HNC-180xpT3
Operation Guide

V 2.0

2015.4

Wuhan Huazhong Numerical Control Co., Ltd

Contents

1 Operation Modes and Display Screen

1.1 Overall Layout

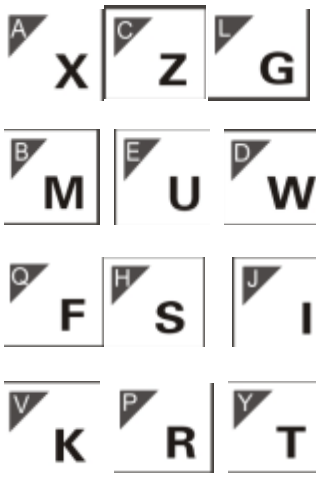
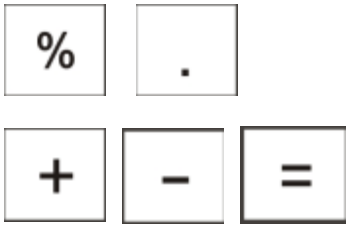
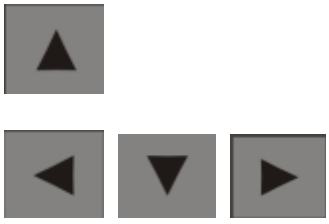
Century Star HNC turning system provides a standard and delicate operation panel, with compact dimension: 420 x 260 x 100 mm (W x H x D). See the figure below:

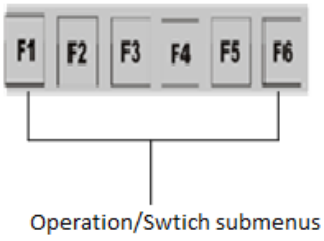
Figure 1.1 Operation panel of HNC turning system

Based on functions, HNC turning system includes HNC-180xpT3 and HNC-808TEA numerical controls.

1.1.1 Manual Data Input Keys

Key	Name	Description
Reset	Rest	Reset the system, stop feeding, clear alarms, etc.
OSD OSD+ OSD−	On-screen display keys (OSD+/OSD-)	Modify the brightness of the screen.
Upper	Upper	Auxiliary function key.
O N	Character input keys	Input a single character.

Key	Name	Description
		Directly input characters. Press the **Upper** key, and then press the keys to input the characters in the upper left corner.
	Numbers/Symbols keys	Directly input numbers. Press the **Upper** key, and then press the keys to input the symbols in the upper right corner.
	Alt	None
	Symbol keys	Input percentage, decimal point, plus, minus, and equal symbols.
	Editing keys	Insert, modify, and delete information for editing programs and fields.
	Direction keys	Move the cursor to the required direction.

Key	Name	Description
PgUp PgDn	Page down and up keys	Under the same screen, move one page up or down as required.
F1 F2 F3 F4 F5 F6	Soft keys	After paging down or up, you can press a soft key to display its submenu and relevant information. See the figure below:  Usage of soft keys: 1. Used to switch submenus on the current screen 2. Used to perform operations on the current submenu such as editing and modifying data or display information

1.1.2 Main Menus

Menu	Description
Prg	the program screen, which consists of eight submenus: OPEN, EDIT, NEW, CUT START, GOTO, VERIFY, RESTART, and BRK.PT. • OPEN: to display, import, delete, open a program, or copy a program from a USB or flash disk. • EDIT: to edit or save an opened program. • NEW: to create a new program. • CUT STAT: to view, edit processing information such as workpiece

Menu	Description
	count and processing duration. • GOTO: to perform operation instructed by a random row in the current program. • VER: to verify a program with simulation running. • RESTART: to restart a program. • BRK.PT.: to resume and save the breakpoint in the program.
Set	Enter the settings screen which consists of five submenus: COORD., CLR RCS, MACRO, CLR ALM, and SKIN. • COORD: to define/view G54 to G59 coordinate information and origin point offset. • CLR RCS: to clear axis values from the relative coordinate system. • MACRO: to display and predefine macro variables. • CLR ALM: to perform alarm settings. • SKIN: to set the background color displayed on the screen.
MDI	The MDI menu consists of one submenu (RETBP) that is used to edit MDI programs and display coordinate information.
Oft	Enter the offset screen which consists of eight submenus: DIA, LEN, XWEAR, ZWEAR, CLR X, CLR Z, X OFF, Z OFF. • DIA: to set X-axis offset for selected tools. • LEN: to set Z-axis offset for selected tools. • X/Z WEAR: to set wear values for selected tools. • CLR X/Z: to clear the defined compensation. • X/Z OFF: to set offset data for X/Y axis.
Dgn	Enter the diagnosis screen which consists of ten submenus: I/O, REG, ALARM, HISTORY, MEMORY, BACKUP, UPDATE, VERSION, REG, and HELP. • I/O: to check Programmable Logic Circuits (PLC) status information. • REG: to view the values of Register Y, R, F, G, B, and P. • ALARM/HISTORY: to check the current alarms and alarm history.

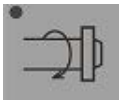
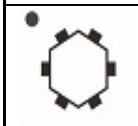
Menu	Description
	• MEMORY: to check the system storage space, memory usage, etc. • BACKUP/UPDATE: to back up and update system software. • VERSION/REGISTER: to view system software version information and perform registration for system software.
Pos	Enter the position screen which consist of six submenus: MCS, WCS, UNION, TEXT, 2D and OTHER. • MCS: to check the position of the current program in the machine coordinate system. • WCS: to check the position of the current program in the workpiece coordinate system. • UNION: to check the position of the current program in the machine coordinate, workpiece coordinate, relative coordinate systems, and distance to go. • TEXT: to check the G code of the current program. • 2D: to check the real-time processing route of the program in an intuitive way. • OTHER: to check the position of the current program in the relative coordinate system.
Par	Enter the parameter screen where you can check and set various parameter information.

1.1.3 Machine Operation Keys

The table below describes the machine operation keys on the operation panel of Century Star HNC Turning System with standard PLC programs.

Key	Name	Function	Description
	AUTO	Enter the automatic operation mode	Automatically and continuously process workpieces; Simulate and verify the processing program; Run command in the

Key	Name	Function	Description
			MDI mode
	Manual	Enter the manual operation mode	Manually change tools, control machine axis, clamp/release chucks, stretch and shrink tailstock, clockwise and counter clockwise rotate spindle, enable and disable coolant, start and stop lubrication, etc.
	Incremental	Enter the incremental operation mode	Step-feed or hand wheel modes, step-feed by default. You may press this key again to enter the hand wheel mode. This key is used to move machine axis at a fixed increment.
	Single Block	Press this key run a program by block.	Valid in the Auto and MDI modes.
	Reference	Enter the Reference mode.	Valid in the Reference mode.
	Start	Press this key to automatically process or simulate processing	Valid in the Auto mode, Single Block mode, and MDI mode.
	Feed-hold	Press this key to pause the feed operation, and press the Start key again to proceed	Valid in the Auto mode and Single Block mode

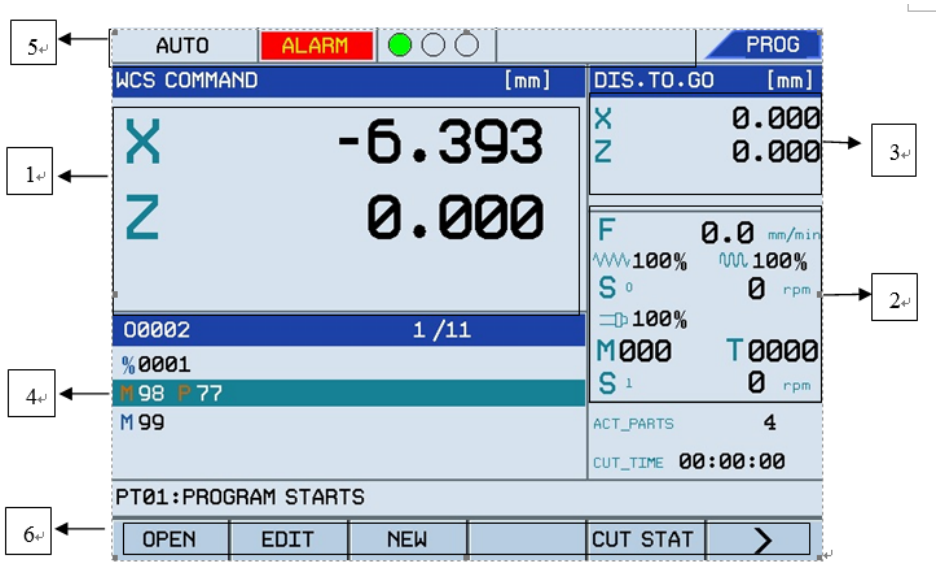
Key	Name	Function	Description
		with the ongoing feed operation.	
	Spindle jog	This key is used to clockwise rotate the spindle at specified speed after each press.	Valid in the manual mode, hand wheel mode, and incremental mode.
  	Spindle control	Press these keys to begin spindle rotation in the clockwise and counter-clockwise direction or stop the spindle rotation.	Valid in the manual mode, hand wheel mode, and incremental mode.
	Block Bypass	Press this key to bypass program blocks that start with a "/" symbol. Disable this key, and the bypass function is invalid.	Valid in the Auto mode, Single Block mode, and MDI mode.
	Turret rotate	After pressing this key, the turret rotates to the selected tool.	Valid in the manual mode, hand wheel mode, and incremental mode.
	Hydraulic control	Enable or disable Hydromantic function	None

Key	Name	Function	Description
	Clamp/release chuck	Press this key to release the workpiece (clamped by default) for workpiece change. Press the key again to clamp the workpiece.	Valid in the manual mode, hand wheel mode, and incremental mode.
	Coolant switch	Press this key to enable or disable the coolant functions. This function is disabled by default.	Valid in the manual mode, hand wheel mode, and incremental mode
	Lubrication control	Press this way to enable the lubrication, and press this key again to disable the lubrication function (disabled by default).	Valid in the manual mode, hand wheel mode, and incremental mode
 	X-axis feed	Press these keys to manually move tools or workstation at the speed specified by the maximum processing speed and feed speed adjustment keys.	Valid in the manual mode, hand wheel mode, and incremental mode
 	Z-axis feed		
	Speed up	Press this key and axis direction keys to speed up tool motion at the specified maximum motion speed.	Valid in the manual mode,

1.1.4 Operation Interface

The figure below shows the operation interface of Century Star HNC Turning System:

Figure 1.2 Operation interface of Century Star HNC Turning System



1. Main interface: Display various information for users such as machine coordinates, graphs, alarm information, register information, and version information.
2. Display various information such as feed rate override, rapid traverse override, spindle override, M command, T command, feed rate, spindle rotation speed, and Metric/Inch programming mode.
3. Display the actual position, distance to go, origin position of the workpiece in the coordinate system. You may use the 

key in the NCP keyboard to switch information.

   Note: **The "distance to go" indicates the distance between the current program end position and the actual position**.
4. Display the G code of the current program.
5. Display the current operation mode, system running status, and three indicators status.
   - **Operation mode**: Switch the operation mode among Auto, Single Block, Manual, Hand Wheel, Reference, Emergency Stop, and Reset.
   - **System running status**: The message "ALARM" will be displayed if an alarm or error occurs.

- **Three indicators**: You may rotate the three adjustment buttons on the operation panel to display the spindle and feed axis operation status in automatic operation. Green light indicates the spindle can rotate and the feed axis can normally move. Yellow light indicates the spindle can rotate but the feed axis cannot move. Red light indicates the spindle cannot rotate and the feed axis cannot move.

6. Display system or PLC messages.

1.2 Overview of Operation Modes

Century Start HNC Turning system provides seven operation mode: Auto, MDI, Single Block, Manual, Reference, Hand Wheel, and Incremental modes.

A. **Automatic mode**

In the Automatic mode, workpiece can be machined automatically based on a program.

B. **Manual Data Input mode**

In the Manual Data Input mode, you may input parameters, and input and execute code.

C. **Single Block mode**

In the Single Block mode, a program can be run based on defined blocks.

D. **Manual mode**

In the manual mode, you may manually perform various operation such as manually feed, speed up, change federate and speed up rate, start and stop spindle, enable and disable coolant and lubrication functions, enable spindle jog, and change tools.

E. **Reference mode**

In the reference mode, each feed axis can home exactly to its reference point.

F. **Hand wheel mode**

In the hand wheel mode, you may control each feed axis motion by manual pulse generator.

G. **Incremental mode**

In the hand wheel mode, press the hand wheel key again to switch to the incremental mode. In the incremental mode, the tool can be moved a number of steps along any axis by pressing the Axis keys. The incremental ratio can be defined by adjusting the feed override ratio.

1.3 Display Screen

This section describes how to switch among submenus, relationship between operations and soft

keys, and detailed operation methods.

Century Star HNC turning system provides seven functional keys (Prg , Set , MDI , Oft , Dgn , Pos , and Par). Each functional key corresponds a main screen which consists of multiple submenus.

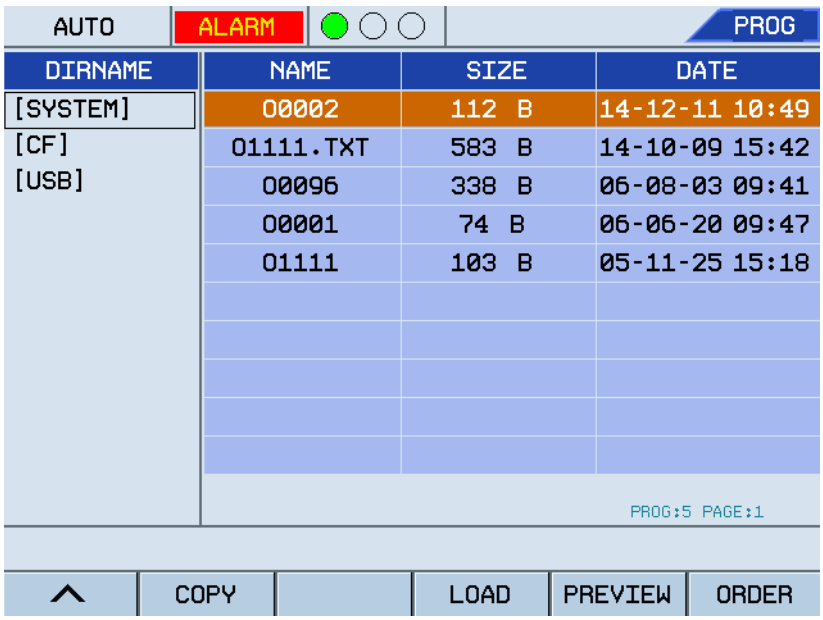
1.3.1 Program

Press the Prg key to enter the programming screen which consists of OPEN , EDIT , NEW , CUT STAT , and > .. You may press keys from F1 to F6 to enter corresponding submenus as required.

1. OPEN

Press the Prg key and then the **F1** OPEN key to enter the screen that displays programs on each storage. See the figure below:

Figure 1.3 Select a program



The storage cards include:

- SYSTEM: programs in system built-in disk

- CF: programs in the CF card
- USB: programs in the USB

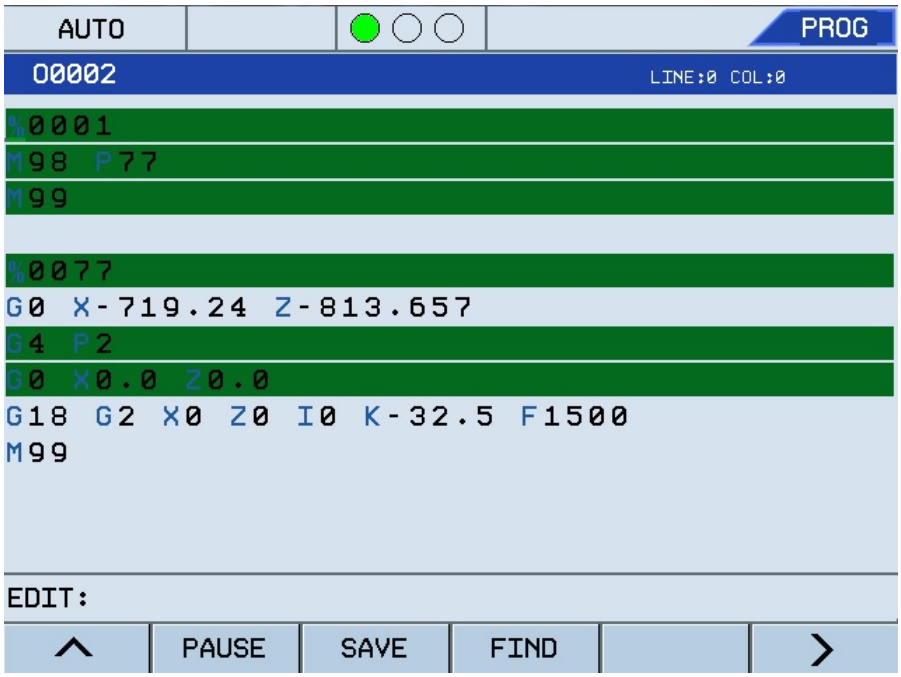
If no storage card is selected, the programs saved in the buffer space previously will be displayed.

Press  and  to switch the program storage card (can display only the programs in the root directory) and program list screens. Press the  ,  , PgUp and PgDn keys to select programs to be run, and press  or  to confirm and add the selected programs. In the Auto or Single Block mode, press  to run the program. If an error occurs, the message "ALARM" will be displayed and the indicator blinks continuously. You may press  to check alarm information.

2. EDIT

Press the  key, select a program to be edited, and then press the F2  key to enter the screen that displays programs on each storage card. See the figure below:

Figure 1.4 Select a program to edit



The shortcut keys used for editing are as below:

Delete: Delete the character behind the cursor and maintain the cursor position. The remaining characters to the right of the cursor will move left with one-character space.

PgUp: Move the program to be edited one screen up to the program header and maintain the cursor position. If it has reached the program header, the cursor will move to the first character in the first row of the program.

PgDn: Move the program to be edited one screen down to the program end and maintain the cursor position. If it has reached the program end, the cursor will move to the first character in the last row of the program.

BS: Delete the character in front of the cursor and the cursor will move left with one-character space. The remaining characters to the right of the cursor will move left with one-character space.

 : The cursor moves left with one-character space.

 : The cursor moves right with one-character space.

 : The cursor moves one row upward.

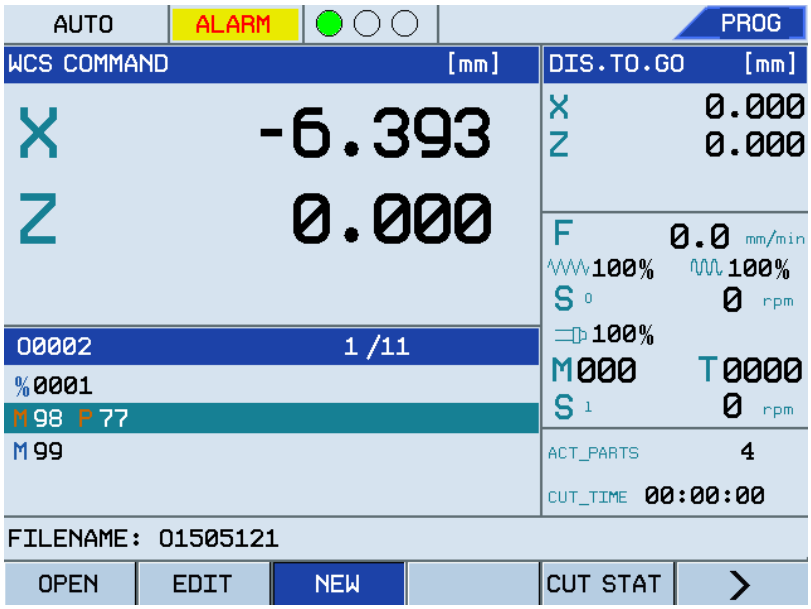
 : The cursor moves one row downward.

Note: The program block displayed in dark color tone cannot be directly edited. You need to press F4 PAUSE, and then press **Y** or Enter after the message "Exit the current running Y/N? (Y)" is displayed. After that, you may edit the program.

3. NEW

Press the Prg key, and then press the F3 NEW key. The system will prompt you to enter a new program name (a random G code name by default). After entering the program name, you may press Enter to create a new program by pressing the character keys on the NCP board. See the figure below:

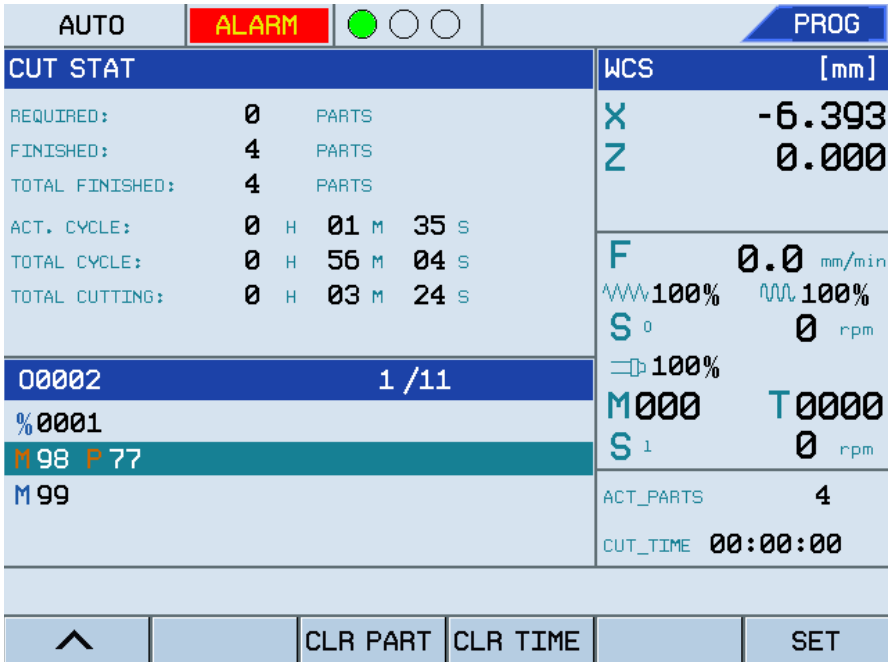
Figure 1.5 Create the selected program



4. CUT STAT

Press the Prg key and then press F5 CUT STAT to enter the screen that displays various running statistics information. This function is used to automatically perform statistics of workpiece count and system running time, which is helpful for improving machine efficiency and work time statistics. See the figure below:

Figure 1.6 Check systemprocessing information



The information in the screen is described as below:

REQUIRED: The total count of workpiece (of the same type) required to be processed.

FINISHED: This indicates the workpieces that have been processed since the machine starts. The processing period indicates that the program executed a M64 (or M30, depending on the machine parameter 001021), and then the completed workpiece count will increase by one.

