

Travel Limit

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: Whether to enable travel limit of each axis.

9.2.7 Machine Origin

REFP Speed

- Unit: mm/min
- Range: [0.001, Max Speed]
- Default value: -
- Effective: Immediately
- Description: The speed of coarse positioning during returning to machine origin.

REFP Dir

- Unit: -
- Range: Positive; Negative
- Default value: -
- Effective: Immediately
- Description: The direction of coarse positioning during returning to machine origin.

Retract Dist

- Unit: mm
- Range: [0,10000]
- Default value: -
- Effective: Immediately
- Description: The additional motion distance after fine positioning during returning to machine origin.

9.2.8 Spindle

ON/OFF Delay

- Unit: ms
- Range: [0, 60000]
- Default value: 5000
- Effective: Immediately
- Description: The waiting time for the spindle to reach normal speed or completely stop after turning on/off spindle.

Spindle Speed

- Unit: r/min
- Range: [0, Max Spindle Speed]
- Default value: 10000
- Effective: Immediately
- Description: The spindle speed.

Max Spdl Speed

- Unit: r/min
- Range: [0, 999999]
- Default value: 24000
- Effective: Immediately
- Description: The maximum spindle speed.

9.2.9 Rotary Axis

Rotary Axis Type

- Unit: -
- Range: 0; 1; 2
- Default value: 0
- Effective: Immediately
- Description: The type of rotary axes.

  – 0: Disabled
  – 1: X-axis
  – 2: Y-axis

Rotary Axis Pu

- Unit: deg/p
- Range: [0,100]
- Default value: 0.006
- Effective: Immediately
- Description: The pulse equivalent of the rotary axis.

mm as Unit

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: The measure unit for the rotary axis.

Rev. Work Radius

- Unit: mm
- Range: [0, 1000000]
- Default value: 10
- Effective: Immediately
- Description: The workpiece radius on the rotary axis.

Rotary Takeoff

- Unit: rad/s
- Range: [0, 1000000]
- Default value: 0.291
- Effective: Immediately
- Description: The takeoff speed of the rotary axis.

Rotary Axis Acc.

- Unit: rad/s
- Range: [0.001, 1000000]
- Default value: 6.981
- Effective: Immediately
- Description: The acceleration of the rotary axis.

Max Rotary Vel.

- Unit: r/min
- Range: [0.006, 6000000]
- Default value: 30
- Effective: Immediately
- Description: The maximum speed of the rotary axis.

9.2.10 A-axis

A Unit

- Unit: -
- Range: 0; 1
- Default value: 0
- Effective: Immediately
- Description: The measure unit for A-axis in the program file.

    - 0: deg
    - 1: mm

A Takeoff

- Unit: rad/s
- Range: [0, 1000000]
- Default value: 0.3
- Effective: Immediately
- Description: The takeoff speed of A-axis.

A Radius

- Unit: mm
- Range: [0, 1000000]
- Default value: 10
- Effective: Immediately
- Description: The workpiece radius on A-axis.

A Acc.

- Unit: rad/s^2
- Range: [0.001, 1000000]
- Default value: 6.981
- Effective: Immediately
- Description: The acceleration of A-axis.

Max A Speed

- Unit: r/min
- Range: [0.006, 6000000]
- Default value: 30
- Effective: Immediately
- Description: The maximum speed of A-axis.

9.2.11 Compensation

Screw Error Comp

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: Whether to enable screw error compensation.

Enable Backlash

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: Whether to enable backlash compensation.

Axis Backlash

- Unit: mm
- Range: [0, 1000000]
- Default value: 0
- Effective: Immediately
- Description: The backlash compensation amount. And it is valid only when parameter **EnableBacklash** is set to **Yes**.

9.2.12 Presetter Thickness

Cali Thickness

- Unit: mm
- Range: [0, Workbench Travel]
- Default value: 10
- Effective: Immediately
- Description: The presetter thickness.

9.2.13 Interpolation Algorithm

Algorithm

- Unit: -
- Range: 0; 1; 2
- Default value: 1
- Effective: Immediately
- Description: The interpolation algorithm.

  - 0: Trapezoid algorithm
  - 1: S-type algorithm
  - 2: Acceleration trapezoid algorithm

9.2.14 Arc Increment

Arc Increment

- Unit: -
- Range: Yes; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to adopt arc increment mode.