TOTAL FINISHED: The total count of workpiece that have been currently completed and previously completed.

ACT CYCLE: The running duration since the machine starts.

TOTAL CYCLE: The current running duration together with previous running duration.

TOTAL CUTTING: The current cutting duration together with previous cutting duration.

Under the Prg main menu, press F5 CUT STAT, and then press **F3** CLR PART or **F4** CLR TIME to clear the workpiece count and time.

In the Prg menu, press F5 CUT STAT, press **F6** SET, and enter the total workpiece count required to be processed. Press the Enter key to confirm the settings.

1.3.2 Set

Press the Set key to enter the settings screen which consists of COORD., CLR RCS., MACRO, CLR ALM., and SKIN submenus. You may press keys from F1 to F6 to enter corresponding submenus as required. See the figure below:

1. COORD.

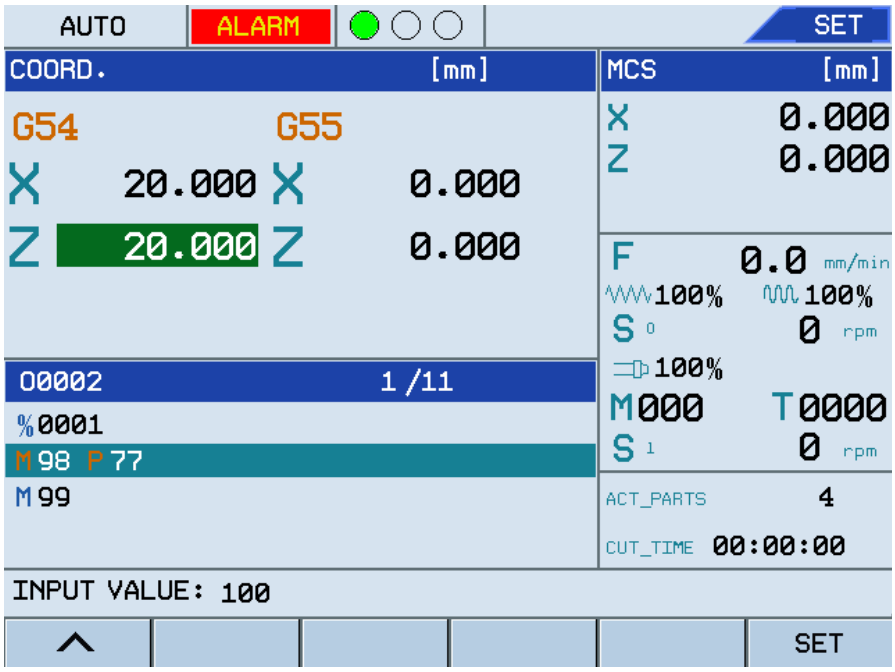
This is used to set X and Z axis in the workpiece coordinate system when cutting a sample workpiece.

a. Press F1 under the Set main menu.

b. Press ◀ and ▶ to select the coordinate type to set. For example, G54-G59 and G92. Press ▲ and ▼ to select X or Z axis to set.

c. Enter required coordinate information, and press Enter. If you press Enter or F6 in the current position as shown in the figure below, enter **20**, and then press Enter, then the X axis value for G54 will be set to **20**. Use the same method to set the value for Z axis. The settings for other coordinates is similar like this.

Figure 1.7 Set X/Y axis value



d. Set floating origin of machine

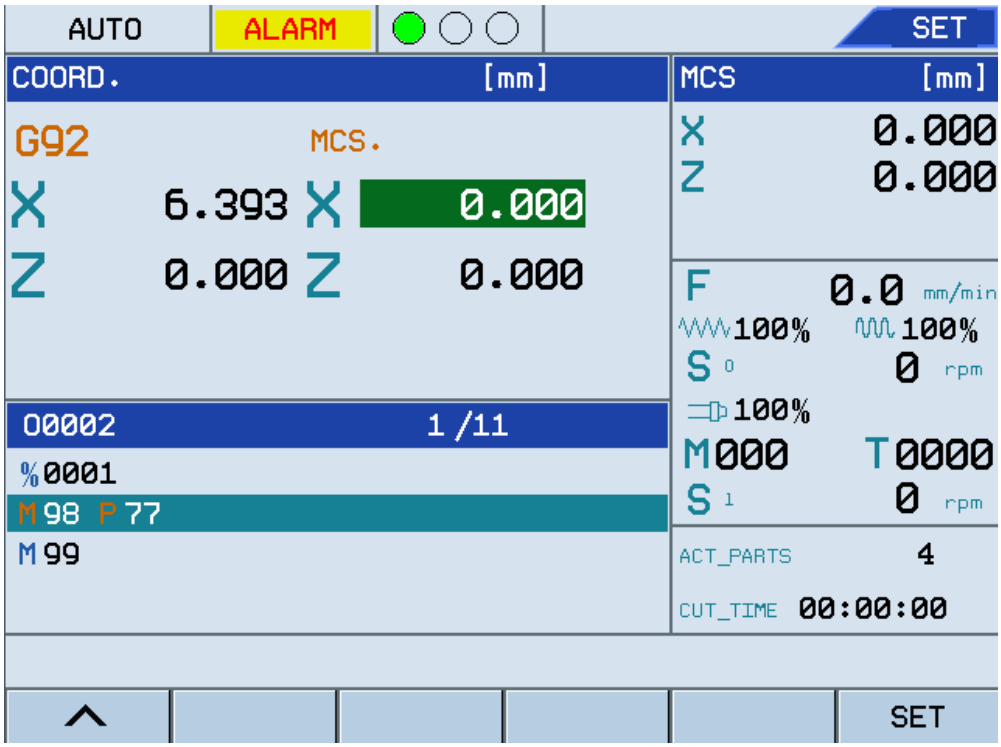
Based on whether a Home Dog is installed, there are two methods to set the floating origin of machine coordinate.

Method 1: If no Home Dog is defined to home the tools to the origin position (specified by the PLC parameter **0070**17, default value: **0**), you may set the floating origin based on the instructions below:

- In the manual mode, move the tool to a position where the tools will not conflict with the workpiece or other parts, and that is appropriate to home to the origin.
- Confirm that the position is the origin of the machine, that is, the actual coordinate value of the point is **0**.
- Press the Set key in the main menu, press the F1 COORD. submenu, and press the ◀ and ▶ keys to switch to the floating origin.
- After that, press the ▲ and ▼ keys to select X or Y axis, enter relative

coordinates, and press the Enter key. See the figure below:

Figure 1.8 Set floating origin



Method 2: If the Home Dog is defined to home the tools to the origin position (The PLC parameter **007017** is set to **1**), the origin of the machine coordinate system depends on the position of the Home Dog, which is near the maximum movement distance in the positive directions of X axis and Y axis. The position of Home Dog is fixed and the origin of the machine coordinate system is fixed. The tool can be homed to the same position every time you press the home key if the Home Dog is normal.

Note: After the floating origin is defined, the machine coordinate system is set. For the CNC system without Home Dog, you need to set the floating origin first when you power on the machine. Normally, you need to set only once, and then the tool can be homed to the same position every time you press the home key.

2. CLR RCS

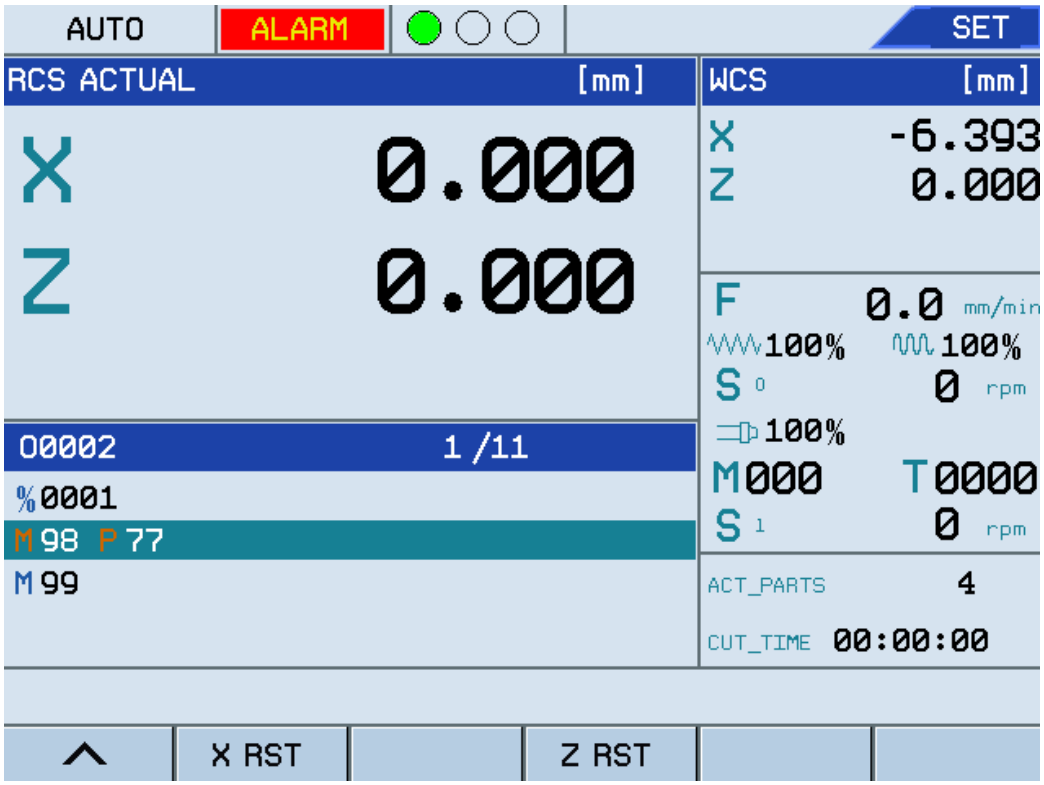
Follow the instructions to clear the axis values in the relative coordinate system:

a. Press the Set menu, and then press F3 CLR RCS.. See Figure 1.9.

b. Press F2, and then set the X axis value to **0** for the relative coordinate system.

c. Press F4, and then set the Z axis value to **0** for the relative coordinate system.

Figure 1.9 Clear axis value in the relative coordinate system



3. MACRO

Press Set, and then press F4 to enter the macro variables screen.

The macro variables screen consists of VAR 1, VAR 2, VAR 3, and other variables. VAR 1 includes 50 variables (**#1250** to **#1299)**. VAR 2 includes 50 macro variables (**#1350** to **#1399)**, and VAR 3 includes 50 macro variables (**#1200** to **#1202)**. You may press PgUp and PgDn keys to display the subscreens as requried, and you may set macro variables either by macro code or directly set on the keyboard. The figures below shows the variables:

Figure 1.10 VAR 1

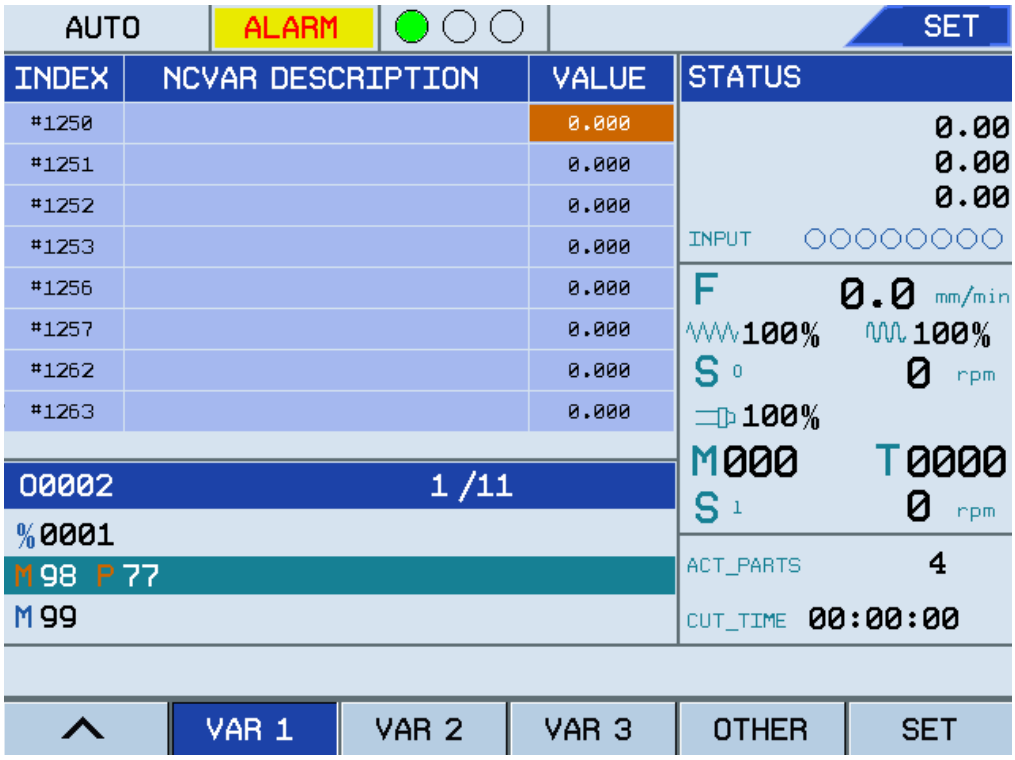


Figure 1.11 VAR 2

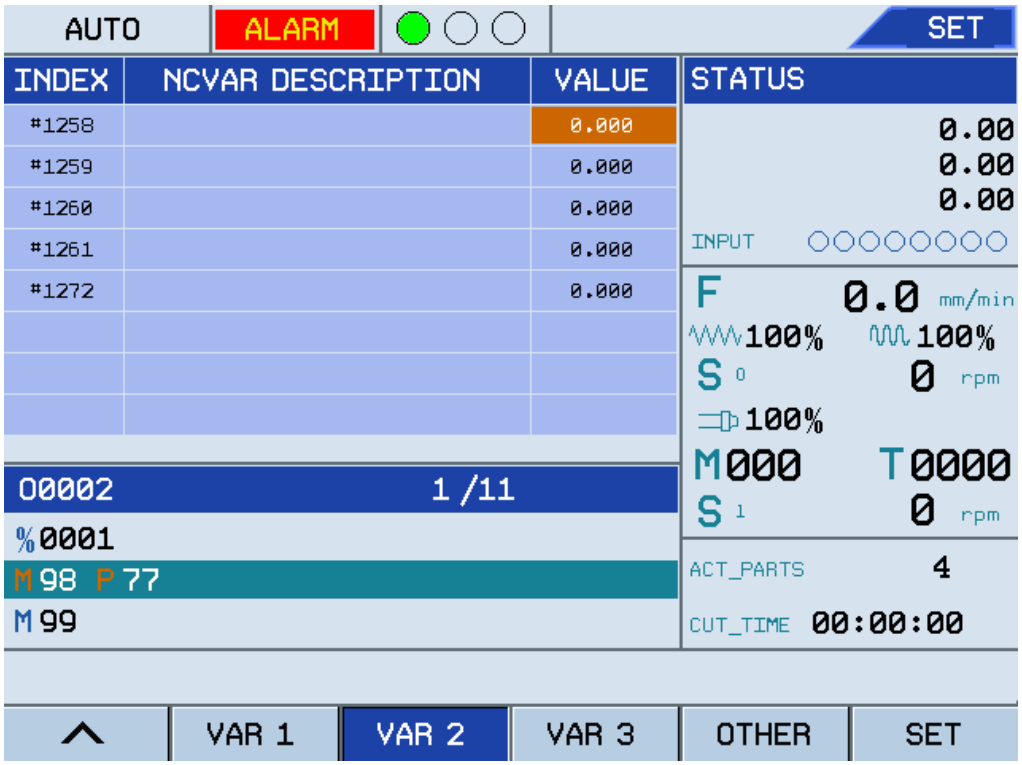


Figure 1.12 VAR 3

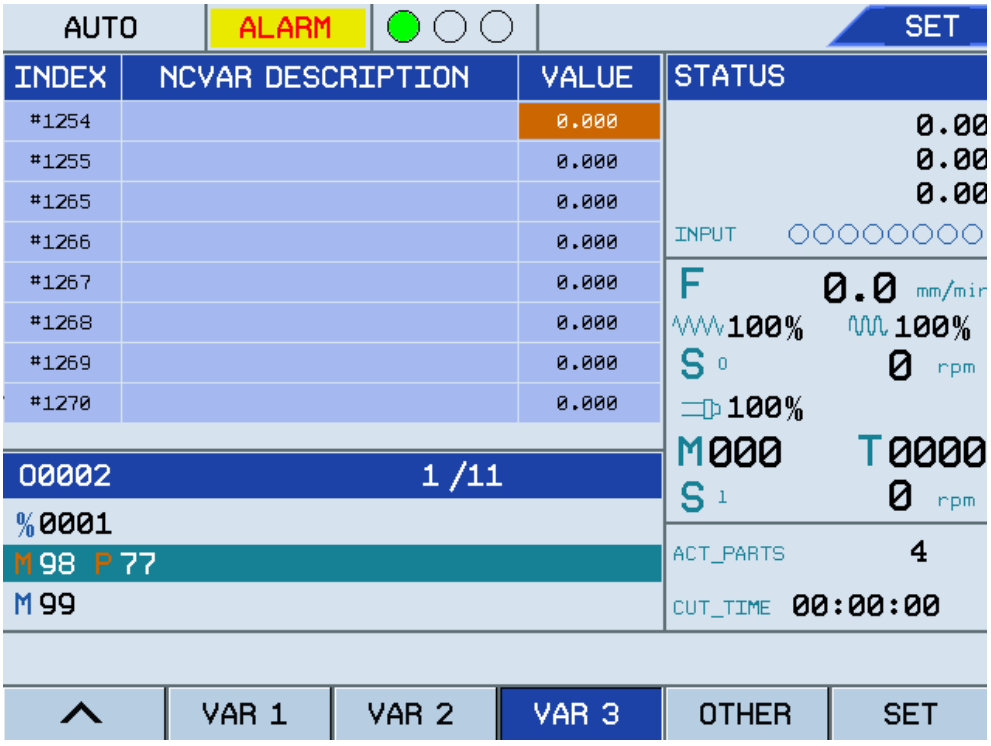
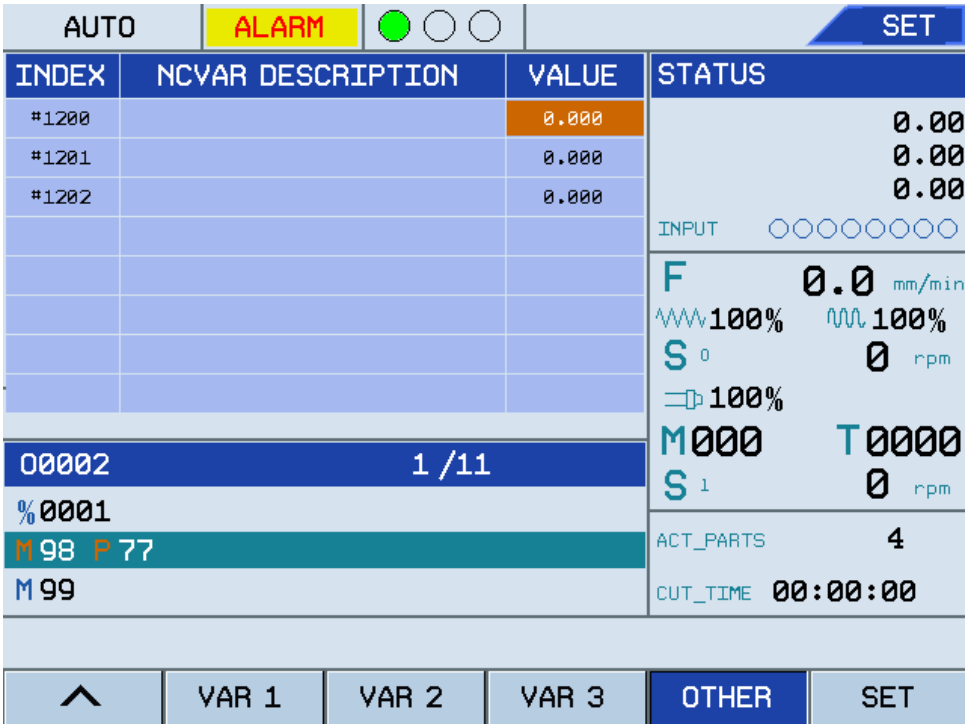


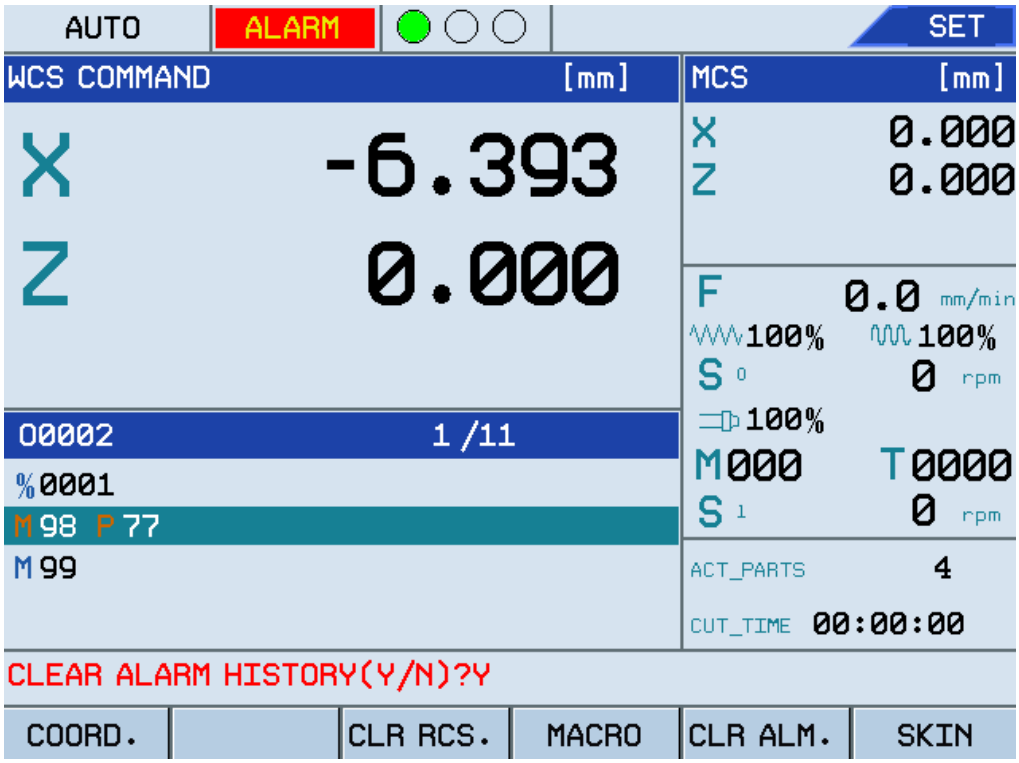
Figure 1.13 Other variables



4. CLR ALM

Press Set, and then press F5 to enter the alarm clearing screen. The system will prompt you whether to clear alarm history. See the figure below:

Figure 1.14 Clear alarms



5. SKIN

Press Set, and then press F6 to change the system background color. The system provides four background colors. See the figure below:

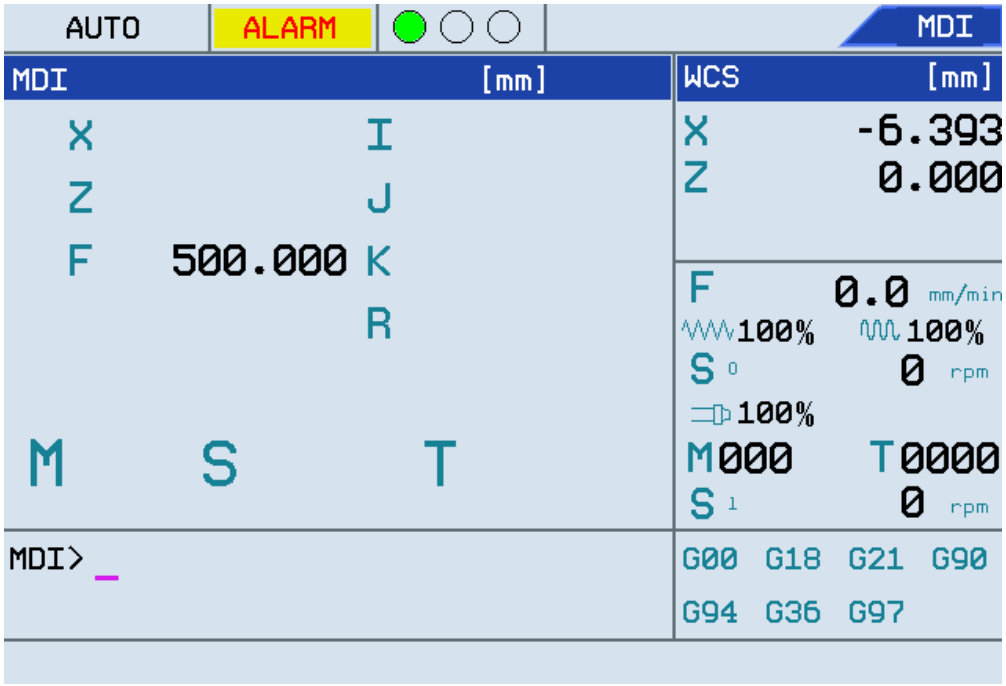
Figure 1.15 Background colors

1.3.3 MDI

Press MDI to enter the MDI screen which includes only the RETBP (F4) submenu.

See the figure below:

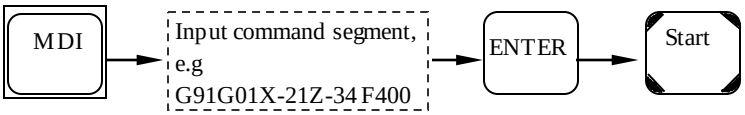
Figure 1.16 Figure MDI screen



Follow the instructions below to perform manual data input:

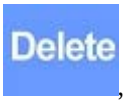
1. Input commands

Set the operation mode to Auto or Single Block mode:



The minimum unit for MDI is a valid command character. Therefore, there are two methods to input an MDI command segment:

- Input by once, that is, input multiple command characters once.
- Input by several times, that is, input only one character every time.

While entering commands, you may see the entered content in the command line. Before pressing the  key, you may press  ,  ,  , and  to edit as required if a wrong command is input.

For example, if you need to input the "G00 X100 Z1000" command segments, you may:

- Directly input "G00 X100 Z1000" and then press  . The values after G, X, Z on the screen change to **00**, **100**, and **1000** respectively.
- Input "G00" first, and then press  . The G code value displayed in the lower right corner displays as **G00**. Input "X100", press  , and then input "Z1000" and press  , "X100" and "Z1000" will be displayed on the screen successively.

2. Execute commands

After entering an MDI command segment, press the  key on the operation panel, and then the system will run the entered MDI command.

If the MDI command is incomplete or has wrong syntax, the system will display related error

messages, and the MDI command cannot be run.

3. Modify commands

Before running MDI command, you may directly input corresponding command characters and values in a command line if you need to modify the command character.

For example, after entering "**X100**" and pressing the Enter key, you may input "**X109**" in the command line and press the Enter key if you need to change the X-axis value to **109**.

4. Clear the current command

After inputting MDI data, you may press **F2** CLEAR to clear all currently input data (other commands are still valid), and all data after X, Z, I, K, R are cleared. You may enter new data as required.

5. Stop running commands

When the system running MDI commands, you may press **F1** STOP to stop running commands.

1.3.4 Offset

Press Oft to enter its corresponding screen which consists of DIA, LEN, XWEAR, ZWEAR, CLR X, CLR Z, X OFF, and Z OFF submenus.

1. Offset

Press the Oft key for the first time to enter the offset settings screen.

There are two methods to set the tool offset: to manually input and to use sample cutting. The system will automatically calculate the tool offset value based on the diameter and length. HNC turning system supports sample cutting to set tool offset values.

Note: The offset value will be reflected in the corresponding workpiece coordinate system. The X-axis origin point is in the centerline of the rotation axis.

The content below describes the method to set offset for tools:

This method requires independent offset value for each tool, which will be reflected in the workpiece coordinate system:

Figure 1.17 Edit tool offset

Follow the instructions below to set the tool offset:

a. Use the  ,  , PgUp, and PgDn keys to move the cursor to the row where you can select the required tool.

b. Do a round face grinding on the sample workpiece, and then exit the tool along with the Z axis (Do not move on X axis during this procedure).

c. Measure the diameter of the sample workpiece, and then manually enter the value for the measured diameter (DIA) as shown in Figure 1.17.

d. Do an end face grinding with the tool on the sample workpiece, and then exit the tool along with the X axis.

e. Measure the distance from the tool's end face to the point of origin in the workpiece coordinate system, and then manually enter the distance value for the Measured Length

(LEN) column.

Repeat the steps above to set the offset for other tools.

Note:

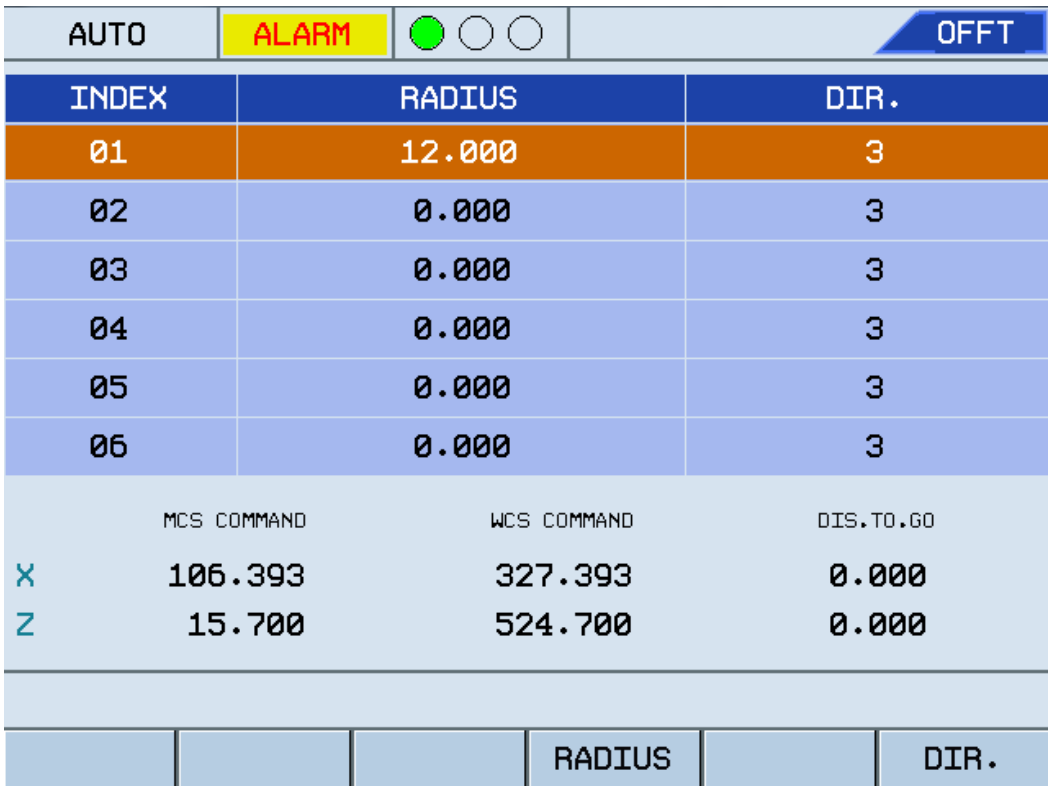
- You need to home the machine to the machine origin point before setting offset.
- The directional distance from the end face of the sample workpiece to the origin point of the workpiece coordinate system is the value on the Z axis in the workpiece coordinate system.
- Do not move the tool on the X axis after the round face grinding on the sample workpiece as the X-axis origin offset of the workpiece coordinate system = X-axis value of the machine coordinate system – measured diameter.
- Do not move the tool on the Z axis after the end face grinding on the sample workpiece as the Z-axis origin offset of the workpiece coordinate system = Z-axis value of the machine coordinate system – measured length.

2. Tool Compensation

Press the 

key to enter the screen as shown below:

Figure 1.18 Set tool compensation



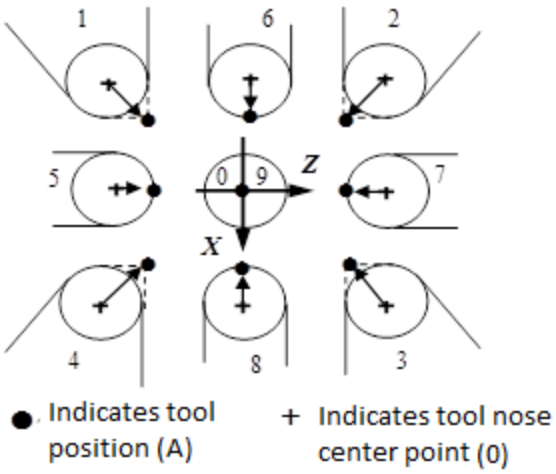
Use the ▲, ▼, PgUp, and PgDn keys to select required tools, and press the **F5** RADIUS and **F6** DIR. to enter the compensation value, press the Enter key to confirm the value.

Definition of Direction:

The tools of the machine can be installed in multiple directions and there are multiple modes of tool nose. To make the CNC system know the tool installation direction and make accurate tool nose radius compensation, we have defined the tool nose position codes.

The direction of the imaginary tool nose can be selected from the eight specifications shown below together with their corresponding codes.

Figure 1.19 Specification of the imaginary tool nose direction



Generally, most tool nose direction is specified as No.3 tool.

3. Tool wear

Use the ▲, ▼, PgUp, and PgDn keys to select required tools, press the **F5** XWEAR and **F6** ZWEAR to enter the tool wear value, and press the Enter key to confirm the value.

Figure 1.20 Set tool wear value

Note:You may specify the machine parameter **001020** to define whether to accumulate the tool wear values.

4. X/Z Offset

Press Oft, press **F6** >, select F5 X OFF or F6 Z OFF, enter the offset value in the input box, such as "16", and then press the Enter key to confirm the value. The system will display a message **"1: ALL 2: EXCEPT CUR_TOOL 3: ONLY CUR_TOOL 4: CANCEL"**, where:

- **1** indicates to perform offset for all tools including the current tool.
- **2** indicates to perform offset for all other tools except for the current tool.
- **3** indicates to perform offset for only the current tool.
- **4** indicates to cancel the offset.

Figure 1.21 Set X/Y offset

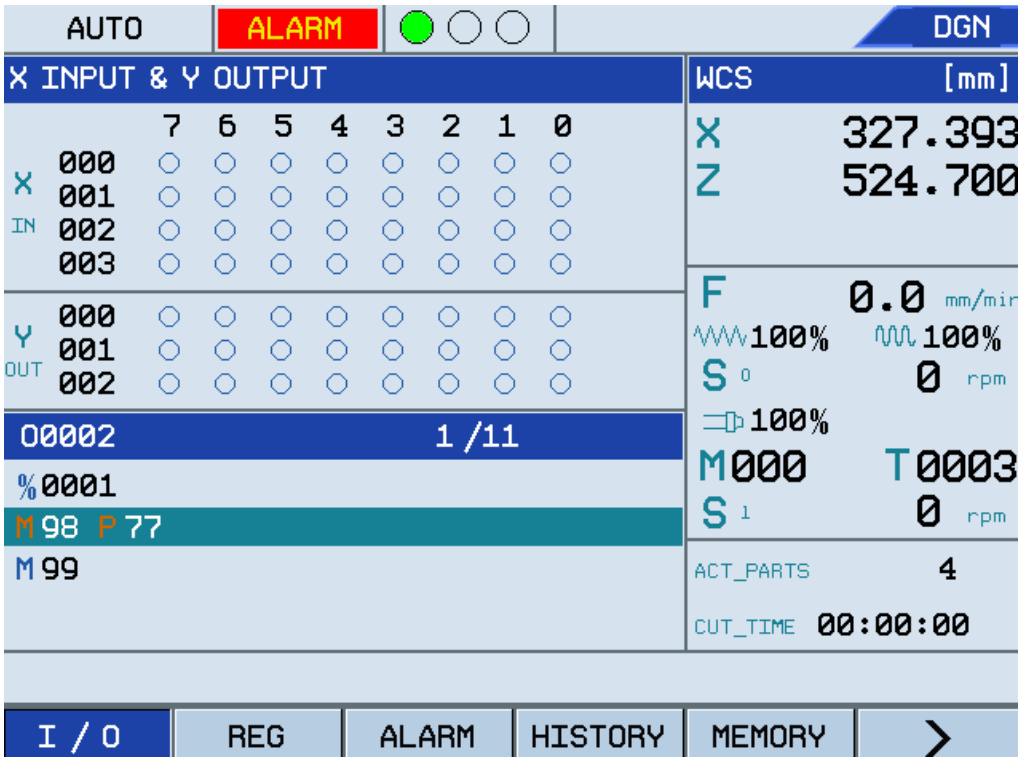
1.3.5 Tool Diagnosis

Press the Dgn key to enter the diagnosis screen which consists of six submenus: I / O, REG, ALARM, HISTORY, MEMORY, and >. You may press keys from F1 to F6 and enter corresponding submenus as required.

1. I/O

Press the Dgn key and then press F1 I / O to view the 128 groups of signals for each X Input and Y Output.

Figure 1.22 Check X input and Y output

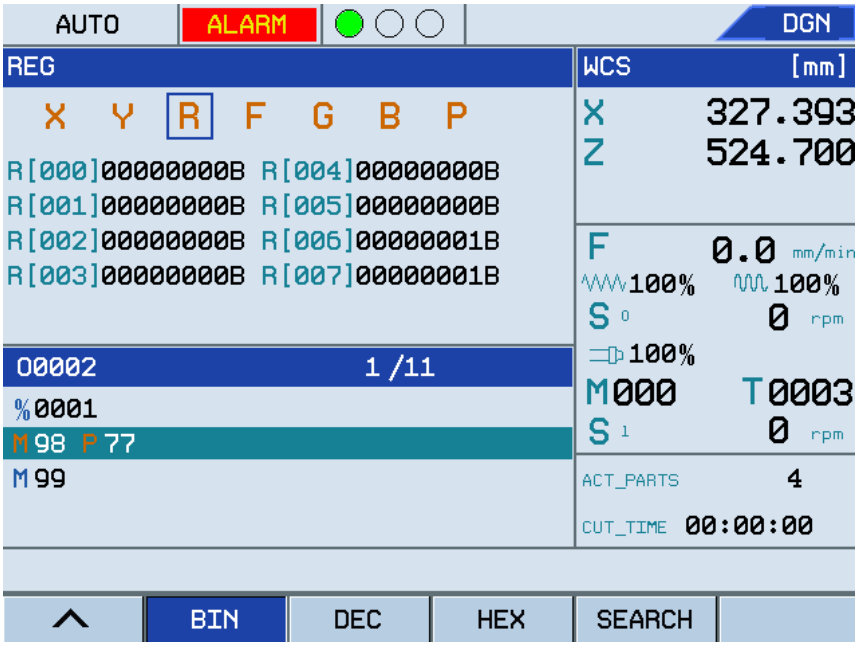


2. REG

Press the Dgn key and then press F2 REG to enter its corresponding submenu.

The system displays the X Register value by default. You may use the PgUp or PgDn key to view the values of other Registers. Press ◀ and ▶ to view the values of Register Y, R, F, G, B, P. Press F1 to locate quickly the Register as required. Press BIN, DEC and HEX to switch between different display modes.

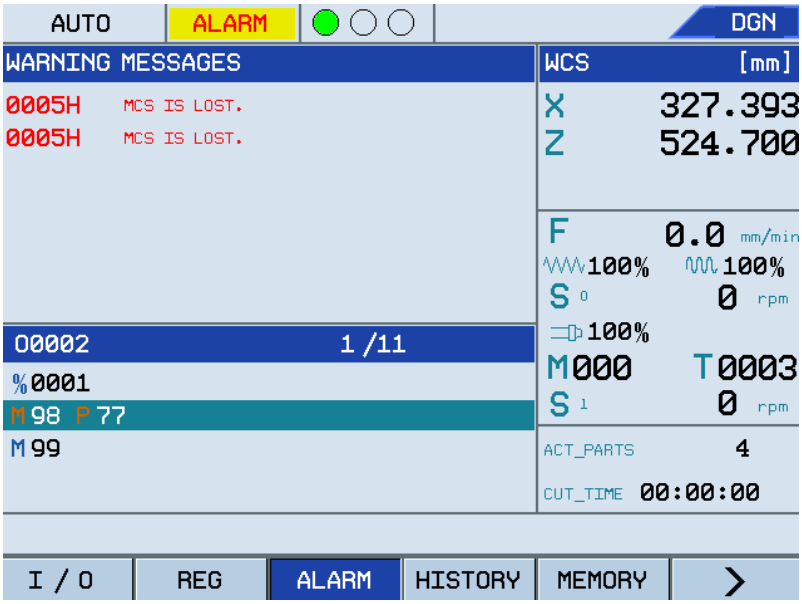
Figure 1.23 Check Register status



3. ALARM

If an error occurs during system running or processing, the message "ALARM" will be displayed on the screen, and the indicator blinks continuously. You may press the Dgn key to check the detailed warning messages. See the figure below:

Figure 1.24 Check warning messages

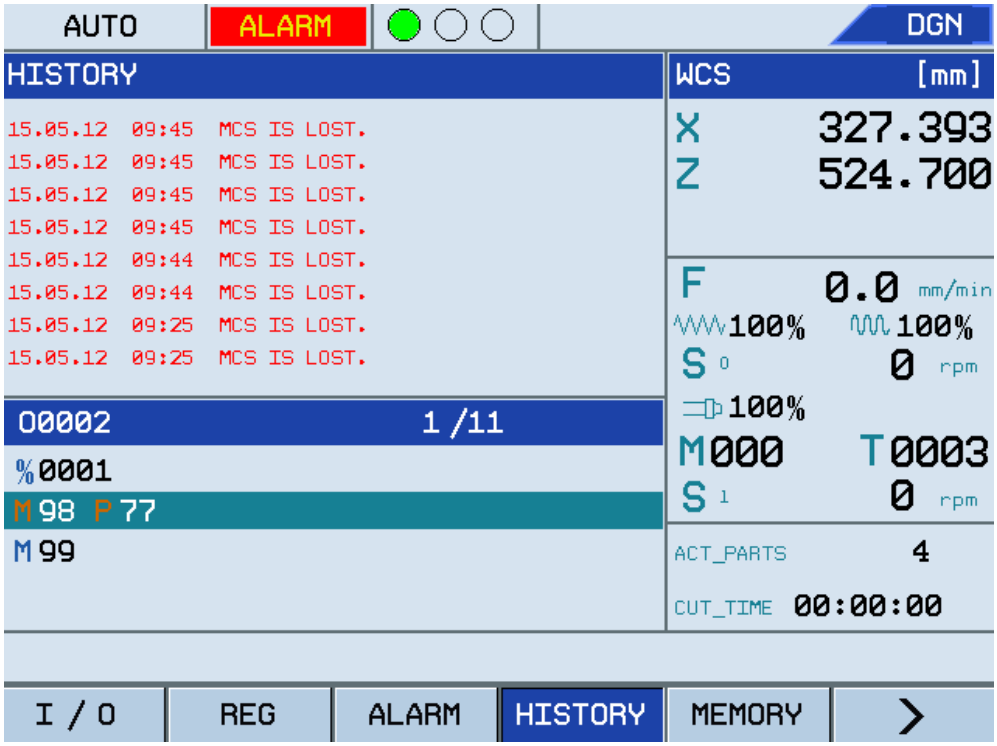


You may perform related operations to restore the system operation based on the displayed alarm information.

4. HISTORY

Press the Dgn key and then press F4 HISTORY to check the alarm history. Press PgUp or PgDn to page up or down as required. This is used to analyze system trouble history and perform related troubleshooting. See the figure below:

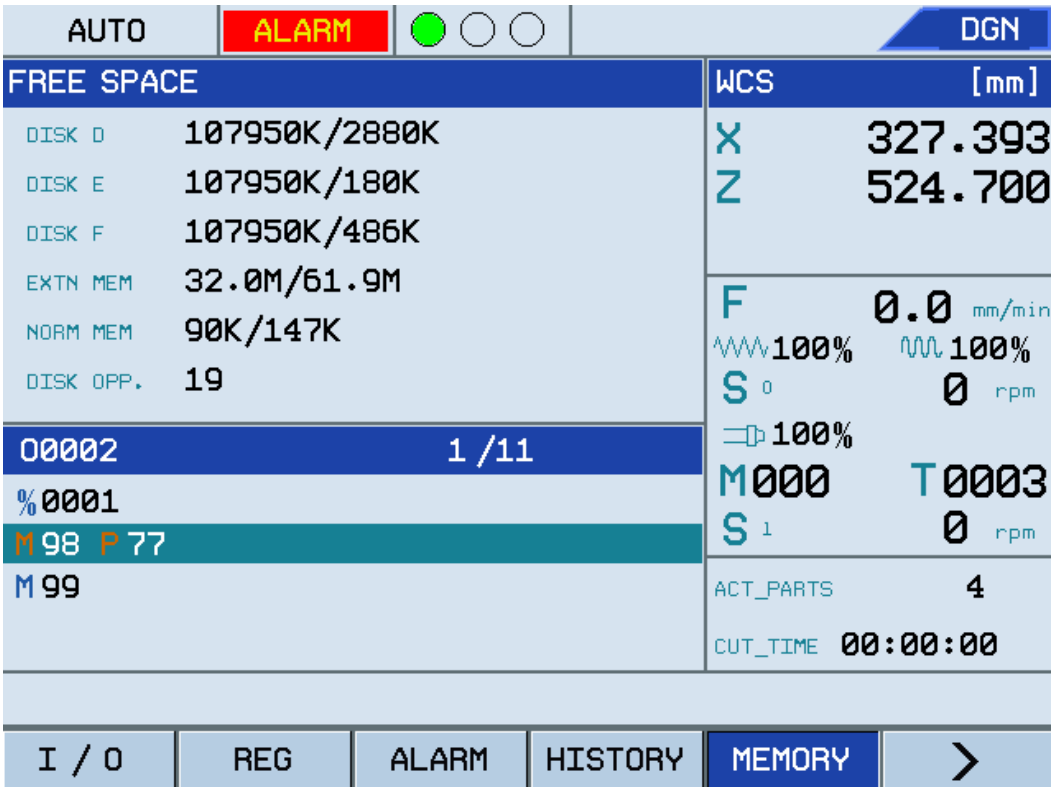
Figure 1.25 Check alarm history



5. MEMORY

Press the Dgn key and then F5 MEMORY to enter its corresponding screen to check the space and internal storage of each disk for the CNC system.