  In arc increment mode, the coordinates of the circle centre are relative to the starting point. Otherwise, they are relative to workpiece origin.

9.2.15 Arc Radius Tolerance

Arc Tolerance

- Unit: mm
- Range: [0.001, 100000]
- Default value: 2
- Effective: Immediately
- Description: In the IJK incremental representation of G02 and G03, the radius is calculated twice. Generally, the two values calculated are not the same and their difference value is called arc radius tolerance.

  Typically, arc instruction does not incur too large tolerance, and the recommended value is about 0.01mm.

9.2.16 Look-ahead Segment

Forward LookSeg

- Unit: -
- Range: [1, 100]
- Default value: 50
- Effective: Immediately
- Description: The maximum look-ahead segments during calculating connection speed.

9.2.17 Origin Sign

Sign of BKREF

- Unit: -
- Range: Yes; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to hide the sign of returning to the machine origin after E-stop.

9.2.18 Safety Height

Safety Height

- Unit: mm
- Range: [0,5000]
- Default value: 10
- Effective: After restart
- Description: The safety height is calculated with respect to workpiece origin. And the horizontal movement at this height is considered to be safe. This parameter is used in breakpoint resume and returning to workpiece origin.

9.2.19 Lubrication

Enable Auto Lube

- Unit: -
- Range: Yes; No
- Default value: No
- Effective: Immediately
- Description: Whether to automatically enable lube at fixed period.

Time Interval

- Unit: s
- Range: [0, 34560000]
- Default value: 5000
- Effective: Immediately
- Description: The time interval between two adjacent lubes.

Duration

- Unit: s
- Range: [0, 34560000]
- Default value: 5
- Effective: Immediately
- Description: The duration to release lube every time.

9.2.20 G00 Feed 100 Percent

G00 Feed 100%

- Unit: -
- Range: Yes; No
- Default value: Yes
- Effective: Immediately
- Description: Whether to enable 100% feedrate override for G00.

9.2.21 Smoothing Time

Smoothing Time

- Unit: s
- Range: [0,0.2]
- Default value: 0
- Effective: Immediately
- Description: The time to smooth toolpath.

  The larger the value is, the smoother the workpiece surface will be. But too large value will affect the dimension of workpiece. So for fine workpiece, its value should not be too large. 0.01 is recommended for a mold machine, and 0.03 for a woodworking machine.

9.2.22 Corner Option

Corner Option

- Unit: -
- Range: 0; 1; 2
- Default value: 0
- Effective: Immediately
- Description: The type of corner smoothing.

  - 0: Not disposed
  - 1: Curve
  - 2: Arc

9.2.23 Corner Tolerance

Corner Toler

• Unit: s
• Range: [0,0.1]
• Default value: 0.1
• Effective: Immediately
• Description: The corner tolerance.

For overall smoothness of workpiece, the tool may not accurately arrive at the specified position at the connection of each two program segments. When the difference value between the tool position and the specified position is equal to the value of this parameter, the system regards the current program segment is machined.

9.2.24 Spindle Space

Spindles Spac

• Unit: mm
• Range: -
• Default value: 0
• Effective: Immediately
• Description: The space between two spindles in X-axis direction.

9.2.25 Decreasing Time to Soft Limit

Soft Limit Time

• Unit: -
• Range: Yes; No
• Default value: Yes
• Effective: Immediately
• Description: The decreasing time to soft limit.

9.2.26 Smoothing Time for Override

Feed Smooth T

• Unit: ms
• Range: [1,50]
• Default value: 20
• Effective: After restart
• Description: The time to smooth override.

9.2.27 Override Acceleration

Max Frp Acc

- Unit: mm/s^2
- Range: [0.01,10000]
- Default value: 1000
- Effective: After restart
- Description: The maximum acceleration during override changes.

9.2.28 Deceleration Time at Pause

Deceleration

- Unit: s
- Range: [0.1,3]
- Default value: 0.3
- Effective: After restart
- Description: The deceleration time from running state to pause state or stop time. Too small value may cause a clash.

10 Drive Parameters and Wiring Diagrams

10.1 Drive Parameters

Drive parameters listed in this section can only make a machine tool normally move, without ensuring machining effects. To get a better machining effects, you need to change the parameter setting according to the specific machine tool.