Figure 1.26 Space and internal storage of each disk



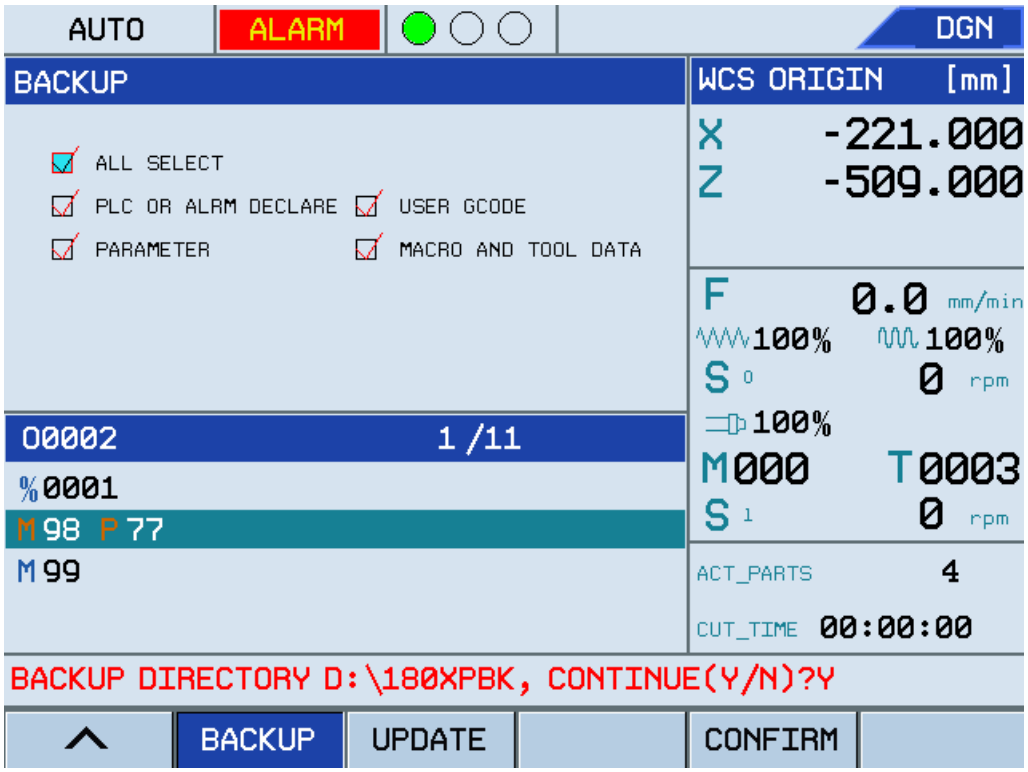
6. Extended

- Software Backup

Press the Dgn key, select F6 >, press **F2** BACKUP, use the ◀, ▶, ▲, and ▼ keys to select the file required to back up, and then press the Enter key to confirm the selection. Specify the file type, and then press **F5** CONFIRM.

A message as shown in the figure below is displayed:

Figure 1.27 Back up software



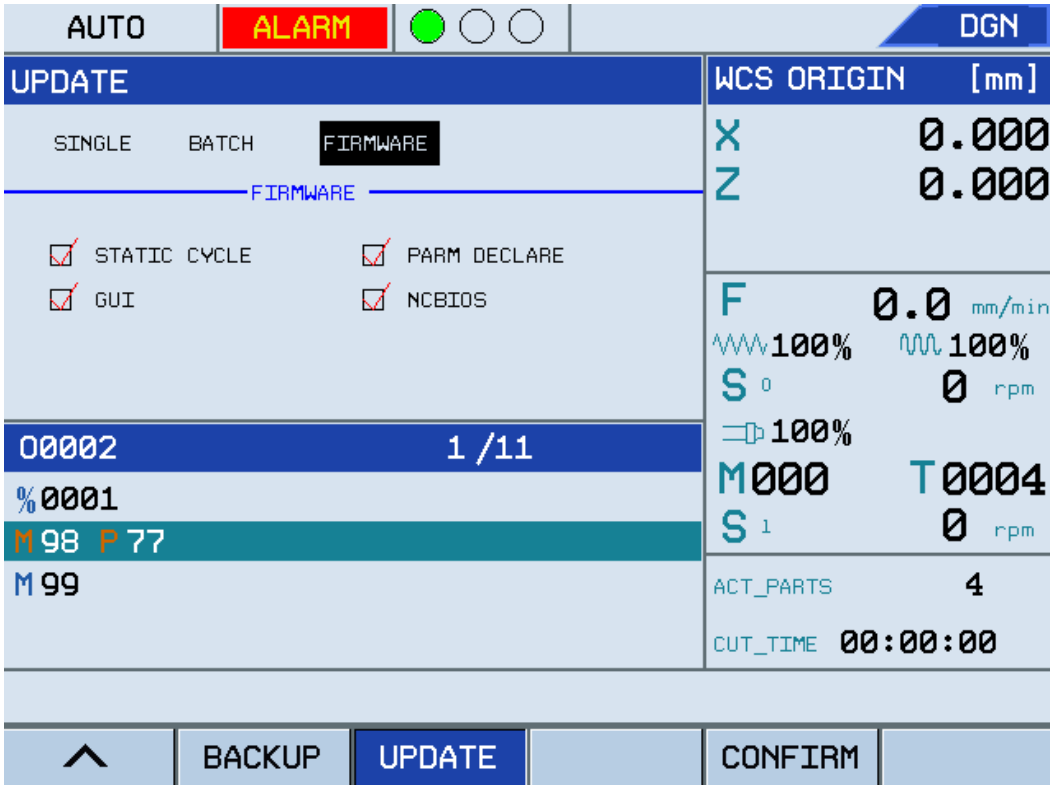
Note: The file backed up is saved in the **180XPBK** folder in the USB flash disk by default. If there is no external USB flask disk, the message "NO USB, UPDATE FUNCTION DISBALED!!!." is displayed during backup.

- Software update

Software upgrade includes three kinds of updates: SINGLE, BATCH, and FIRMWARE.

Press the Dgn key, select F6 >, and then press F3 UPDATE. Use the ◀ and ▶ keys to select "SINGLE", "BATCH", or "FIRWARE". See the figure below:

Figure 1.28 Software update



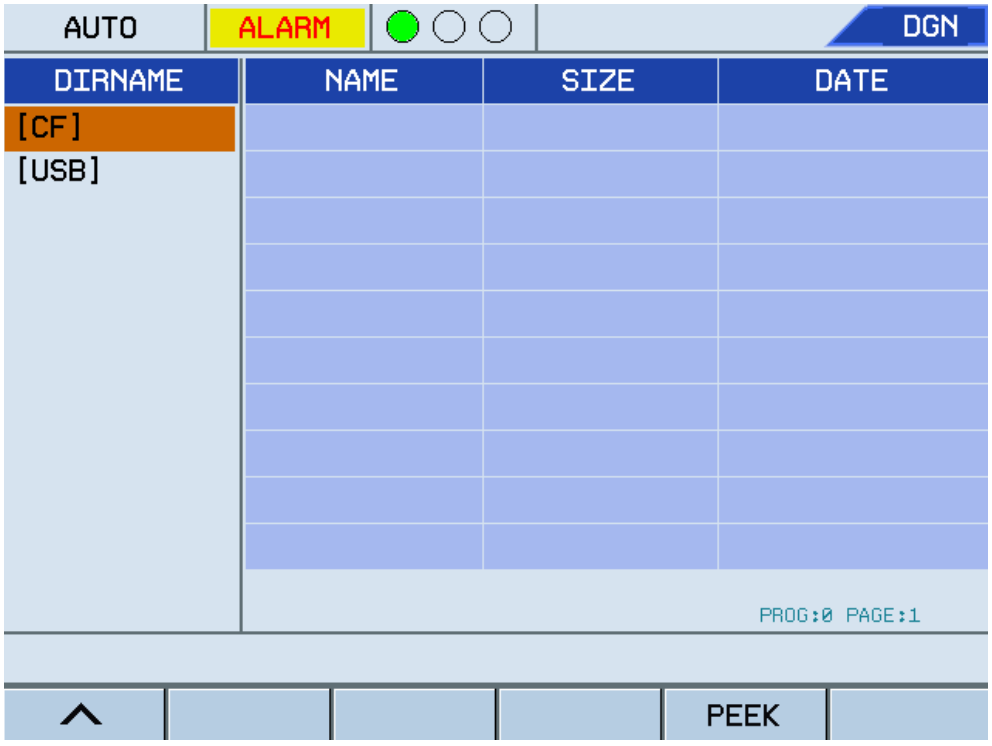
Note: If there is no external USB flask disk, the message "NO USB, UPDATE FUNCTION DISBALED!!!." is displayed during update. Do not power off during update, otherwise the system cannot start.

- Firmware update:

Firmware update intends to update the drive program, upper-layer software, lower-layer software, and configuration file at one time. Firmware update is generally used to update the newly released software.

To update firmware, press the Dgn key, select **F6** >, press **F3** UPDATE, and then use the ◀ and ▶ keys to select **FIRMWARE**. Make sure that the four files of Firmware are all selected (see Figure 1.28), and then press **F5** CONFIRM to select the file for update. See the figure below:

Figure 1.29 Select update file



Use the  ,  ,  , and  keys to select required directory and files, press the  key to confirm the selection. The system then starts the update. After the update, the system will automatically restart.

Note: Do not power off during the update, otherwise the system cannot start.

- SINGLE update

SINGLE update intends to independently update the upper-layer software, lower-layer software, PLC, parameters, and fixed circular files with a single file. This is very convenient for users to update the software, and thereby avoiding the efforts on update package development. **SINGLE** update is generally used for update after critical errors are fixed or users modify PLC or parameters.

To perform **SINGLE** update, press the  key, select **F6**  , press **F3**

UPDATE, use the ◀ and ▶ keys to select **SINGLE**, and then press the ▲ and ▼ keys to select a single file to update. Press the Enter key and select the checkbox in front of the file, and then press **F5** CONFIRM. See the figure below:

Figure 1.30 SINGLE update

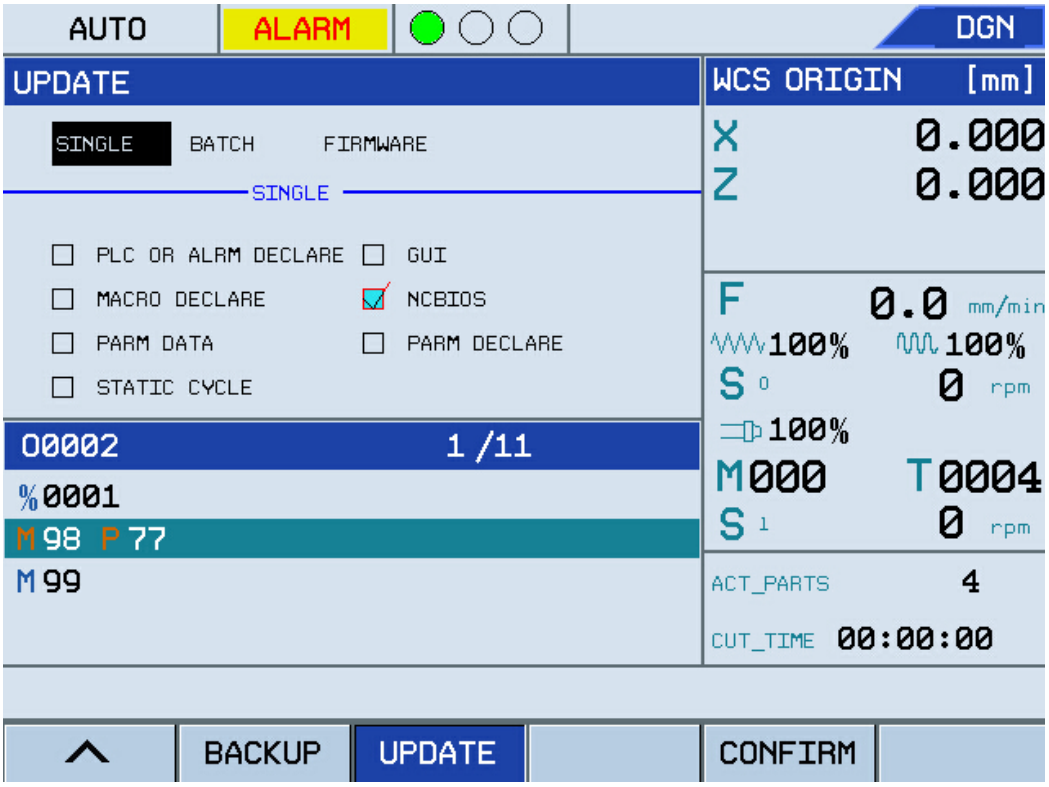
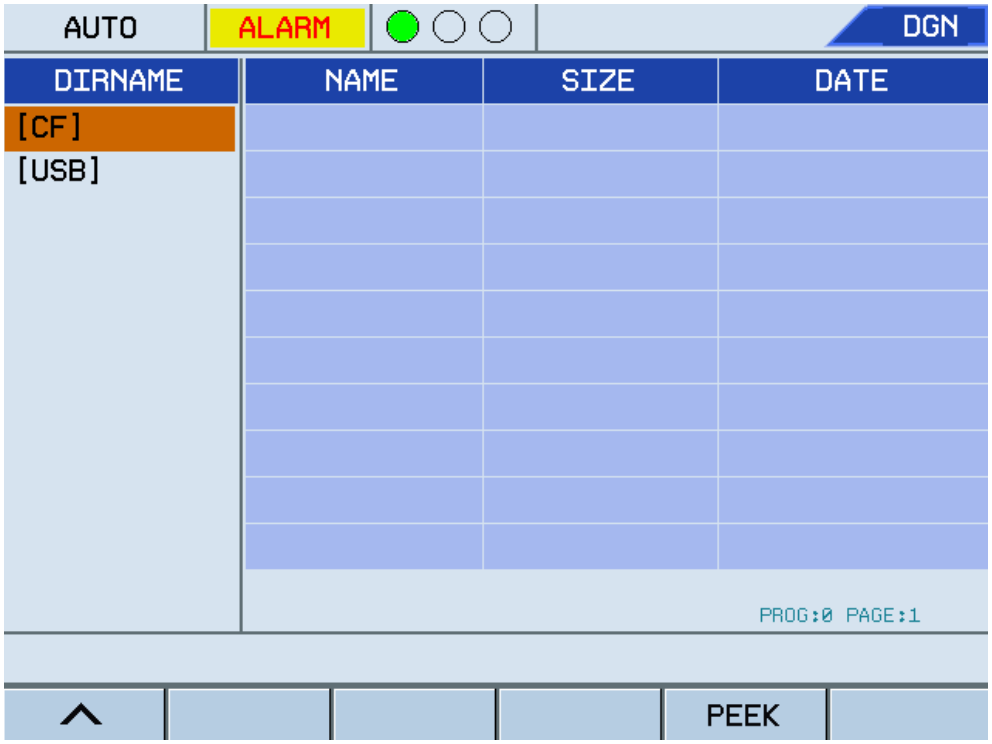


Figure 1.31 Select an update file



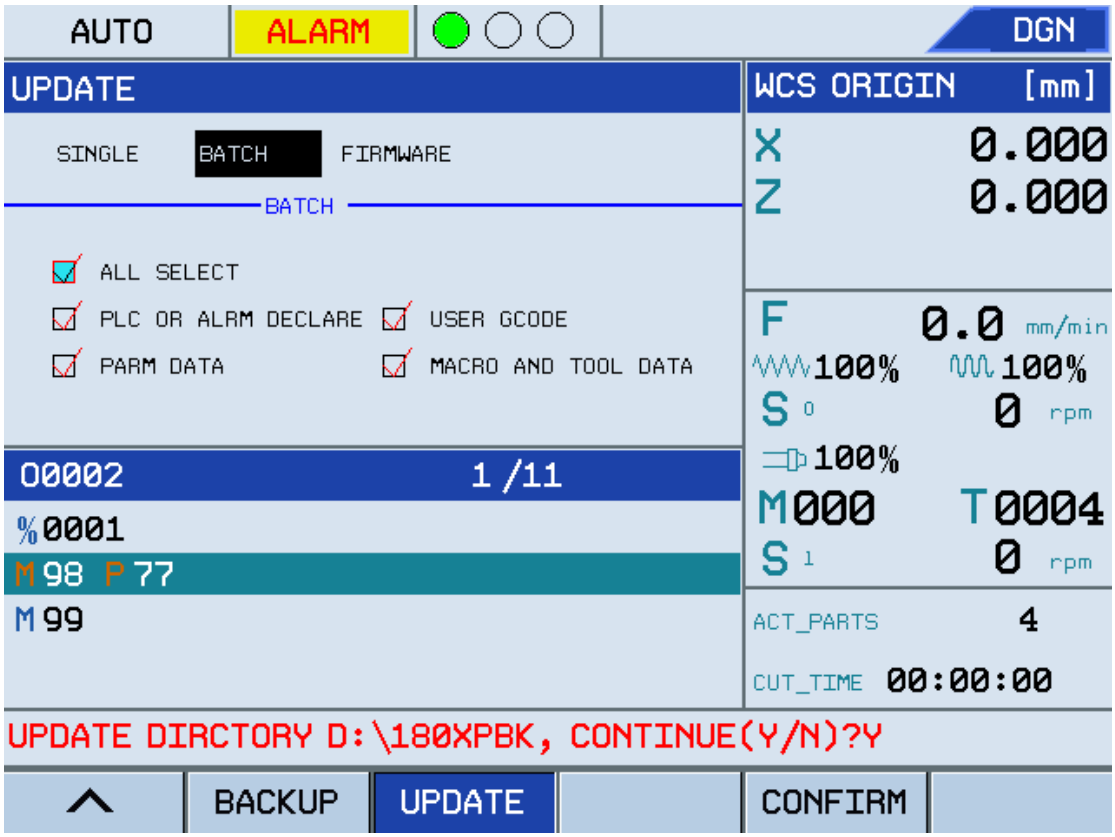
Note: Do not power off during the update, otherwise the system cannot start.

- BATCH update

BATCH update intends to update PLC, parameters, G code and tool compensation, which is convenient for users to commission machines of the same type in batches. (This is generally used for the update of batch commissioning.)

To perform **BATCH** update, press the Dgn key, select **F6** >, press **F3** UPDATE, use the ◀ and ▶ keys to select **BATCH**, and then press the ▲ and ▼ keys to select files to upgrade. Press the Enter key to confirm the selection, select the checkbox in front of the files, and then press **F5** CONFIRM. The system will prompt users the information about the batch file update directory. See the figure below:

Figure 1.32 Batch file update



Note: The files for batch-file update is saved in the **180XPBK** folder in the USB flash disk by default. Do not power off during update, otherwise the system cannot start.

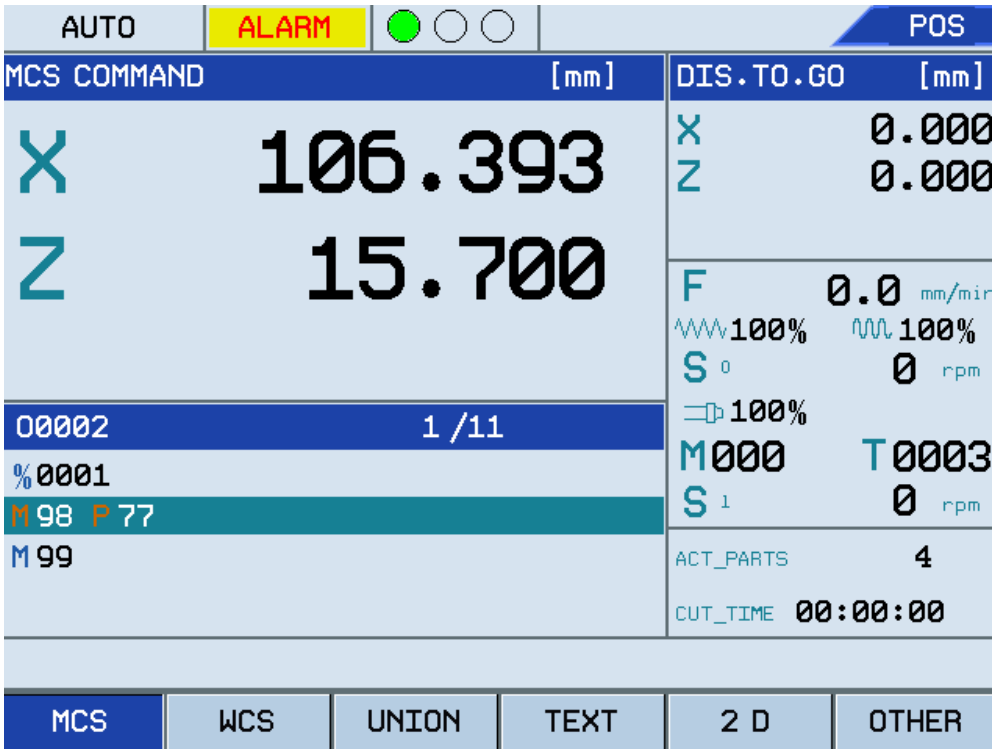
1.3.6 Position

Press the Pos key to enter the position screen, which consist of six submenus: MCS, WCS, UNION, TEXT, 2 D, and OTHER. You may press keys from F1 to F6 to enter corresponding submenus as required.

1. Machine Coordinate System (MCS)

During program running, press Pos, and then press the **F1** MCS key to check the position of the current tool in the machine coordinate system. See the figure below:

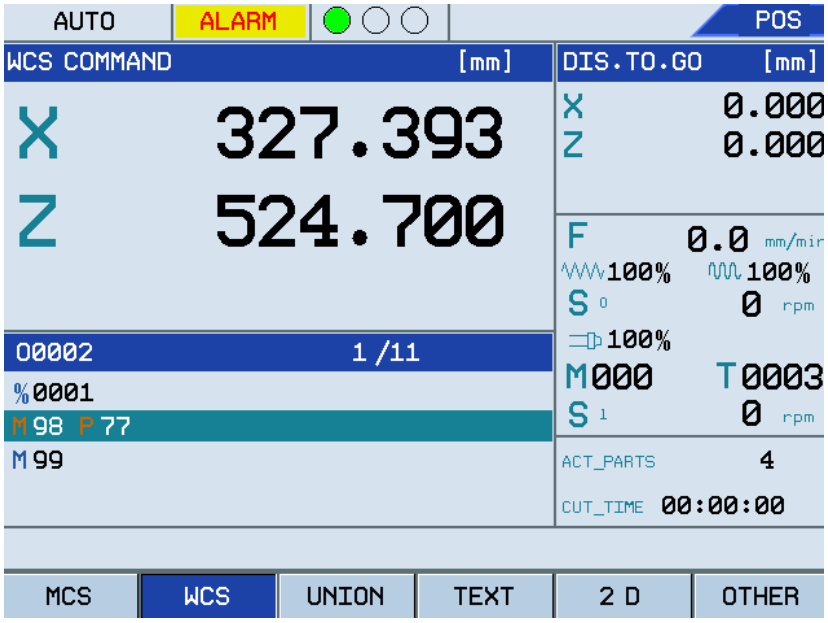
Figure 1.33 Command values in the MCS



2. Workpiece Coordinate System (WCS)

During program running, press Pos, and then press the **F2** WCS key to check the position of the current tool in the workpiece coordinate system. See the figure below:

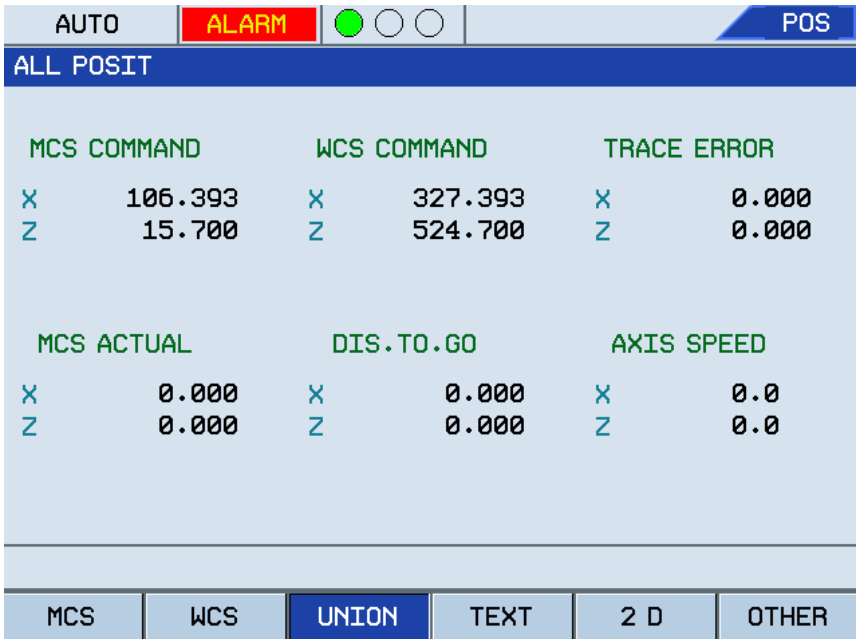
Figure 1.34 Command values in the WCS



3. Union

During program running, press Pos, and then press the **F3** UNION key to check the position of the current tool and "DIS. TO. GO." in the machine, workpiece, and relative coordinate systems. See the figure below:

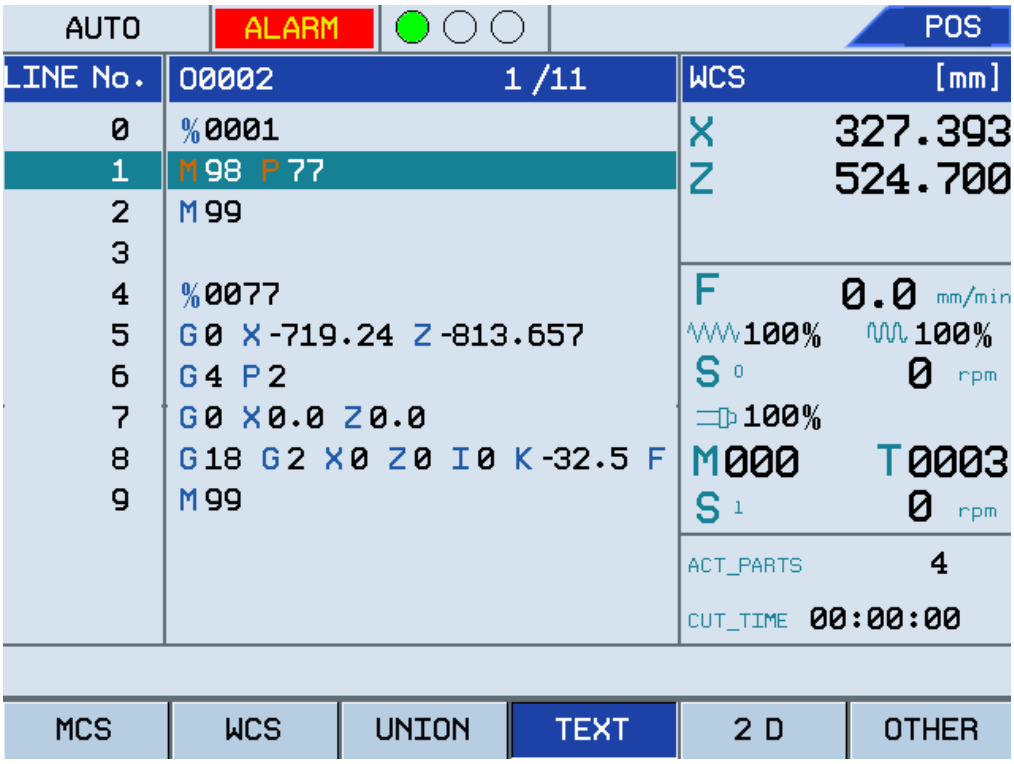
Figure 1.35 Axis values in multiple coordinate systems



4. TEXT

During program running, press Pos, and then press the **F4** TEXT key to check the G code, workpiece command, Line No., M command, Feed rate, etc. See the figure below:

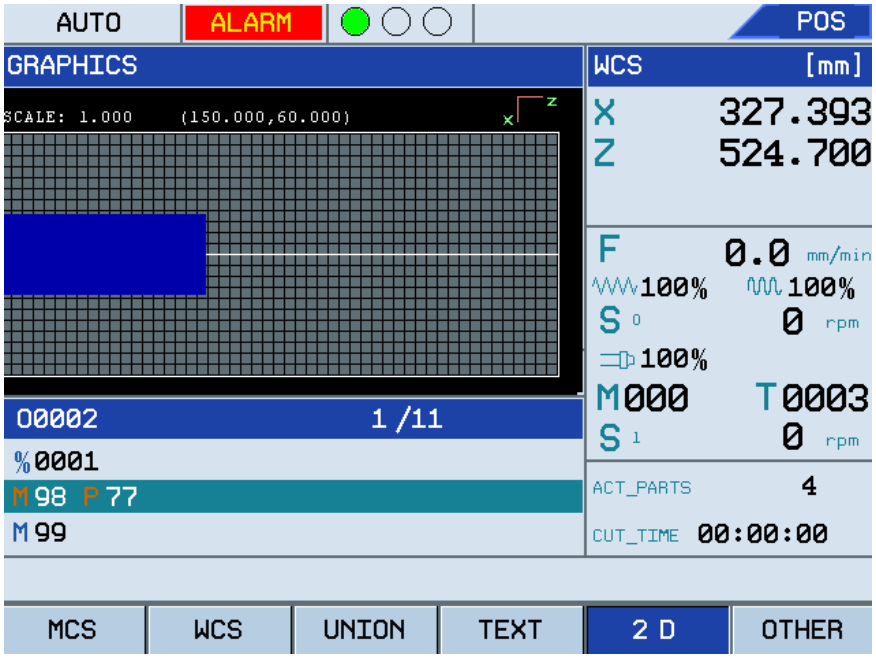
Figure 1.36 Text display mode



5. 2D (Graph)

During program running, press Pos, and then press the **F5** 2 D key to check the real-time processing route. Use the ◀, ▶, ▲, and ▼ keys to change the display graph of the sample workpiece. See the figure below:

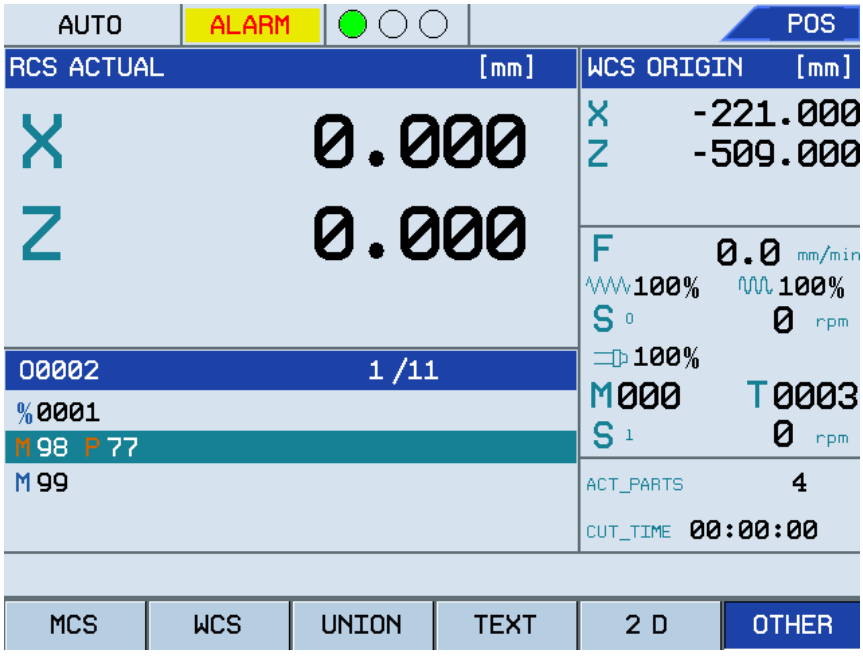
Figure 1.37 Graph display mode



6. OTHER

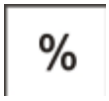
During program running, press Pos, and then press the **F3** OTHER key to check the position of the current tool and the distance to go in the machine, workpiece, and relative coordinate systems. See the figure below:

Figure 1.38 Other information



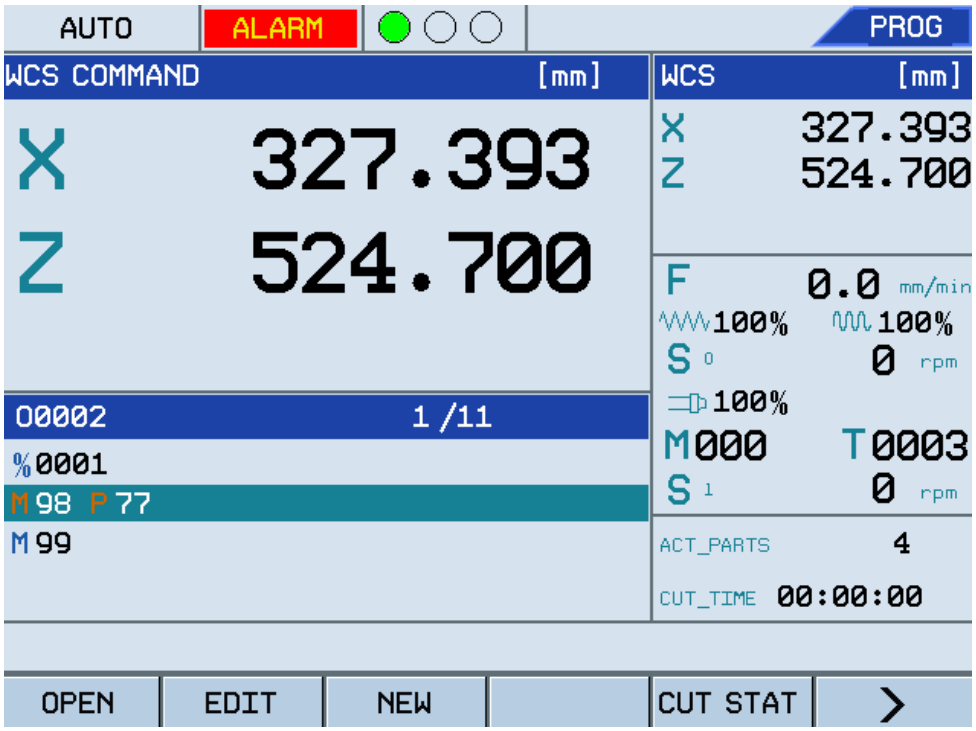
Auxiliary Content

The auxiliary content are displayed in the upper-right corner. By default, the information about "DIS. TO. GO." is displayed. The content in this area depends on the current menu, and can be switched by pressing relevant keys.

Press the 

key to change the auxiliary content displayed in the right corner. You may switch between DIS. TO. GO, WCS COMMAND, WCS ORIGIN, and MCS ACTUAL. The figure below shows that the auxiliary content has changed from DIS. TO. GO to WCS COMMAND.

Figure 1.39 Switch auxiliary content



1.3.7 Parameters

Press the 

key to enter the parameters screen. Use the  and  keys to select the parameter type on the left, and then use the  and  keys to switch between parameter type and parameter value.

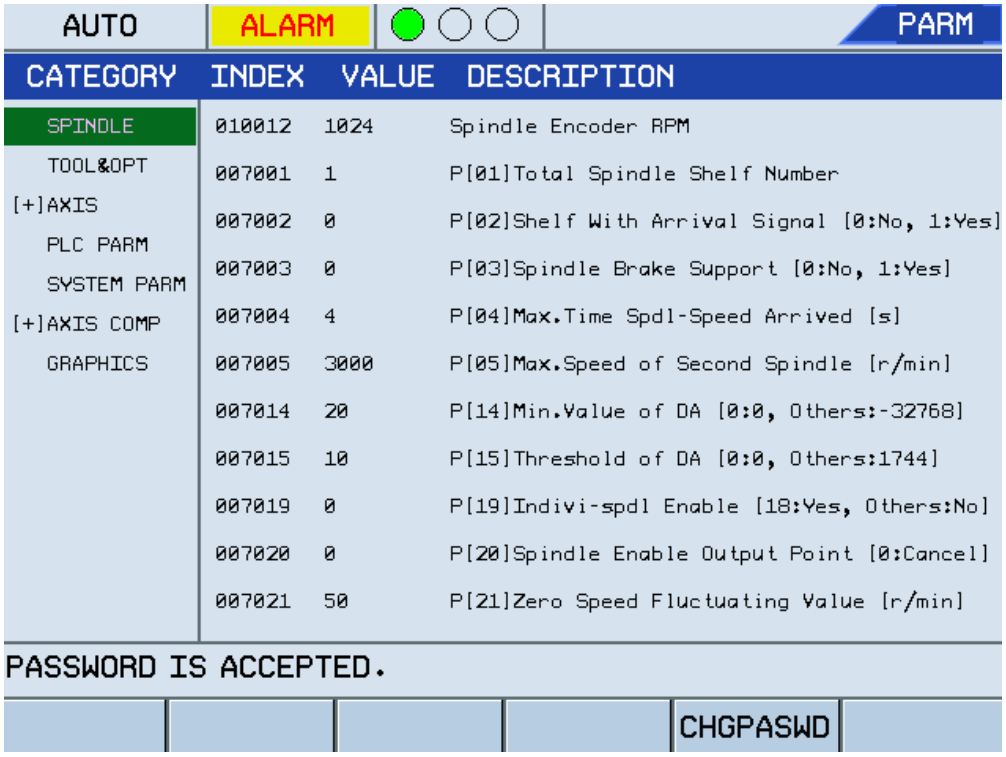
1. Edit parameters

You need to enter the password for editing parameters to avoid random system parameter modification, which may cause damage. Follow the instructions below to edit parameters:

a. Press the Par key and **F1** PASSWORD, enter the required password, and then press the Enter key for confirmation.

- If the password is correct, the message "PASSWORD IS ACCEPTED" as shown in the figure below is displayed.
- If the password is incorrect, you cannot modify the parameters.

Figure 1.40 Edit system parameters



b. Use the ▲ and ▼ keys to select the parameter type to edit.

c. Press the ▶ key to select the parameter to edit.

d. Press the Enter key to enter the editing state, and then press the BS key to clear the previous value.

e. Enter a new value, and press the Enter key for confirmation.

f. Press the PgUp or PgDn key to edit other parameters.

g. After editing all required parameters, press **F4** SAVE to save the modified information and repower on the system.

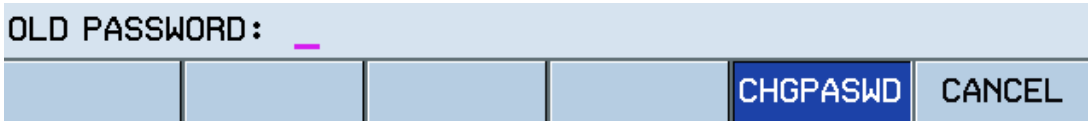
2. Change parameter editing password

Follow the instructions below to change the parameter editing password:

a. Press the Par key, and then press **F5** CHGPASWD.

The system prompts you to enter the old password. See the figure below:

Figure 1.41 Change parameter editing password



b. Enter the old password, and press the Enter key for confirmation.

c. If the password is correct, the system prompts you to enter a new password. Press the Enter key to confirm the new password.

d. Enter the new password once again and press the Enter key for confirmation.

- If the two new passwords entered are the same, then the password is changed successfully.
- If the new passwords you entered are different, the system prompts you that the new passwords are not the same and the password modification failed.

2 Power on/off and Safety Protection

This section describes the operations on power on/off, emergency stop, reset, reference home, hardware and software limits, Over-travel protection, etc.

2.1 Power on

Follow the instructions below to power on the machine:

1. Check and make sure the machine is normal to run.
2. Make sure the power supply and voltage to meet requirements and the wiring is correct and tight.
3. Power on the machine (for more details, see the user manual of the machine).
4. Power on the CNC system.
5. Make sure the indicators on the machine control panel are normal.

After powering on the CNC system, the system automatically runs system software and enter the Programming main menu.

Figure 1.2 shows the software operation interface after power on, and the operation mode is Auto.

2.2 Power off

Follow the instructions below before power off:

1. Make sure the X and Z axis of the CNC system is in the stop status.
2. Auxiliary functions (such as main spindle, and pumps) are disabled.
3. Cut off the power for the servo motor, then for the CNC system, and lastly for the machine.

2.3 Home to Reference Point

The machine coordinate system is the precondition of machine operation. Therefore, homing to the reference point is required before powering on or resetting the machine. Follow the instructions below:

1. Switch the mode to the Reference mode by pressing the Reference key on the control panel.
2. Press "+X" or "+Z" to move the machine along with the selected axis. After both axis homering to the reference point, the machine coordinate system is established.

Note:

1. After power on, each axis must be homed to the reference point, and then other operations

can be performed. If the floating origin function has been enabled, the operation of homing is not required. For more details, see section 1.3.2.

2. Press "+X" and "+Z" simultaneously to home both X axis and Z axis to the reference point at the same time.
3. After homing each axis to the reference point, you do not need to home the axis to the reference point during operation unless a servo driver alarm is reported.

2.4 Over-Travel Protection

2.4.1 Hardware Over-Travel Protection

Each axis has a hardware limit and a software limit at each end of the axis. If a hardware switch is encountered on an axis, the hardware over travel occurs (the running status displayed on the screen is "Emergency stop" with the indicator blinking continuously), and the system will perform emergency stop accordingly based on the actual situation.

The tool can be retrieved from an over-travel status by following the operations below:

1. If no over-travel release key is available, press the 

key to reset the system.
2. If an over-travel release key is available, press the over-travel release key, and then press the 

key until the system is reset.
3. In the Manual mode, move the tool backward on the axis to be out of over-travel.
4. Release the over-travel release key.

If the running status displayed on the screen changes from "Emergency stop" to "Normal", the system restores and may proceed with operations.

Note: Please pay attention to the motion direction and speed when moving the tool out of over-travel to avoid tool collision with the machine.

2.4.2 Software Over-Travel Protection

Positive and negative software limits are defined for each axis using feed parameters, which takes effect after homing to reference point. When the axis is commanded to go beyond these limits (over-travel), the software switch will be encountered, corresponding alarms will be reported, and

the motion ceases and no further motion in this direction is permitted. To clear the alarm, press the

key, and move the tool backward on the axis.

2.5 Emergency Operation

2.5.1 Reset

If a CNC system error or axis error occurs, you may press the Reset key to reset the system. In this way, all axis stop motion and all auxiliary functions unavailable; the machine stops running, and the system is in the initiation status; All system alarms are cleared, and the processing program is reset.

2.5.2 Emergency Stop

During machine operation, you may press the  key in case of emergency. Once the Emergency Stop button is pressed, the system enters into the "Emergency Stop" mode, and the servo and spindle are disabled (the feed driver power supply is cut off in the control cabinet). Release the Emergency Stop button or twist the ES button clock-wise, then the system enter the reset status.

Make sure that the trouble has been solved before releasing the Emergency Stop button, and home the axis to the reference point again after releasing the Emergency Stop button to ensure correct coordinate position (if the machine origin point has not been set, you need to set the floating origin point).

For information about circuit connection, see HNC-180 Series Installation and Connation Guide.

Note: Press the Emergency Stop button before turning the power on or off to avoid device crash.

2.5.3 Feed Hold

The "Feed Hold" button is used to pause the ongoing machining operation. This operation cannot stop the machine immediately during thread cutting and cycle command running.

2.5.4 Power off

During machine operation, you may cut the power supply off in case of emergency to avoid

accidents. However, the coordinate position displayed after re-powering on may be different from the actual position, and therefore you need to perform tool settings again.