This part mainly introduces parameter settings for the following drive brands:

- WISE servo drive (Analog·pulse)

- WISE servo drive (M-II Bus)

- YASKAWA Σ–Ⅱ servo drive

- PANASONIC MINAS_A4 servo drive

- MITSUBISHI MR-E servo drive

- DELTA ASDA-A servo drive

- DELTA ASDA-A2 servo drive

- FUJI FALDIC-β servo drive

- STONE GS servo drive

10.2 Wiring Diagrams between NK105(E) Control Box and Different Drives

10.2.1 Stepping Drive with Differential Input
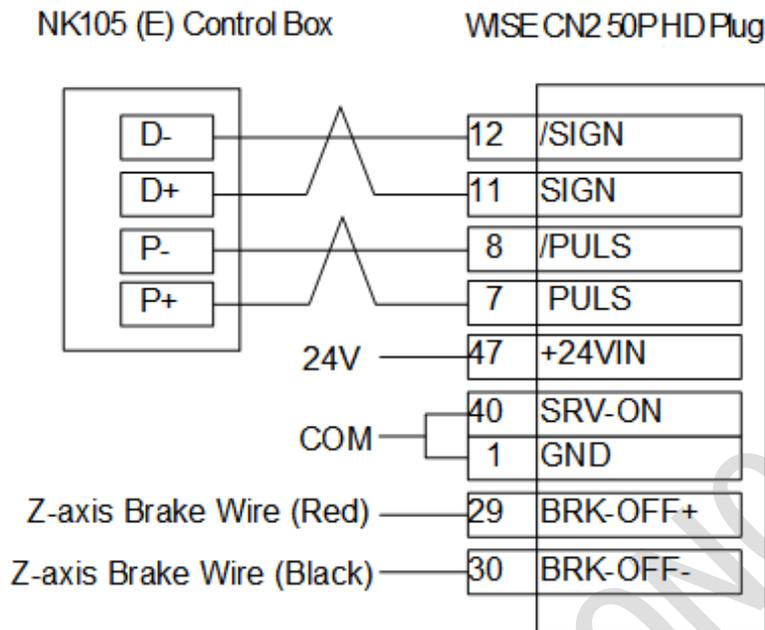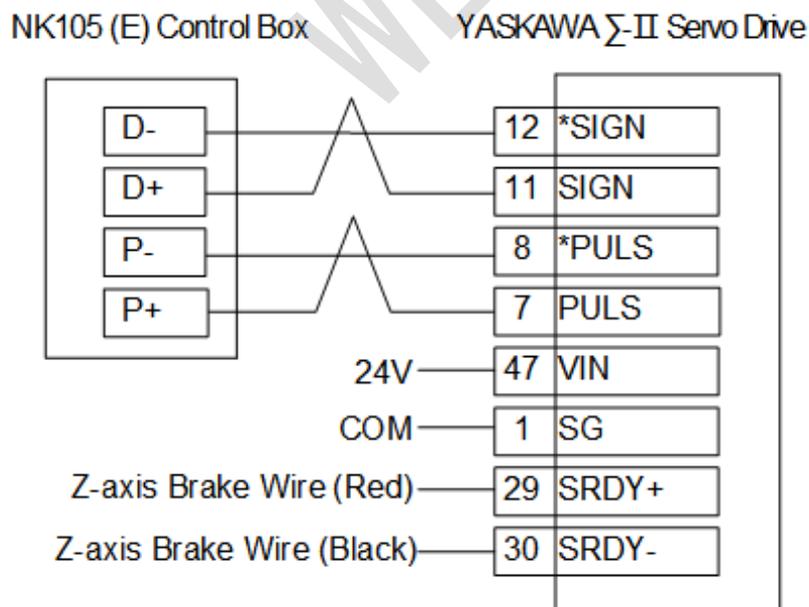
The wiring diagram is as follows:

10.2.2 WISE Servo Drive

The wiring diagram is as follows:
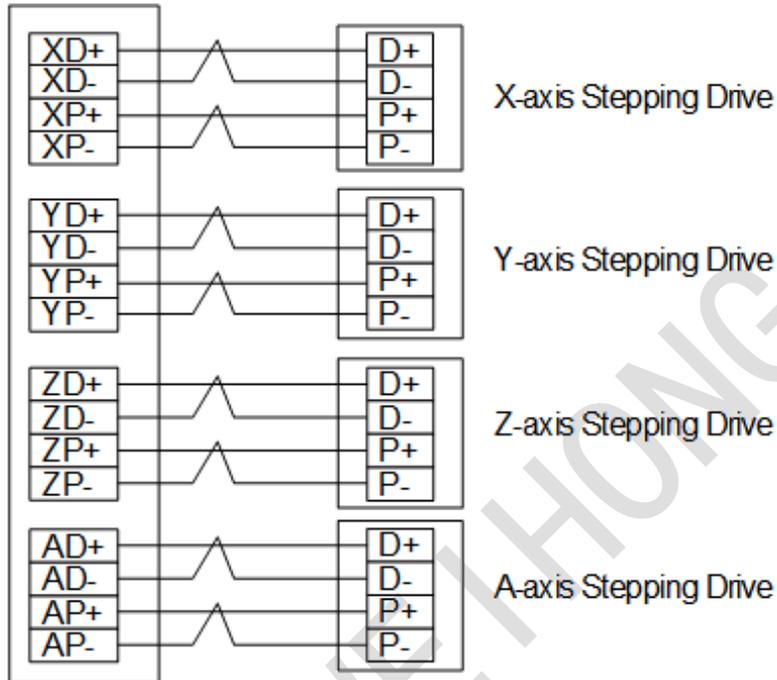
Note: twisted pair is adopted for differential signal
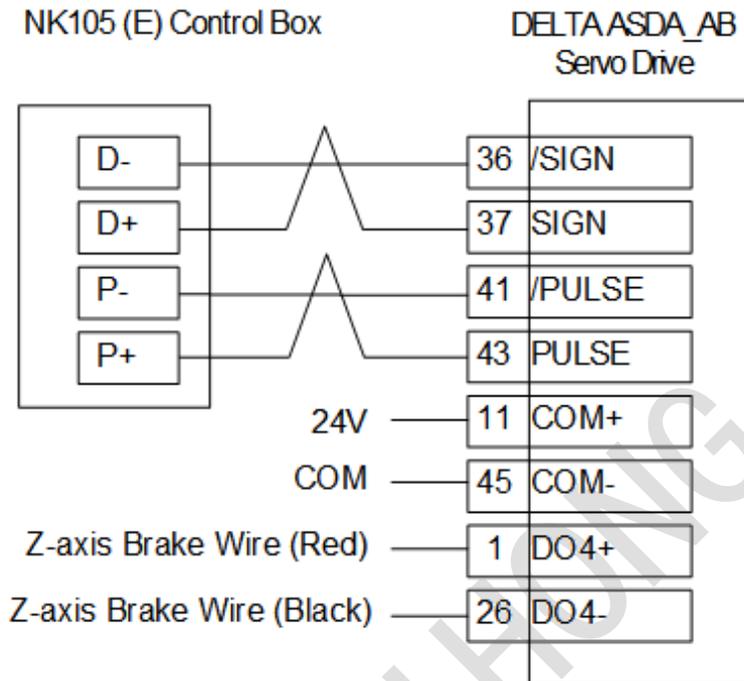
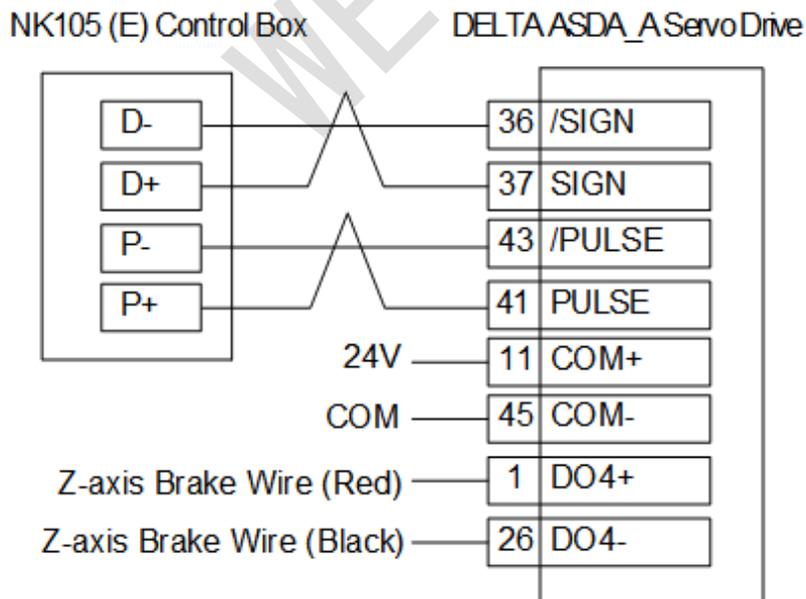## 10.2.3 YASKAWA Σ–Ⅱ Servo Drive