3 Manual Operation

3.1.1 Manual Feed

Press the  key (the indicator is lit) to enter the Manual mode. In this mode, you may move the cutting tool along a motion axis with jog feed. The following information describes the motion along the X axis:

1. Press the "+X" or "–X" key (the corresponding indicator is lit) to move the X axis continuously in the positive of negative direction.
2. Release the "+X" or "–X" key (the corresponding indicator is off) to decrease and stop the motion along the X axis.

Perform similar operation to move the Z axis continuously in the positive or negative direction. In the Manual mode, press the X and Z axis keys simultaneously to manually control the X and Z axis motion at the same time.

3.1.2 Rapid Traverse Feed Rate

In the manual feed mode, press the  key to move the tool rapidly in the positive or negative direction on relevant axis.

3.1.3 Manual Feed Rate Adjustment

Use the  button to adjust the feed rate with the manual feed "override ratio", which is between 0% to 120%.

The default spindle feed rate after power on is 100%. Press the minus "-" key to decrease the feed rate by 10%. The minimum value is 10%. Press the plus "+" key to increases the feed rate by 10%. The maximum value is 150%.

The default rapid traverse federate after power on is 20%. Press the minus "-" key to decrease the feed rate by 10%. The minimum value is 10%. Press the plus "+" key to increases the feed rate by

10%. The maximum value is 100%.

3.1.4 Hand-wheel Feed

Press the Hand wheel key (the indicator is lit) to enter the hand wheel mode. Rotate the feed rate adjustment button to set the feed rate for the cutting tool. Clockwise rotation moves the selected axis in the positive axis direction. Counter-clockwise rotation moves the selected axis in the negative axis direction. The table below describes the relationship between the increment of hand wheel motion and the amount of magnification (X1, X10, X100).

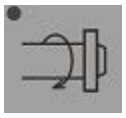
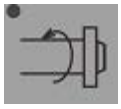
Magnification	x 1	x 10	x 100
Increment (mm)	0.001	0.01	0.1

The rotation direction determines the feed direction. Generally, Clockwise rotation moves the selected axis in the positive direction. Counter-clockwise rotation moves the selected axis in the negative direction. If the feed rate direction is opposite to this, you may switch the A and B signals on the hand wheel.

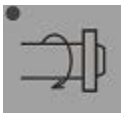
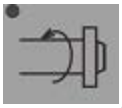
3.2 Other Manual Operation

3.2.1 Clockwise and Counter Clockwise Rotation of Spindle

In the manual mode, if the machine is powered on for the first time, follow the instructions below:

1. Press the  key to rotate the spindle in the clockwise direction at the specified speed. The LED indicator is lit.
2. Press the  key to rotate the spindle in the counter-clockwise direction at the specified speed. The LED indicator is lit.
3. Press the  key to stop the spindle manually. The LED indicator is lit.

If you manually rotate the spindle after auto running, the spindle rotation speed will maintain that in the auto running.

Note: If you press one of the  ,  , and  keys (the indicator is lit), the other keys will be invalid (the indicators turn off).

3.2.2 Spindle Jog

In the manual mode, press and hold the Spindle Jog key (the indicator is lit), the spindle will rotate at the speed specifying by machine parameters. Release the key (the indicator turns off)d, the spindle will stop running.

3.2.3 Coolant Control

In the Manual mode, press this key to start or stop the coolant flow. Coolant function is enabled if the LED light is lit and disabled if the LED light is off.

3.2.4 Lubrication Control

1. Manual-Lubrication

In the manual operation mode, press this key to enable or disable the lubrication function.

2. Auto-Lubrication

You may define the lubrication duration by specifying PLC parameters **007054** and **007055**. The system will start lubrication for a period specified by **007055** after power on, and then stop lubrication for a period specified by **007054**. This procedure will be repeated.

3.2.5 Turret Rotation

In the manual mode, press the  key to rotate the turret to the next tool number in the clockwise direction. The number of the tool where the turret will rotate to depends on the times you press the key. For example, if the current tool number is **1**, and you may continuously press the key for three times to rotate the turret to the fourth tool.

4 Automatic Operation

4.1 Auto-Operation

This section describes the operation related to automatic operation.

4.1.1 Enable Auto-Operation

After a program file is loaded and verified, you may enable the auto operation function.

1. Press the  key (the indicator is lit) on the operation panel to enter the auto operation mode.
2. Press the  key (the indicator is lit) on the operation panel to automatically run the loaded program.

4.1.2 Pause Auto-Operation

During program operation, you may follow one of the methods below to pause the on-going operation:

1. During program operation, press the  key (the indicator is lit) to enter the Feed Hold mode and pause the on-going operation, and then press the  key to proceed with the auto operation.
2. Use the **M00** command to pause the on-going auto operation, and then press the  key to proceed with the auto operation.
3. Use the **G04** command to pause the on-going auto operation. The pause duration is specified by the **G04** parameter. Press the  key to proceed with the operation.

4.1.3 Stop the Auto-Operation

During auto operation, you may follow one of the methods below to stop the auto operation:

1. Press the  key to stop the operation of all axis. In this case, the M code and S code

functions are unavailable, and the system terminates the auto operation, G code function and status maintaining function.

2. Press the  key to stop the operation in case of emergency (only when the external ES signal is valid). In this case, the CNC system enters the Emergency Stop mode, the motion is stopped, and all other operations such as spindle rotation and coolant flow are stopped. Release the key to clear the alarm and enter the Reset mode.
3. Press the 

key, press the F2 

key, and then press the F4 

key. The system prompts you whether to stop the program. Press **Y** or press the **Enter** key to stop the program.

4.2 Start from a Radom Line

When the auto operation is paused, the program can be restarted from the location where it is paused or from a random line. Press the  key, select a program, press F6 

under 

, and then press F2 

to start the operation as required.

4.2.1 Start from the Red Line

Follow the instructions below to restart the operation from the red line

1. Press the  key to enter the feed hold mode (the indicator is lit).
2. Use the 

and 

keys to move the blue bar to the desired start line, and at this time the blue bar turn red.
3. Press F2 

, and select the row from which to start the operation (the red bar turn blue).
4. Press the  key to start the operation from the blue bar (that is, the red line).

4.2.2 Start from the Specified Line

Follow the instructions below to start the operation from the specified line:

1. Press the Feed Hold key on the operation panel to enter the feed hold mode (the indicator is lit).
2. Press F3 

. The system prompts you to enter the line number from which the operation starts.
3. Enter the line number and press the 

key to confirm.
4. Press the  key to start the operation from the specified line.

4.3 Single Block Operation

Press the  key to enter the Single Block mode. The program will be executed block by block.

1. Press the  key to execute a block of the program, and then the motion speed on the axis will decrease to stop, and the tool stops.
2. Press the  key again to execute the next block of the program, and after the execution is completed, the operation will stop.

4.4 Over-Block Operation

If you need neither to execute a block of program nor to delete it, you may enable the over-block operation function. Press the  key on the operation panel, and the system will automatically skip the block of the program beginning with the "/" symbol. Release the key to disable the over-block operation.

Note: If the over-block operation function is disabled, the system will not skip the program beginning with the "/" symbol.

4.5 Breakpoint Related Operation

During processing, there are always some large workpieces which requires complicated operation and long duration. It is necessary to save and restore the breakpoint for further processing. In other words, you may save the breakpoint (let the system remember various status of the time point) and cut the power supply. After a period of time, you may turn on the power supply and restore the breakpoint (let the system restore the status when the processing is broken) to proceed with the processing from the breakpoint.

4.5.1 Save the Breakpoint

Follow the instructions below to save the breakpoint:

1. Press the  key to enter the feed hold mode (the indicator is lit). This operation can be performed only when the program is in the Auto operation mode.
2. Press the Prg key. Press F6 >, press F6 BRK.PT., and then press F4 SAVE. The system will prompt you to enter the file name for the breakpoint. See the figure below:

Figure 4.1 Save breakpoint file

3. Press Enter, and the system will prompt you to select a storage path with "Save to: 1 CF card; 2 USB; 3 Cancel". You may select 1 or 2 as required. The system will automatically create a file named with the current program name and suffixed with **BP1** in the root directory of the selected storage card. You may modify the name without a suffix.

Note: **It's not allowed to save breakpoint during program running**.

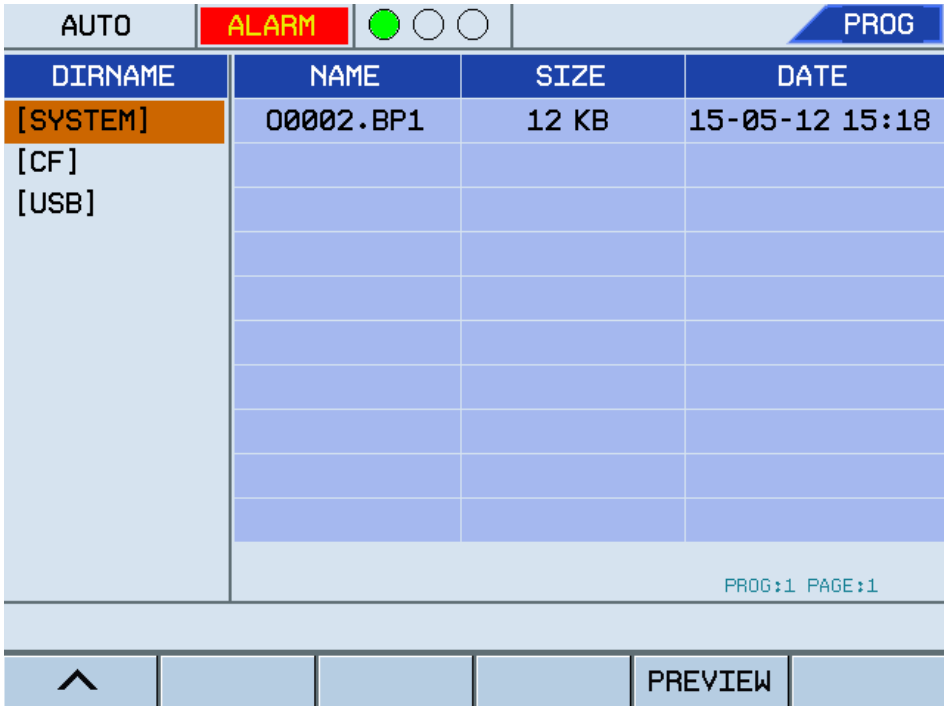
4.5.2 Resume Breakpoint

Follow the instructions below to resume a breakpoint:

1. If the power is off after you save the breakpoint, you need to home the tool back to the reference point after power on. Otherwise, skip this step to step 2.

2. Press Prg, press **F6** >, F6 BRK.PT., and then press RESUME. The system displays all saved breakpoint files. See the figure below:

Figure 4.2 Select a breakpoint file to resume



3. Use the ▲, ▼, PgUp, and PgDn keys to select a breakpoint file to resume as required.

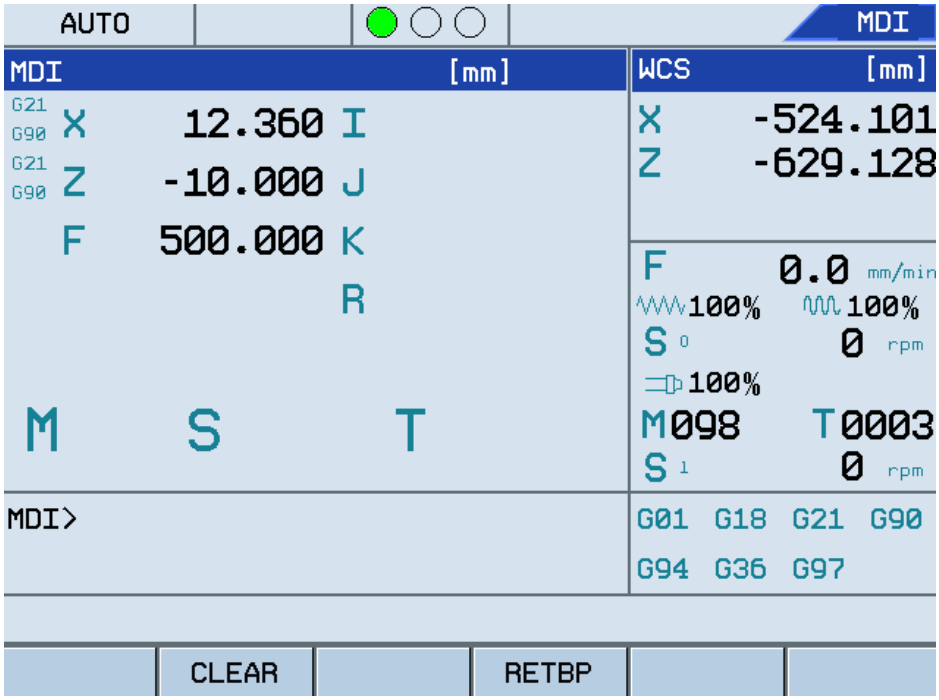
4. Press Enter. The system will resume the breakpoint of the program based on the information saved in the breakpoint file. In this way, you may back to the breakpoint in the MDI mode. You may press the F5 key to delete the breakpoint file.

4.5.3 Return to Breakpoint

After saving the breakpoint, you may proceed with the program through breakpoint location if no axis motion has been performed. Follow the instructions below:

1. Resume the breakpoint based on the instruction in section 4.5.2.
2. Move the cutting tool near to the break point, and make sure no collision during this operation.
3. Under the MDI main menu, press F4 RETBP. The system automatically inputs the breakpoint information into the MDI program segment. See the figure below:

Figure 4.3 Return to breakpoint



4. You may also manually enter the data information, and then press Enter.
5. In the Single Block or Auto mode, press  to run the program from the breakpoint.

4.6 Operation During Program Running

4.6.1 Adjust Feed Speed

In the Auto or Single Block mode, you may adjust the feed speed by using the override keys if the feed rate of the F-code programming is too high or too low. The adjustment range is **0%** to **120%**.

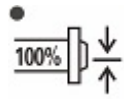
4.6.2 Adjust Rapid Traverse Speed

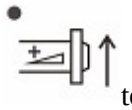
In the Auto or Single Block mode, you may press the rapid traverse adjustment button to adjust the

rapid traverse speed. Pressing the minus "-" key reduces the feed rate by 10% of the 100% rapid traverse speed. Pressing the minus "+" key increases the feed rate by 10% of the 100% rapid traverse speed. The range of adjustment is **0** to **100%**.

4.6.3 Adjust Spindle Speed

In the Auto or Single Block mode, you may adjust the spindle speed by using the three override keys if the federate of the S-code programming is too high or low (except for the tapping command).

Press 

(the indicator is lit) and the spindle speed is set to the full value (**100%**). Press  to increase the speed by 10%. Press  to decrease the speed by 10%.

The adjustment range is 10%-150%.

Do not adjust the spindle speed during mechanical gear rate switching.

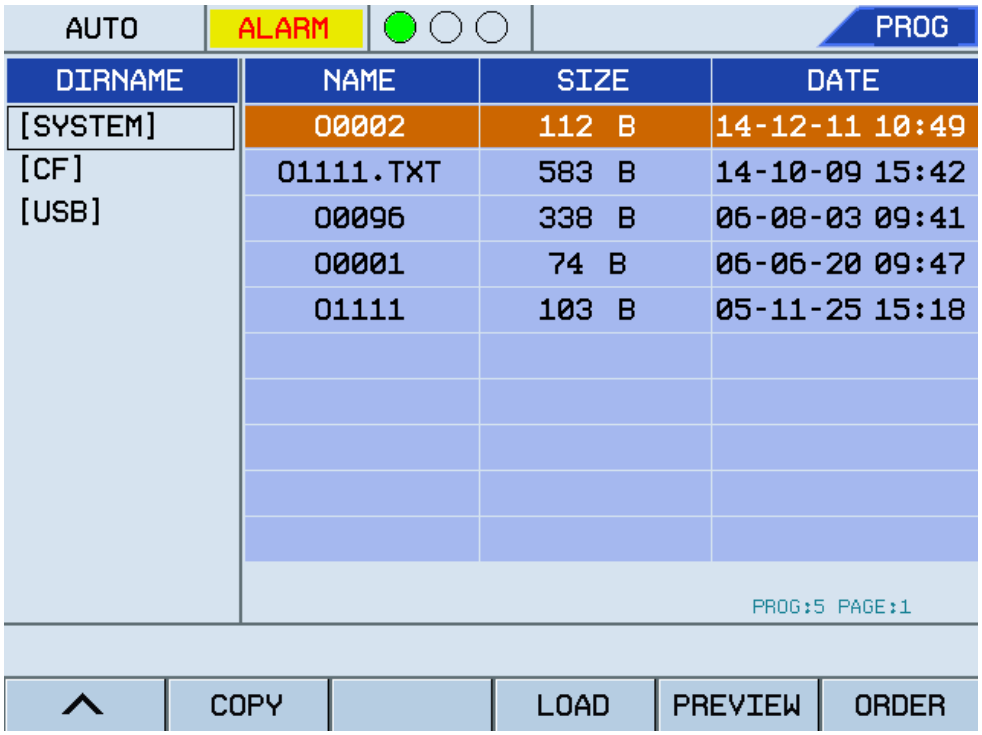
5 Program Editing and Management

This section describes the program-related operation under the Prg menu, such as selecting a program, editing a program, verifying a program, and rerunning a program.

5.1 Select a Program

Press Prg, and press the F1 OPEN submenu. The system displays the programs in each storage card. See the figure below:

Figure 5.1 Select a program file in a storage card



5.1.1 Select a Program

You can select a program from the storage cards as required.

1. System disk: G-code programs in the built-in system flash disk.
2. USB: G-code programs in the USB flash disk.

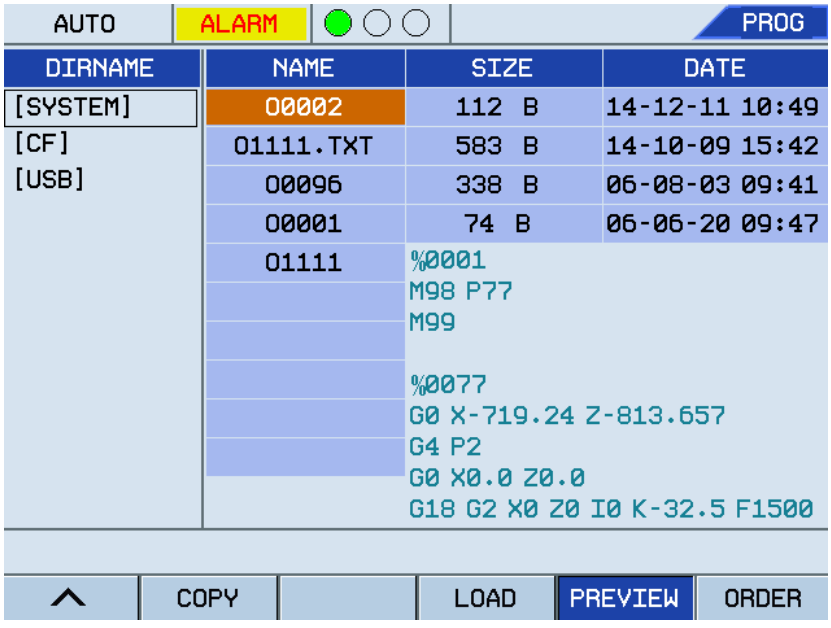
If you select no program, the system displays the program saved in the buffer zone previously.

Press ◀ and ▶ to switch between the program storage card (displays only the programs in the root directory) and corresponding program list screens. Press ▲, ▼, PgUp and PgDn keys to select programs to be run, and press Enter or F4 LOAD to confirm and add the selected programs. In the Auto or Single Block mode, press ● to run the program. If an error occurs, the message "ALARM" will be displayed and the indicator will blink continuously. You may press Dgn to check detailed alarm information.

5.1.2 Preview Programs

When the cursor is placed over the selected program, press F5 PREVIEW to preview the program. Use the ▲, ▼, PgUp, and PgDn keys to preview other programs as required. See the figure below:

Figure 5.2 Pre-view a selected program

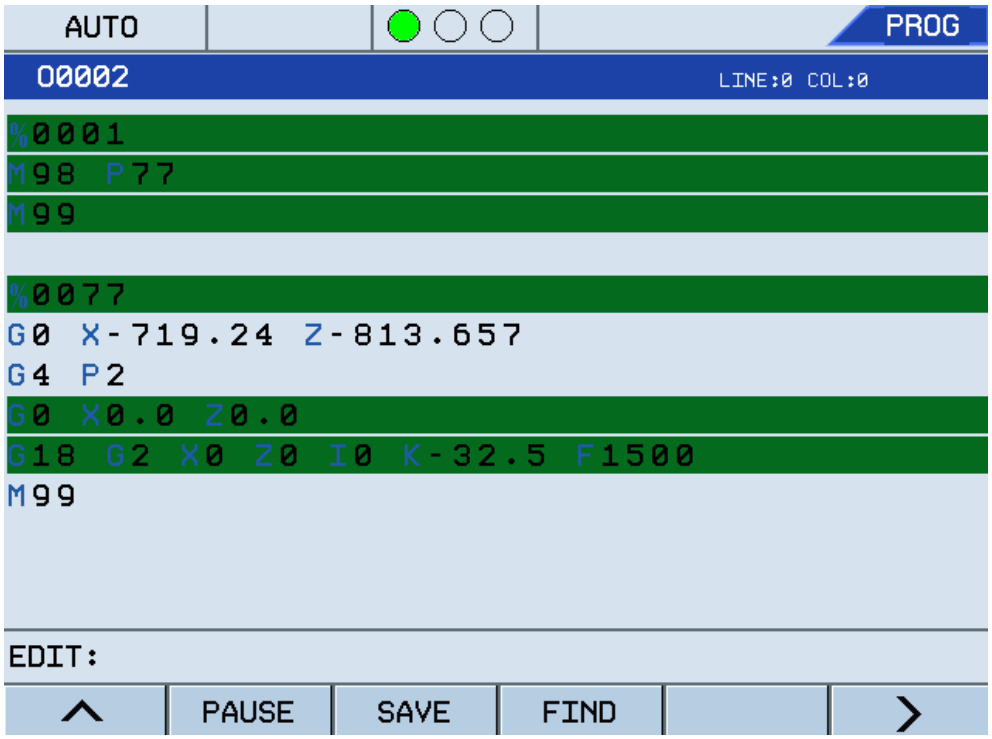


5.2 Edit a Program

5.2.1 Edit a program

If a program error occurs or a program is required to edit, you may press Prg to select the program, and then press **F2** EDIT to edit it as required. See the figure below:

Figure 5.3 Edit a selected program



The following shortcut keys are used to edit a program:

Delete: Delete the character right to the cursor and maintain the cursor position. The remaining characters right to the cursor will shift leftwards accordingly with one-character space.

PgUp: Move the program to be edited one screen up to the program header and maintain the cursor position. If it has reached the program header, the cursor will move to the first character in the first row of the program.

: Move the program to be edited one screen down to the program end and remain the cursor position. If it has reached the program end, the cursor will move to the first character in the last row of the program.

: Delete the character left to the cursor and the cursor will shift leftwards with one-character space. The remaining characters right to the cursor will shift leftwards accordingly with one-character space.

: The cursor shifts leftwards with one-character space.

: The cursor shifts rightwards with one-character space.

: The cursor shifts one row upwards.

: The cursor shifts one row downwards.

Note: The program block displayed in the dark color tone cannot be directly edited. You need to press F4 

, and then press **Y** or 

after the message "Exit the current running Y/N? (Y)" is displayed. After that, you may edit the program.

5.2.2 Save a Program

To save an edited program or a created program, press the F3 

. The system automatically saves the program with the previous program name.

If the system failed to save the edited program, a corresponding failure message will be displayed. In this case, the program is a read-only file and cannot be saved with modification, but you may save it as another file. If the failure is caused of insufficient storage space, you may delete unnecessary files and resave the program.

5.2.3 Search Information

To search specific content (numbers or characters) within a program, press F4"Search", and the

system prompts you to enter the content to search. Press Enter, the system highlights the first block of content that meets the search conditions. You may press F5 NEXT to view the next block of content that meets the search conditions.

5.2.4 Locate a Line

To rapidly locate a line within a program, press F6 >, and then press F3 TO LINE. The system prompts you to enter the line number to locate. Enter the line number and press Enter, and the cursor will automatically go to the end of the line within the program.

5.2.5 Save as

To save an edited or created program as another file, press F6 >, and then press F6 SAVE AS. The system displays the default file name. Press Enter to save the current program file with the default name. You may change the file name, and then press Enter to save the program with another name. It is recommended to use a file name that has not existed. If the system fails to save the edited program, a corresponding failure message will be displayed. In this case, the program is a read-only file and cannot be saved with modification, but you may save it as another file. If the failure is caused of insufficient storage space, you may delete unnecessary files and resave the program.

5.2.6 Replace a String of Characters

To replace a string of characters in the program with other characters, press F6 >, and then press F2 REPLACE. The system prompts you to enter the characters to be replaced. Press Enter after entering required information. After that, enter the characters to replace the

previous ones, and then press Enter. The system will automatically replace the string of characters as required.

5.2.7 Delete a Line

To delete a line within a program, move the cursor to the line, press F6 >, and then press F4 DEL LINE. The system will automatically delete the content of the line.

5.2.8 Edit a Block

To edit a block within a program, press F6 >, and then press F5 BLC OPRT. Based on the cursor position, press F1 BLC HEAD or F2 BLC TAIL as required, and then press F3 CUT, F4 COPY or F5 PASTE to cut, copy, or paste block information.

5.3 Create a Program

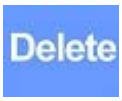
Press the Prg key, and then press the F3 NEW key. The system prompts you to enter a new program name (a random G code name by default). After entering the program name, you may press Enter to create a new program by pressing the character keys on the NCP board.

5.4 Delete a Program

You may delete unnecessary program files when there is no sufficient space with too many unnecessary files. Do as follows:

1. Press Prg, press F1 OPEN, and then use the ▶, ◀, ▲, and ▼ keys to select the disk where files required to delete are placed.

2. Select a file from the program list on the right, and use the  ,  ,  , and  keys to select the program to be deleted.

3. Press  , and the system displays the message  .

4. Enter **Y** to delete the program.

5.5 Copy a Program

Follow the instructions below to copy a program:

1. Press  , press F1  , and then use  ,  ,  , and  keys to select the disk where files required to copy are placed.

2. Select a file from the program list on the right, and use the  ,  ,  , and  keys to select the program to be copied.

3. Press F2  , and select a target directory by using  and  .

4. Press F3  to paste the program.

5.6 Verify a Program

Selected programs can be automatically verified and errors will be reported if there is any. New programs that have not been run on the machine should be verified before auto running. Follow the instructions below to verify programs:

1. Select a program required to be verified.

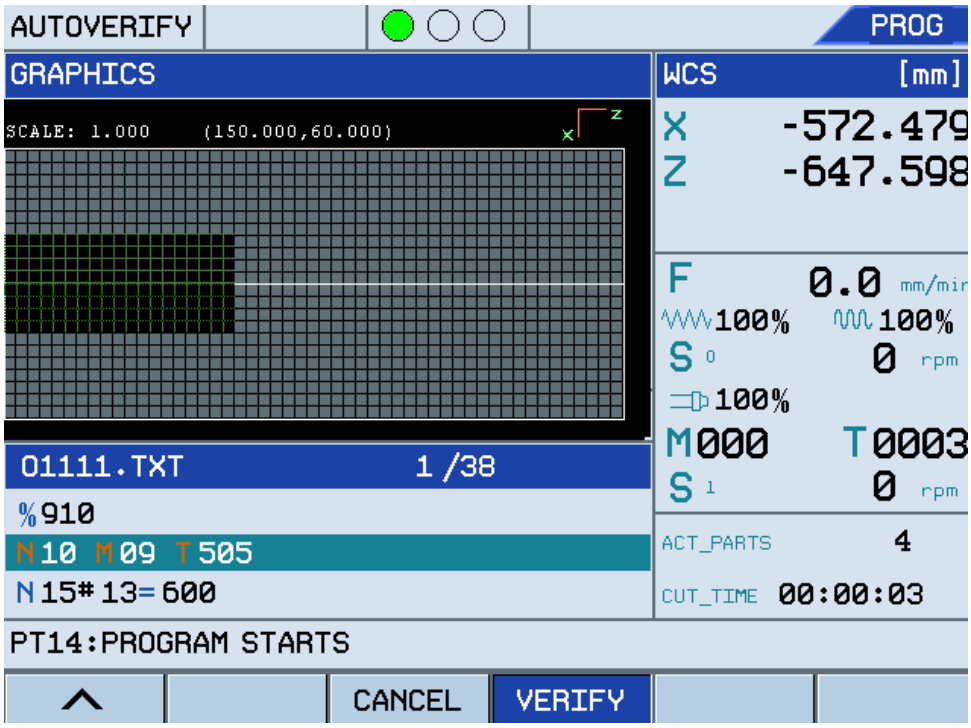
2. Press F6  under  , and press F4  to select a

program verification mode.

3. Press  or  to enter the program running mode.

4. Press  to start verifying the program,

Figure 5.4 Verify programs



If the program is correct, the cursor returns to the head of the program after verification. If the program is incorrect, the system displays the error message on the screen and the indicator blinks.

You may press 

to view related error information.

Note:

- The machine cannot run during program verification.
- It is recommended to select difference graphs for verification results to ensure correct program. For more information, see section 1.3.5 (Figure 1.37).

5.7 Restart a Program

If you need to restart a program from the beginning after the program is stopped, press 

, and press F5 

. In the Auto or Single Block mode, press  to restart the program from the beginning.

6 User Operation and Maintenance Information

6.1 Environmental Requirements

The table below describes the environmental requirements:

Items	Description
Working temperature (ºC)	**0** to **+45**, non-freezing
Temperature change	< **1.1** ºC/min
Relative humidity	**90%** RH or lower (non-condensable) Normal: **75%** or lower Short period (within a month): Max. **95%**
Storage temperature (ºC)	**-20** to **+60**, non-freezing
Storage humidity	Non-condensable
Surrounding environment	Indoor (non-exposed to sun), anti-sepsis, anti-burning, anti-fog, anti-dust
Height	A maximum of **2000** meters above sea level
Vibration (m/s)	**10** to **60** Hz: **5.9** (**0.6**G) or lower

6.2 Grounding

Correct grounding is critical for the numerical control unit and other electrical devices. Correct grounding may:

1. Protect operators from electric shock or injury caused by no grounding or incorrect grounding.
2. Protect the electrical devices from inductive interference which may lead to errors or unexpected results.

When installing machine, reliable grounding must be provided. The neutral line in the power grid cannot be used as the grounding line, otherwise, it may cause device damage or abnormal operation, or even casualties.

6.3 Power Conditions

The power supply of HNC turning system is provided by the electric cabinet. For more information about machine power supply, see the installation guide of the machine.

6.4 Clean Fan Filter

Filters are used on cooling fans to prevent dust from entering into devices. However, it may prevent adequate cooling if the filters become clogged. It is recommended to clean the filters every three months. In dusty environments, clean the filters more often.

6.5 Operations After Being Left Unused

After a long period of being left unused, numerical control devices should be cleaned and dried, so should the wiring and ground connections. Once power is resumed after being left unused, observe the operation for several hours to make sure there is no problem.

7 Appendix 1 Parameter Settings

This section describes the parameter structure and settings of Century Star HNC turning system.

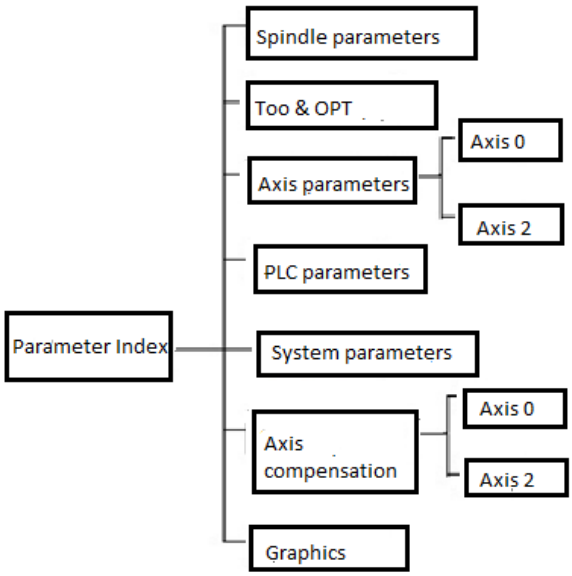
7.1 Overview

You need to understand the function of a specific parameter and its original value before modification. Incorrect parameter settings or modification may cause critical unexpected results. The modification takes effect only after the CNC system restarts.

Common terms and descriptions:

Parameter tree: consists of multiple levels of parameters. See the figure below:

Figure 7.1 Parameter tree



Window: An area where to display and modify parameter values.

Common functional buttons to check and modify parameters:

1. F1 to F6: Enters corresponding menus and windows to achieve specific functions.
2. Enter:
   a. Confirms to modify parameters.

b. Confirms the entered content.

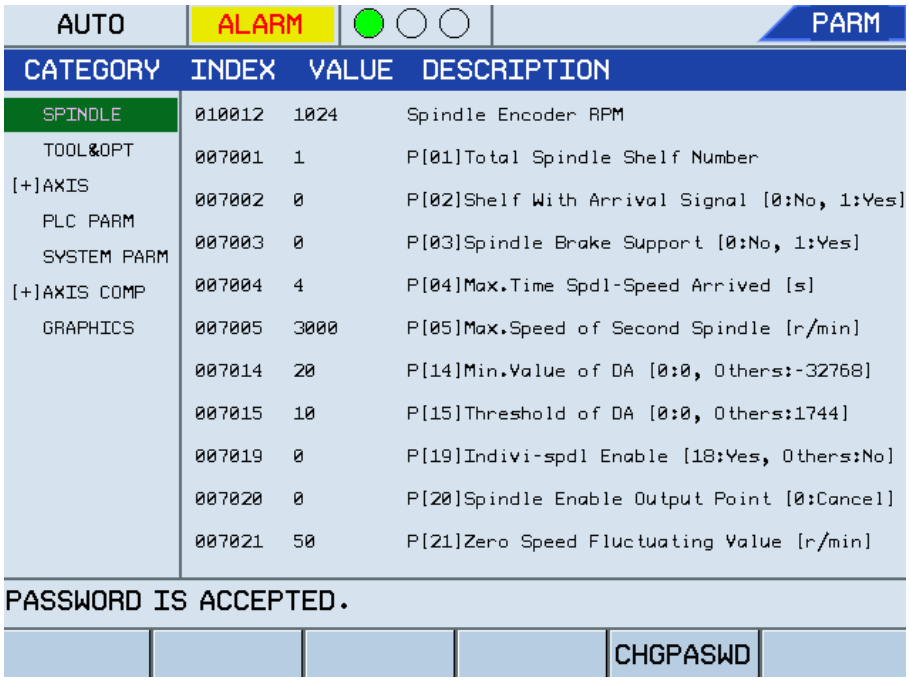
3.  ,  ,  , and  : moves the cursor or cursor bar within a menu or window.

4.  and  : shifts the page up or down.

7.2 Check and Set Parameters

Under  , you may press  and input a correct password to enter the parameter window. See the figure below:

Figure 7.2 Parameter window



Follow the instructions below to check parameters and perform settings:

1. Press the  and  keys to select the parameter type to be edited, and then press the  key to select the parameter to be edited. The parameter information is displayed in the lower corner of the window.

2. Press the  and  keys to select the parameter to be checked or set. Press PgUp or PgDn to shift the page up or down.
3. Press Enter to enter the parameter settings mode (The cursor blinks besides the parameter value).
4. Enter a required parameter value.
5. Press the Enter key,
6. Press F4 SAVE after modifying all required parameters.
7. Repower the CNC system after successful saving so that the modification takes effect.

7.3 Detailed Parameter Description

7.3.1 Spindle Parameters

- **Spindle Encoder RPM [CNC factory]**

  Value: **-32768** to **32767**, default value: **1024**

  Description: The pulse number fed back from the encoder to the numerical control device after each revolution of the spindle.

- **Total Spindle Shelf Number [user]**

  Value: **-32768** to **32767**, default value: **1**

  Description: The total shelf number of the spindle.

- **Shelf with Arrival Signal [user]**

  Value: **0**, **1**

  Description: The value **0** indicates no spindle shelf arrival signal while **1** indicates that there is spindle shelf arrival signal.

- **Spindle Brake Support [user]**

  Value: **0**, **1**

  Description: The value **0** indicates that the spindle does not support brake while **1** indicates

that the spindle supports brake.

- **Max. Time for Spdl-Speed Reached [user]**

  Unit: Millisecond

  Value: **0** to **32767**, default value: **4**

  Description: The maximum time to receive the signal that the spindle reaches a specific speed.

- **Max.Speed of Second Spindle [user]**

  Unit: Revolution(s) Per Minute (rpm)

  Value: **0** to **32767**, default value: **3000**

  Description: The theoretical maximum speed of the second spindle motor under 10V voltage.

- **Min.Value of DA [user]**

  Value: **-32768** to **32767**, default value: **0**

  Description: The minimum digital and analog (DA) value of the spindle voltage from 0 to 10V.

- **Threshold of DA [user]**

  Value: **-32768** to **32767**, default value: **0**

  Description: The DA threshold value of the spindle voltage from 0 to 10V.

- **Enable Indivi-Spdl Output [user]**

  Value: **0** to **32767**, default value: **0**

  Description: The value **18** indicates that there is individual Spindle Enable output point while other values indicates that there is no individual Spindle Enable output point.

- **Enable Spindle Output Point [user]**

  Value: **0** to **32767**, default value: **0**

  Description: The value **0** indicates no output point while other values indicate signal point. For example, **23** indicates **Y2.3**.

- **Zero Speed Threshold [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **50**

Description: The actual spindle speed lower that this value is considered as the zero speed status.

- **Error Between Cmd. And Act. Speed [user]**

  Value: **0** to **32767**, default value: **20**

  Description: The difference value, in percentage, between the spindle command speed and actual speed. If the difference value is set to **0**, then the spindle speed is reached when the detected spindle speed reaches the value specified by **Zero Speed Fluctuating Value**,

- **Spindle Brake Wait Time [user]**

  Unit: Millisecond (ms)

  Value: **0** to **32767**, default value: **0**

  Description: The interval from receiving spindle stop command to outputting brake signal.

- **Spindle Brake Duration [user]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

  Description: The duration during which the spindle brake status is maintained. The brake will be released when the duration exceeds this value.

- **Original Manual Spdl-Speed [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **300**

  Description: The default spindle speed after power on.

- **Max. Detected Speed at Shelves 1/2/3/4 [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **0**

  Description: The theoretical maximum detected speed at each spindle shelf.

- **Min. Allowed Speed at Shelves 1/2/3/4 [user]**

  Unit: R rpm

  Value: **0** to **32767**, default value: **0**

  Description: The theoretical minimum speed at each spindle shelf.

- **Max. Allowed Speed at Shelves 1/2/3/4 [user]**

Unit: Revolution(s) Per Minute

Value: **0** to **32767**, default value: **0**

Description: The theoretical maximum speed at each spindle shelf.

- **Initial Spindle Override [user]**

  Unit: percentage (%)

  Value: **0** to **150**, default value: **0**

  Description: The initial spindle override ratio after power on.

- **Star/Triangle Start [user]**

  Value: **0**, **1**

  Description: The value **0** indicates that star/triangle start is not supported while **1** indicates that star/triangle start is supported.

- **Switch Interval Time [user]**

  Unit: ms

  Value: **0** to **32767**, default value: **2**

  Description: The interval for the spindle switching from star output to triangle output.

- **Star Output Point [user]**

  Value: **0** to **32767**, default value: **0**

  Description: The value **0** indicates no output point while other values indicates signal points. For example, the value **23** indicate **Y2.3**.

- **Triangle Output Point [user]**

  Value: **0**, default value: **0**

  Description: The value **0** indicates no output point while other values indicates signal points. For example, the value **23** indicate **Y2.3**.

- **Spindle negative 10V [user]**

  Value: **0**, default value: **0**, **1**

  Description: The value **0** indicates that the spindle analog voltage is **0** to **10V** while **1** indicates that the spindle analog voltage is **-10V** to **10V**.

- **Spindle Encoder Exists [user]**

  Value: **0**, default value: **0**, **1**

Description: The value **0** indicates that there is no feedback from the spindle encoder while **1** indicates that there is feedback from the spindle encoder.

- **Max Spindle Shift Time [user]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

  Description: The maximum duration for spindle shelf shift. If the shelf shift is not completed within this specified value, an alarm will be reported.

- **Spindle Shift With Shake [user]**

  Value: **0**, **1**

  Description: The value **0** indicates spindle shift without shake while **1** indicates spindle shift with shake.

- **Spindle Reverse Mode [user]**

  Value: **0**, **1**

  Description: The value **0** indicates counter-clockwise spindle rotation controlled by Y signal while **1** indicates counter-clockwise rotation controlled by the positive and negative polarity (-, +) of analog voltage.

- **Delay of Spindle Feedback Time [user]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

  Description: The delay allowed for spindle feedback.

- **Max. Tapping Speed [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **1000**

  Description: The allowed maximum spindle speed during tapping.

- **Min. Tapping Speed [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **0**

  Description: The allowed minimum spindle speed during tapping.

- **Pre-stop Tapping Numerator [user]**

Value: **1** to **32767**, default value: **1**

Description: The numerator of pre-stop tapping.

- **Pre-stop Tapping Temp-Numerator [user]**

  Value: **1** to **32767**, default value: **1**

  Description: The temporary numerator for pre-stopping tapping.

- **Spindle Speed When Shield Open [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **0**

  Description: The spindle speed when the shield is open.

- **Specified Speed When Shift [user]**

  Unit: rpm

  Value: **0** to **32767**, default value: **0**

  Description: The revolution speed during spindle shelf shift.

- **Stop Spdl at ProgEnd[user]**

  Value: **0**, **1**

  Description: The value **0** indicates to stop the spindle after the program ends while **1** indicates not to stop the spindle after the program ends.

- **Spindle Encoder Direction [user]**

  Value: **32**, **33**

  Description: The value **32** indicates the spindle encoder direction is positive while **33** indicates the spindle encoder direction is negative.

7.3.2 Tool & OPT

- **Tool Positions [user]**

  Value: 4, 6, 8, others

  Description: The value **4**, **6**, or **8** indicates the tool station number on the turret, while others indicates the turret type is **Horizon**.

- **Tool Change Duration [user]**

  Unit: ms

  Value: **0** to **32767**, default value: **6**

Description: The maximum duration allowed for tool change. If the tool change exceeds this specified duration, an alarm will be reported.

- **Turret Delay Time [user]**

  Unit: ms

  Value: **0** to **32767**, default value: 5**0**

  Description: The delay for turret clockwise rotation.

- **Turret Brake Time [user]**

  Unit: ms

  Value: **0** to **32767**, default value: **1000**

  Description: The delay for turret counter clockwise rotation.

- **Chuck Type [user]**

  Value: **0**, **1**, **2**

  Description: The value **0** indicates **Manual**, **1** indicates **Persistent**, while **2** indicates **Pulse**.

- **Protection Door [user]**

  Value: **0**, **1**

  Description: The value **0** indicates not supporting protection door while **1** indicates supporting protection door.

- **Sleeve [user]**

  Value: **0**, **1**

  Description: The value **0** indicates not supporting sleeve while **1** indicates supporting sleeve.

- **Initial Chuck Mode [user]**

  Value: **0**, **1**

  Description: The value **0** indicates external chuck while **1** indicates internal chuck.

- **Turret Brake Signal [user]**

  Value: **0**, **1**

  Description: The value **0** indicates no turret brake signal while **1** indicates there is turret brake signal.

- **T-Code Pre-Finished Delay[user]**

  Unit: ms

Value: **0** to **32767**, default value: **0**

Description: The delay of pre-finishing the T-command. The value **0** indicates invalid.

- **Tool Arrival Signal Always [user]**

  Value: **0**, **1**

  Description: The value **0** indicates not to detect tool arrival signal while **1** indicates to always detect tool arrival signal.

- **Chuck Signal [user]**

  Value: **0**, **1**

  Description: The value **0** indicates no chuck clamping and releasing arrival signal while **1** indicates chuck clamping and releasing arrival signal.

- **Turret Direction [user]**

  Value: **0**, **1**

  Description: The value **0** indicates positive X-axis pointing down while **1** indicates positive X-axis pointing up.

7.3.3 Feed Axis Parameters (X axis)

- **Axis Type [user]**

  Value: **0**, **1**, **2**, **3**

  Description: The value **0** indicates no axis, **1** indicates Linear axis, while **2** and **3** indicates rotation axis.

- **External Pulse Equivalent Numerator and External Pulse Equivalent Denominator [machine factory]**

  Value: **-32768** to **32767**, default value: **1**

  Description: The value of External Pulse Equivalent Numerator/External Pulse Equivalent Denominator is the actual pulse equivalent, that is, the actual axis motion distance corresponding to each position unit, or the system electric gear rate.

  The unit of the external pulse equivalent numerator is micrometer (μm), and there is no unit for the external pulse equivalent denominator.

$$\frac{\textit{External pulse equivalent numerator}\ (\mu m)}{\textit{External pulse equivalent denominator}} = \frac{\textit{Machine motion distance per motor revolution}\ (\mu m)}{\textit{Pulse sent by CNC per motor revolution}}$$

Users may change the electric gear rate by setting the external pulse equivalent numerator and the external pulse equivalent denominator, or change the motor rotation direction by changing the positive or negative symbol (-, +) of the electric gear rate.