The wiring diagram is as follows:

10.2.4 DELTA ASDA Servo Drive

The wiring diagram is as follows:

- DELTA ASDA-AB

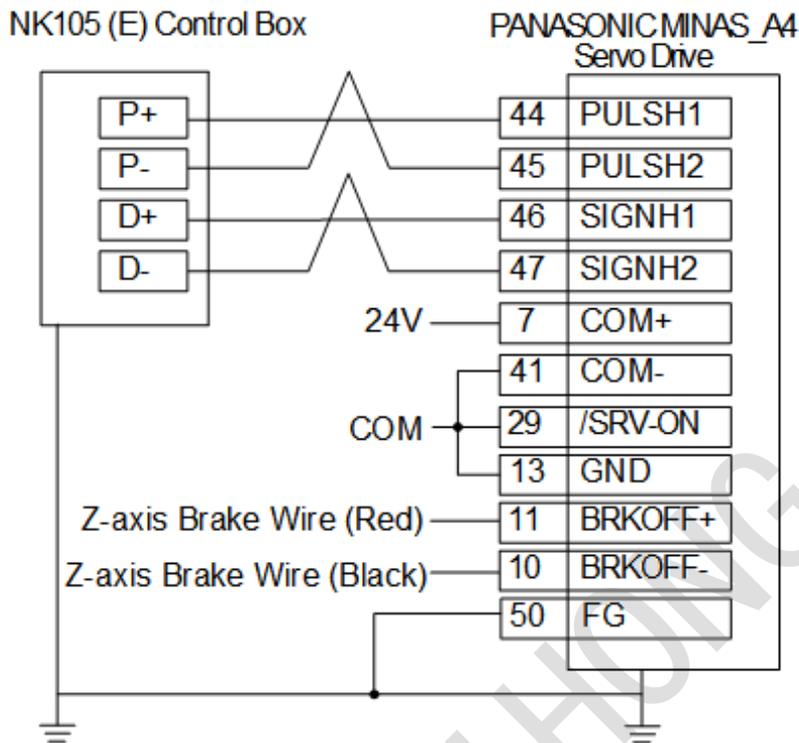- DELTA ASDA-A

10.2.5 PANASONIC MINAS_A4 Servo Drive

The wiring diagram is as follows:

NK105 (E) Control Box — PANASONIC MINAS_A4 Servo Drive

NK105 (E) Control Box		PANASONIC MINAS_A4 Servo Drive
P+	44	PULSH1
P-	45	PULSH2
D+	46	SIGNH1
D-	47	SIGNH2
24V	7	COM+
	41	COM-
COM	29	/SRV-ON
	13	GND
Z-axis Brake Wire (Red)	11	BRKOFF+
Z-axis Brake Wire (Black)	10	BRKOFF-
	50	FG

10.2.6 MITSUBISHI MR-E Servo Drive

The wiring diagram is as follows:

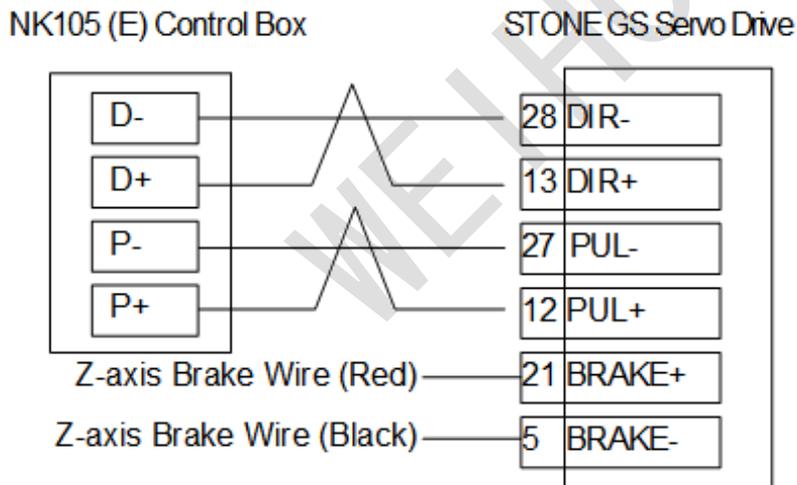| NK105 (E) Control Box		MITSUBISHI MR-E Servo Drive
D-	24	NG
D+	25	NP
P-	22	PG
P+	23	PP
24V	1	VIN
COM	13	SG
Z-axis Brake Wire (Red)	12	MBR
Z-axis Brake Wire (Black)	13	SG
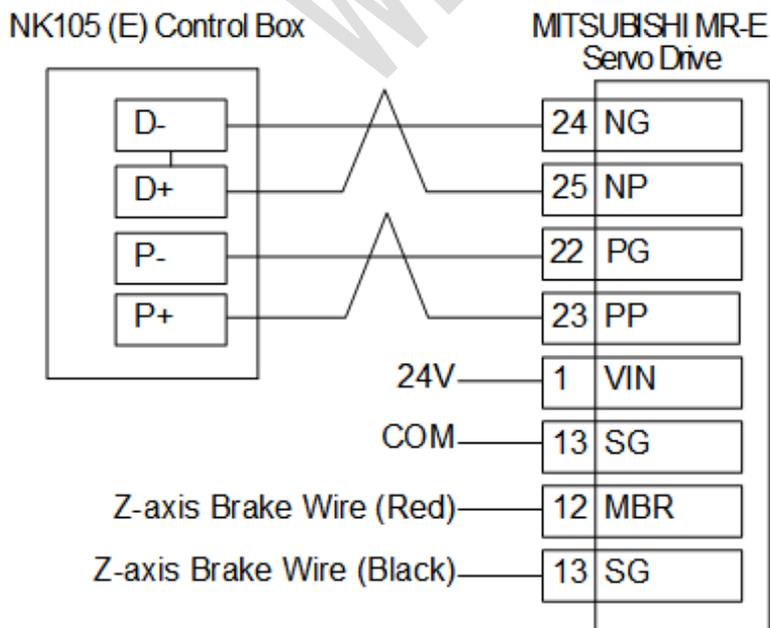
10.2.7 FUJI FALDIC-β Servo Drive

The wiring diagram is as follows:
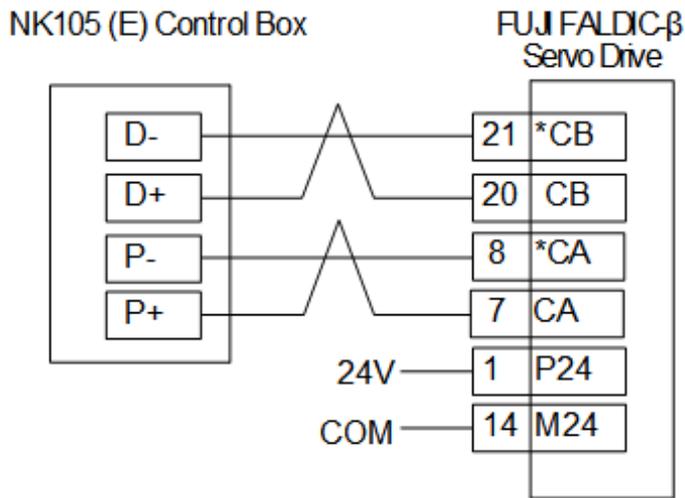
10.2.8 STONE GS Servo Drive

The wiring diagram is as follows:

11 Legal Notices

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.

11.1 Description of Further Rights and Restrictions:

- You may install for use one copy of the Software Product on a single machine;
- You may make a copy of the Software Product for archive or backup purposes and the copy is to be used on this machine only;
- You may transfer the Software Product and the license agreement to a third party, provided that the third party accepts the terms and conditions stated in this agreement, with prior express permission from Weihong Company;
- 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;
- 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;
- You may NOT sublicense, assign or transfer the license agreement;
- You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
- You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
- 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.

11.2 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.

11.3 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.

11.4 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.

11.5 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.

11.6 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.

SPECIALIZED / CONCENTRATED / FOCUSED

Shanghai Weihong Electronic Technology Co., Ltd.

Address: No. 1590, Huhang Rd., Fengxian, Shanghai, China, 201401

Hot-line: 400 882 9188

Website: www.weihong.com.cn/en