Example:

GK Series servo motor (2500 threads encoder) with HSV-16D driver; screw: 6 mm; gear rate: 2: 3

Then after each motor revolution, the CNC system will send 40000 pulses (quadruple frequency for motor encoder and four-point subdivision for command, thus sent pulses = 2500*4*4=40000). The machine moves 4 mm (6 mm *2/3 = 4 mm), that is 4000 μm. Therefore, the value of External Pulse Equivalent Numerator/External Pulse Equivalent Denominator is 1/10 (4000/40000=1/10).

It is equivalent to set the External Pulse Equivalent Numerator to **1** and External Pulse Equivalent Denominator to **10**, or to set the External Pulse Equivalent Numerator to **2** and External Pulse Equivalent Denominator to **20**.

- **Positive Software Limit [machine factory]**

  Unit: Micrometer (μm)

  Value: **-2147483648** to **2147483647**, default value: **8000000**

  Description: The positive software limit point. This parameter value takes effect only after the machine homes to the reference point.

- **Negative Software Limit [machine factory]**

  Unit: μm

  Value: **-2147483648** to **2147483647**, default value: **8000000**

  Description: The negative software limit point. This parameter value takes effect only after the machine homes to the reference point.

- **REF Direction [machine factory]**

  Value: -, +

  Description: The initial motion direction of axis to reference point after corresponding reference command is sent. If users have pressed the button for homing the axis to the reference point when the corresponding command is sent, the initial motion direction

depends on the mode for homing to the reference point.

- **Reference Point Position [machine factory]**

  Unit: μm

  Value: **-2147483648** to **2147483647**, default value: **0**

  Description: The coordinate position of the reference point in the machine coordinate system. Generally, the origin point of the machine coordinate system is set as the reference point. Therefore the reference point is generally set to **0.**

- **Reference Point Offset [machine factory]**

  Unit: μm

  Value: **-32768** to **32767**, default value: **0**

  Description: When homing to the reference point, after the axis reaches the Z pulse, the point will not be defined as the reference point. The axis will continue to move a distance specified by this value, and then the point will be defined as the reference point.

- **Rapid Traverse Speed at RER [machine factory]**

  Unit: Millimeter/Minute (mm/min)

  Value: **0** to **65535**, default value: **500**

  Description: The rapid traverse speed before the reference point key is pressed.

  Note: This value must be smaller than the maximum rapid traverse speed.

  If this speed is set too fast, the distance between the reference point switch and nearby limit switch (positive limit switch) cannot be too small. Otherwise, an over-limit alarm will be reported and the system fails to home to reference point if the speed for homing to the reference point is too fast to speed down and the limit switch is pressed.

  In addition, the effective travel distance to the reference point cannot be too short, otherwise, the system may fail to home to the reference point if the speed for homing to the reference point is too fast to speed down before crossing the reference point.

- **Positioning Speed at REF [machine factory]**

  Unit: mm/min

  Value: **0** to **65535**, default value: **200**

  Description: The positioning speed after the reference point key is pressed.

Note: This parameter value must be less than the value specified by **Rapid Traverse Speed at RE**R.

- **Unidirectional Positioning Offset [machine factory]**

  Unit: μm

  Value: **-32768** to **32767**, default value: **1000**

  Description: In the unidirectional positioning mode of G60 workstation, the difference value between the speed slowdown point and the positioning point when the rapid traverse speed changes to the positioning speed.

  Uni-direct Positioning Offset > 0: positive positioning

  Uni-direct Positioning Offset < 0: negative positioning

- **Maximum Rapid Traverse Speed [machine factory]**

  Unit: mm/min

  Value: **0** to 65535, default value: **1000**

  Description: The maximum rapid positioning speed of G00 (no processing) When the override ratio of Rapid Traverse Speed reaches the maximum.

  Note: The value of **Maximum Rapid Traverse Speed** must be the largest value among all speed values of the axis. The value of **Maximum Rapid Traverse Speed** is very closely related to the value of **External Pulse Equivalent Numerator and External Pulse Equivalent Denominator**. You need to properly define this value to make sure the value does not exceed the motor revolution speed range. For example, if the rated speed of the motor is **2000** rpm, and connected to a ball screw that is six millimeter away from the screw with a pair of synchronous gear belts whose gear ratio is **2**: **3**, then the maximum rapid traverse speed cannot exceed 8000 mm/min (≤ 2000 * (2/3) * 6 = 8000).

- **Maximum Feedrate [machine factory]**

  Unit: mm/min

  Value: **0** to **65535**, default value: **500**

  Description: The maximum feed rate allowed by the executed commands (e.g G01, G02) under certain precision conditions.

  Note: This parameter value is related to processing requirements, mechanical transmission

and load. The maximum feed rate must be less than the maximum rapid traverse speed.

- **Jerk Time for Rapid Traverse [machine factory]**

  Unit: ms

  Value: **0** to **800**, default value: **64**

  Description: The time when the speed increases from **0** to **1** m/min or from **1** m/min to **0** during G00 positioning (no processing). The larger the time constant is, the slower the acceleration and deceleration.

  Note: This value depends on the motor size, the driver performance, and the load. It is generally between **32** to **150**. For example, the servo motor is generally **32** and the stepper motor is **64**.

- **Jerk for Rapid Traverse [machine factory]**

  Unit: ms

  Value: **0** to **150**, default value: **32**

  Description: The acceleration time constant during rapid traverse, which is generally **16**, **32**, or **64**. The larger the time constant is, the slower the acceleration.

  Note: This value depends on the motor size, the driver performance and the load. It is generally between **16** to **100**. For example, the servo motor is generally **16** and the stepper motor is **32**.

- **Jerk Time for Machining [machine factory]**

  Unit: ms

  Value: **0** to **800**, default value: **64**

  Description: The time when the speed increases from **0** to **1** m/min or from **1** m/min to **0** during processing (G01, G02…). The larger the time constant is, the slower the acceleration and deceleration.

  Note: This value depends on the processing requirements and load, the motor size, the driver performance and the load. It is generally between **32** to **150**. For example, the servo motor is generally **32** and the stepper motor is **64**.

- **Jerk for Machining [machine factory]**

  Unit: ms

Value: **0** to **150**, default value: **32**

Description: The acceleration time constant during processing, which is generally **16**, **32**, or **64**. The larger the time constant is, the slower the acceleration.

Note: This value depends on the motor size, the driver performance and the load. The value generally ranges between **16** and **100**. For example, the servo motor is generally **16** and the stepper motor is **32**.

- **Positioning Tolerance [machine factory]**

  Unit: μm

  Value: **0** to **255**, default value: **20**

  Description: The maximum allowed positioning error.

  Note: This value depends on the motor size, the driver performance and the load. The value generally ranges between **10** and **50**. If the value is too small, the system may not reach the positioning tolerance value and stop the motor. If the value is too large, the processing precision may be affected. Generally, the larger the machine is, the larger this value is. The lower the machine transmission and precision is, the larger this value is.

  If a stepper motor is used, it is recommended that you set the value to integral multiple of the movement distance corresponding to each step.

- **Servo Type [machine factory]**

  Value: **45, 46**, default value: **45**

  Description: Set this value to **45** for the servo motor that uses pulse interface with feedback. Set this value to **46** for the servo motor that uses pulse interface without feedback (e.g stepper servo motor).

- **Maximum Track Error [CNC factory]**

  Unit: μm

  Value: **0** to **65535**, default value: **12000**

  Description: This parameter is used to report an alarm if the track error exceeds the specified value. The value **0** indicates no track error alarm. This parameter must be set based on the maximum speed and servo loop hysteresis properties. See the following expressions (approximate formula):

Maximum speed * (10000 – *position loop feedforward coefficient* * 0.7)/*position loop proportional coefficient*/3

Unit:

a. Maximum track error: um

b. Maximum speed: mm/min

c. Position loop feedforward coefficient: 1/10000

d. Position loop proportional coefficient: 0.01 1/s

- **Pulses per Motor Revolution [CNC factory]**

  Value: **0** to **65535**, default value: **10000**

  Description: The pulse count received by the CNC system per motor revolution. That is, the pulses fed back from the servo system or the servo motor to the CNC system. As the system has four subdivision,

  the value of ***Pulses per Motor Revolution*** *= the actual pulse count of the servo motor position encoder* x 4

- **Stepper Motor Pulses [CNC factory]**

  Value: **0** to **65535**, default value: **0**

  Description: e.g. If the pulse of a two-phase stepper motor is **4,** then this value should be set to **4**.

- **E-Gear Rate Numerator (0 for no feedback) [CNC factory]**

  Value: **0** to **65535**, default value: **-4**

  Description: Set this parameter to **0** if no feedback is required.

- **E-Gear Rate Denominator (0 for no feedback) [CNC factory]**

  Value: **0** to **65535**, default value: **1**

  Description: Set this parameter to **0** if no feedback is required.

  E-Gear Rate Numerator/E-Gear Rate Denominator = CNC system command/position value fed back by the servo motor to the CNC system

  This is used to adjust the situation when the CNC command is inconsistent with the feedback information. As the system has four subdivision, these two parameter values are generally **4: 1** or **-4**: **1** if the feed driver does not perform multiplication or subdivision frequency for CNC commands.

- **Enable Zero-Pulse Input [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to disable Zero-Pulse Input while **1** indicates to enable Zero-Pulse Input.

- **Stepper Motor [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates non-stepper motor while **1** indicates stepper motor

7.3.4 PLC Parameters

- **REF Alarm [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates no alarm is reported when the feed axis is not homed to the original point after power on, while **1** indicates an alarm is reported when the feed axis is not homed to the original point after power on and the work mode cannot switch to the Auto or Single Block mode.

- **MPG Axis Chosen [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates no axis selection for MPG while 1 indicates axis selection is provided for MPG.

- **Installed Home Dog [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates no Home Dog is installed and the floating reference point automatically takes effect, while **1** indicates the Home Dog is installed.

- **Lubrication Off Time [CNC factory]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

  Description: The duration during which the lubrication function is disabled.

- **Lubrication On Time [CNC factory]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

Description: The duration during which the lubrication function is enabled.

- **Lubrication Alarm Detection Delay [CNC factory]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

  Description: If no lubrication signal is detected when the duration exceeds this value, an alarm will be reported.

- **Lubrication Mode [CNC factory]**

  Unit: meter (m)

  Value: **0**, other values; default value: **0**

  Description: The value **0** indicates lubrication based on time while other values indicates lubrication based on distance.

- **Hydraulic Alarm Detection Delay [CNC factory]**

  Unit: ms

  Value: **0** to **32767**, default value: **0**

  Description: If hydraulic alarm signal is constantly detected during this specified duration, an alarm will be reported.

- **Initial Rapid Override [CNC factory]**

  Unit: %

  Value: **0** to **100**

  Description: The initial rapid override ratio after power on.

- **Each Axis Limited [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates no independent axis limit for each axis and all axis use the same input point (X2.0), while **1** indicates independent axis limit for each axis.

- **MPG Pulse Reverse [CNC factory]**

  Value: **0**, **1**

  Description: When the actual axis motion direction is in the opposite direction of the MPG command, you may set this parameter value to **1** to change the direction.

- **Mach-Brake Includes MST [CNC factory]**

Value: **0**, **1**

Description: The value **0** indicates the MST function is available after machine brake while **1** indicates the MST function is unavailable after machine brake.

- **Rapid Jog Feed Has SelfLock [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates no self-lock for rapid jog feed while **1** indicates available self-lock for rapid jog feed.

- **(Lathe: X Mill: Z) Wait Time for Brake Signal [CNC factory]**

  Unit: ms

  Value: **0**, other values

  Description: The value **0** indicates no brake while other values indicates the wait time before the brake signal is output.

- **MPG Coding Form [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates independent MPG coding form while **1** indicates compound MPG coding form.

- **Manual Speed of Feed [CNC factory]**

  Value: **1** to **10**, default value: **1**

  Description: The value **1** indicates the minimum manual feed speed while **10** indicates the maximum manual feed speed.

- **Speed to Floating Origin [CNC factory]**

  Value: **1** to **10**; default value: **1**

  Description: The value **1** indicates the minimum speed for homing to the floating origin point while **10** indicates the maximum speed.

- **No MCP Stop Signal [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to detect the MCP Stop signal while **1** indicates not to detect the MCP Stop signal.

- **No MPG Stop Signal [CNC factory]**

Value: **0**, **1**

Description: The value **0** indicates to detect the MPG Stop signal while **1** indicates not to detect the MPG Stop signal.

- **No Drivers Ready Signal [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to detect the driver ready signal while **1** indicates not to detect the driver ready signal.

- **No Limit Alarm Signal [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to detect the limit alarm signal while **1** indicates not to detect the limit alarm signal.

- **No External Alarm Signal [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to detect the external alarm signal while **1** indicates not to detect the external alarm signal.

- **No Spindle Alarm Signal [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to detect the spindle alarm signal while **1** indicates not to detect the spindle alarm signal.

- **Fluctuate Zero Lose [CNC factory]**

  Value: **0**, **1**

  Description: The value **0** indicates to detect the floating origin loss signal while **1** indicates not to detect the floating origin loss signal.

7.3.5 System Parameters

- **Color (0, 1, 2, 3) [User]**

  Value**: 0**, **1**, **2**, **3**

  Description: The value **0** indicates the default color while **1**, **2**, **3** indicate three specific colors.

- **Initial Interface (0, 1) [User]**

Value**: 0, 1**

Description: The value **0** indicates the program interface while 1 indicates the macro variables settings interface.

- **Status (0-7) [User]**

  Value**: 0** to **7**

  Description: The value indicates the number of the group that requires status monitoring.

- **Language [User]**

  Value**: 0**, **1**

  Description: The value **0** indicates **Chinese** while **1** indicates **English**.

- **Register mode [User]**

  Value**: 0**, **1**

  Description: The value **0** indicates registration through keyboard input while **1** indicates registration through USB files.

- **Y-axis [User]**

  Value: **1**, **-1**

  Description: The value **1** indicates to install Y-axis while **-1** indicates not to install Y-axis.

- **C-axis [User]**

  Value**: 5, -1**

  Description: The value **5** indicates to install C-axis while **-1** indicates not to install C-axis.

- **Diameter/Radius at X (1/0) [User]**

  Value: **0, 1**

  Description: The value **0** indicates radius programming while **1** indicates diameter programming.

- **Metric/Inches (1/0) [User]**

  Value: 0, 1

  Description: The value **0** indicates Metrics programming while **1** indicates Inches programming.

- **Power Off Protection [User]**

  Value: **0**, **1**

Description: The value **1** indicates to use power off protection while **0** indicates not to use power off protection.

- **Coord Display [User]**

  Value: **0**, **1**

  Description: The value **0** indicates command value while **1** indicates the actual value.

- **Show time [User]**

  Value: 0, 1

  Description: The value **0** indicates not to display system time while **1** indicates to display system time.

- **Pulse Type [User]**

  Value: **0**, **1**, **2**

  Description: The value **0** indicates single pulse, **1** indicates dual-pulse, while **2** indicates AB phase orthogonal pulses.

- **Metric Precision [User]**

  Value: **0**, **1**

  Description: The value **0** indicates three digits after the decimal point while **1** indicates four digits after the decimal point.

- **Inches Precision [User]**

  Value: **0**, **1**

  Description: The value **0** indicates three digits after the decimal point while **1** indicates four digits after the decimal point.

- **CF Card [User]**

  Value: **0**, **1**

  Description: The value **0** indicates to support CF card while **1** indicates not to support CF card.

- **Original Speed of MDI [User]**

  Unit: mm/min

  Value: **0** to **32767**, default value: **500**

  Description: The feed speed of the running program when no feed rate value is specified

under MDI.

- **Wear Out Accumulation [User]**

  Value: **0**, **1**

  Description: The value **0** indicates tool wear accumulation while **1** indicates no tool wear accumulation.

- **Workpiece Counting [User]**

  Value: **0**, **1**

  Description: The value **0** indicates that the command **M30** makes the completed workpieces accumulation while **1** indicates that the command **M64** makes the completed workpieces accumulation.

- **G94/G95 [User]**

  Value: **0**, **1**

  Description: The value **0** indicates the initial program mode is G94 while **1** indicates the initial program mode is G95.

- **Protection-Mode [User]**

  Value: **0**, **1**, **2**

  Description: The value **0** indicates not to enable the program editing protection. The value **1** indicates to enable the program editing protection. In this case, you cannot edit or create a program. The value **2** indicates no program display.

- **Check wear value in real time [User]**

  Value: **0**, **1**

  Description: The value **0** indicates not to check the wear value in real time while **1** indicates to check the wear value in real time.

- **Tool's offset only in Jog [User]**

  Value: **0**, **1**

  Description: The value **0** indicates **No** while **1** indicates **Yes**.

7.3.6 Axis Compensation (X-axis)

- **Reversed Gap [machine factory]**

  Unit: μm

Value: **0** to **65535**, default value: **0**

Description: The measured value in the workspace of the machine. This value can be set to **0** if bidirectional pitch compensation is used.

- **Pitch Comp. Type [machine factory]**

  Value: **0**, **1**, **2**, default value: **0**

  Description: The value **0** indicates no pitch compensation; **1** indicates unidirectional; **2** indicates bidirectional.

- **Number of Compensation [machine factory]**

  Value: **0** to 127 (**0** to **5000**), default value: **0**

  Description: The number of pitch offset compensation. For unidirectional compensation, the maximum number is **128**; for bidirectional compensation, the maximum number is **64**.

- **Index of Reference Point [machine factory]**

  Description: The sequencing index of the reference point in the offset table.

  Sequencing rule: Based on the position of each compensation point in the coordinate system, sequencing from the negative direction to the positive direction and starting from **0**.

  Example:

  a. If the compensation points are **-180**, **-120**, **-60**, **0**, and the reference point is **0**, then the index of the reference point is **3**.

  b. If the compensation points are **0**, **-60**, **-120**, **-180**, and the reference point is **0**, then the index of the reference point is **0**.

- **Pitch Compensation Point Interval [machine factory]**

  Unit: um

  Value: **0** to **4294967295**, default value: **0**

  Description: The distance between two adjacent compensation points.

- **Offset [machine factory]**

  Unit: μm

  Value: **-32768** to **32767**, default value: **0**

  Description: The absolute compensation.

  Offset= machine command value - machine actual value

The difference value between the actual value moved on the axis and the command value, that is, the value the tool need to move further or less to make the tool move to the accurate position on the axis.

For bidirectional pitch compensation, you need to enter correct positive pitch offset value first, and then enter negative pitch offset value. The compensation value (positive and negative) must be entered based on the position of the compensation point in the machine coordinate system and based on the axis direction. For example, for the two points **(-150** and **-100**), you need to enter the pitch offset value for **-150** first and then for **-100**. In this case, the offset index for **-150** is on top of **-100**.

Example: If there are ten compensation points, and bidirectional pitch compensation is used, then **0** to **9** are positive compensation values, and **10** to **19** are negative compensation.

Example:

If the command machine coordinate value is **100** mm and the actual machine coordinate value is **100.01**mm,then the offset is **-10** μm (*100 - 100.01 = -0.01mm = -10 μm*).

You need to pay attention to the symbols of the axis value when calculating offset value. For example, if the command machine coordinate value is **-100** mm and the actual machine coordinate value is **-100.01**mm, then the offset is **10** μm (*-100 - (-100.01) = 0.01mm = 10 μm*)

The following information describes the pitch compensation method:

Known conditions:

a. X axis, the reference value **0**

b. Homing to reference in the positive direction

c. positive software limit: 2000 (2mm)

d. Negative software limit: -602000 (-602 mm)

e. Compensation interval within travel: 40 mm

f. Compensation point: (600/40) + 1 = 16

g. Compensation point coordinate value (from left to right): -600, -560, -520, -480, -440, -400, -360, -320, -280, -240, -200,-160, -120, -80, -40, 0

If the reference point value is **0,** then the index of reference point is **15**.

The procedure for pitch offset measurement is as below:

%0110: File header

G92 X0 Y0 Z0: Set temporary coordinate (should start from the reference point)

WHILE [TRUE]: Infinite loop (endless loop)

G91 X1 F2000: Positively move 1 millimeter on the X-axis

G04 P4: Pause for four seconds

G91 X-1: Negatively move 1 millimeter on the X-axis and return to the measurement position, and then clear the reverse gap

: The measurement system is cleared.

G04 P4: Pause for four seconds, and the measurement system records data.

M98 P1111 L15: Activate the subprogram for negative movement 15 times, program No.: 1111

G91 X-1 F1000: Negatively move 1 millimeter on the X-axis

G04 P4: Pause for four seconds

G91 X1: Positively move 1 millimeter on the X-axis and return to the measurement position, and then clear the reverse gap

G04 P4: Pause for four seconds and the measurement system records data.

M98 P2222 L15: Activate the subprogram for positive movement 15 times, program No.: 2222

ENDW : End of the loop program

M30: Stop and return

%1111: Subprogram for negative movement on the X-axis is 1111.

G91 X-40 F1000: Negatively move 40 mm on the X-axis

G04 P4: Pause for four seconds and the measurement system records data.

M99: End of the subprogram

%2222: Subprogram for positive movement on the X-axis is 2222.

G91 X40 F500: Positively move 40 mm on the X-axis

G04 P4: Pause for four seconds and the measurement system records data.

M99: End of the subprogram

When measuring pitch offset, you need to set the reverse gap to **0** firstly. Before changing the measurement direction, you need to clear the reverse gap of the coordinate.

The offset [**0**] to [**15**] are the offset values of the compensation points (-600, -560, -520, -480, -440,-400, -360, -320, -280, -240, -200,-160, -120, -80, -40, 0) on the X-axis, moving in the positive direction.

The offset [**16**] to [**31**] are the offset values of the compensation points (-600, -560, -520, -480, -440,-400, -360, -320, -280, -240, -200,-160, -120, -80, -40, 0) on the X-axis, moving in the negative direction.

7.3.7 Graphics

- **Graphics Amplification Coefficient [user]**

  Default value: **1.0**

  Description: The amplification coefficient for graphic display.

- **External Diameter [user]**

  Default value: **60.0**

  Description: The actual external diameter of the sample workpiece.

- **Internal Diameter [user]**

  Default value: **0.0**

  Description: The actual inner diameter of the sample workpiece.

- **Workblank Length [user]**

  Default value: **120.0**

  Description: The actual length of the rough workpiece.

- **Internal End Surface [user]**

  Default value: **-120**

  Description: The actual inner end surface of the rough workpiece.