ADT-DK300A/400A

User's Manual

Copyright

Adtech (Shenzhen) Technology Co., Ltd.

Version History

Item No.	First uploaded	Version No.	Pages	Compiled by	Typeset by
BZ001M056A	2012-10-9	A0101	40	Shi Tingliang/ Tang Xiaobing	

Revision			
Date	Version/Pag	Result	Confirmed by
2012-9-18			Shi Tingliang

Remark:

We have collated and checked this Manual strictly, but we can't ensure that there are no error and omission in this Manual. Due to constant improvement of product functions and service quality, any products and software described in this manual and the content of the manual are subject to changes without prior notice.

Contents

1. Operation Introduction

1.1 LCD and key panel

Fig. 3.1 DK300A Operating Panel Diagram

Note:

Press the submenu buttons to perform the operations of submenus.

Manual axis moving and edit & input are composite. It has different definitions according to the modes.

System working mode switch section is used to switch working modes, which can improve the security and system performance. Handwheel and single step mode are switched with Repeat button.

LCD unit

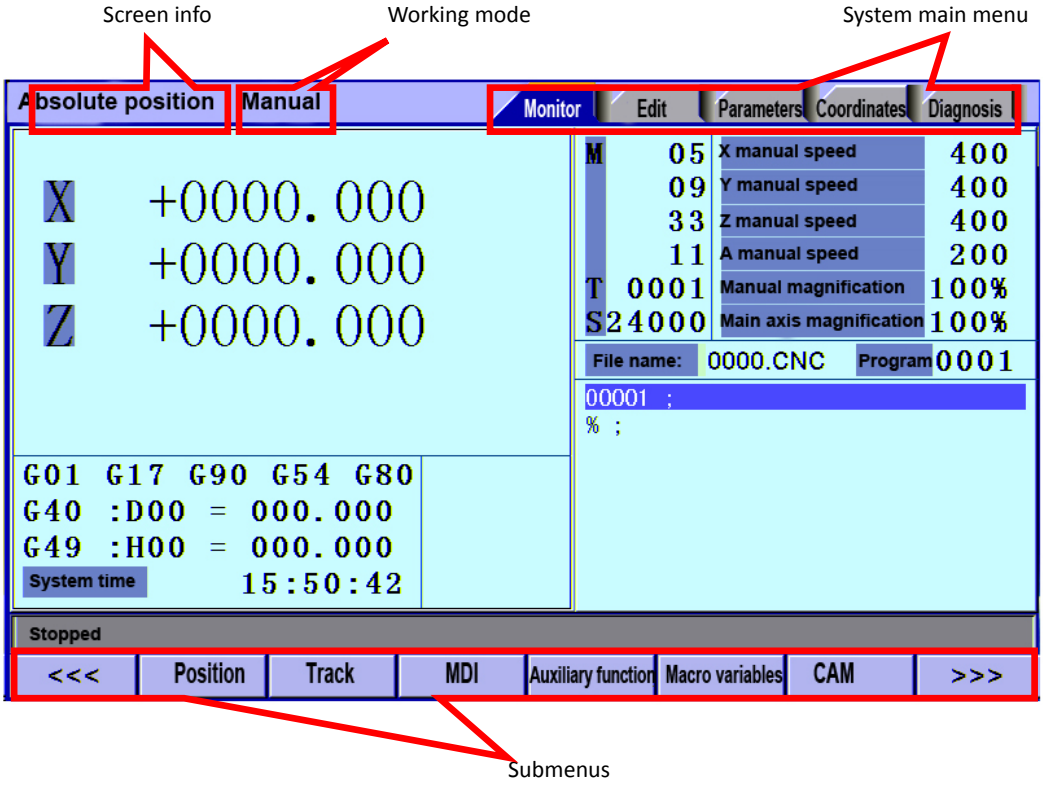


Fig. 3.2 DK300A LCD Screen Diagram

Note:

Screen info shows the information of current window

Working mode info shows currently selected working mode

System main screen shows current main screen.

The submenu options are used to switch submenus with left triangle, F1~F6 and right triangle. The right arrow is used to turn pages, and the left arrow is used to close the submenus in next level and previous menu.

1.2 System menus

DK300A system uses cascading menu structure. You can press the following keys to operate the menus.

Press a key to show the corresponding content in the bottom of the LCD.

Key in the left  : Return to previous menu

Key in the right  : Turn pages to show other menus of same level

The main menus of the system include [运转 MONITOR], [编辑 PROG], [参数 SYSTEM], [坐标 COORD] and [诊断 DGNOS]. Each main menu contains several submenus, which are shown below:

1.3 Operating keys

The keys of DK300A system are defined below:

Key	Figure	Purpose
[复位/RESET]	复位 RESET	Cancel alarm, reset CNC
Address/number keys	0 *, 1 A- , ... 9 A+) A U, G B ..... Z K	Enter letters, numbers, etc.
[EOB], [取消/CAN]	EOB, 取消 CAN	Confirm or cancel operation
[EOB], [取消/CAN], [删除/DEL]	EOB, 取消 CAN, 删除 DEL	Program edit (insert, delete, modify)
Mode switch key	自动 AUTO, 手动 JOG, 录入 EDIT 手轮 MPG / STEP 单步, 归零 HOME	Select operating mode
Cursor moving key	+%, -% ⊕, ⊖	Four keys are available: Up/Down: adjust ration, move cursor between subsections; Left/Right: move cursor to left/right

Key	Figure	Purpose
Page key		Up/Down: Turn pages
Menu keys		Select the menus
Principal axis positive rotation		Press it to rotate the principal axis positively, and press it again to stop rotating
Principal axis reverse rotation		Press it to rotate the principal axis reversely, and press it again to stop rotating
Coolant		Coolant on/off
Lubricant		Lubricant on/off
[跳选/BDT]		Block delete on/off
[单段/SBK]		Single block function on/off
[暂停/PAUSE]		Pause automatic running
[启动/START]		Start automatic running

1.4 Manual operation

1.4.1 Returning to reference point manually

The engraving machine tool has specific mechanical position, which is called as reference point and for tool exchange and coordinates setting. Generally, when the power supply is connected, the tool should be moved to the reference point. This operation is also called as home operation, which will make the CNC system confirm the origin of machine tool.

The home operation includes program and mechanical mode:

For program home, the action completes when the coordinates of machine tool are 0, and won't check whether origin switch is in position;

For mechanical home, the external home sensor switch is used to locate the origin of the machine tool; two checking modes are available:

With the external sensor switch, the home operation completes when the sensing is successfully repeatedly.

The external sensor switch is used as deceleration switch, and the servo home is enabled as home signal after sensing and then the sensing stops.

You can set the "Home mode" in [Parameter] [Comprehensive Parameter], and move to comprehensive home mode (36# parameter) EOB to select. You can also press [SBK] key in home mode to switch among "Mechanical – Program – Mechanical…" quickly. This method doesn't conflict with parameter setting. You can select accordingly. To use servo home as the home signal, you need to set "Axis phase Z home enable" to "1" in [Parameter] [Axis Configuration] in mechanical home mode, and the setting will take effect in next home checking.

Several methods are available for tool returning to reference point and the steps follow:

(1) Each axis returns to reference point separately

Press the mode switch key [归零 HOME] to select home operation;Press the composite key [Z- 3 =], [Y- 2 #], [X- 4 [], [A- 1 ,] in the numbers section to return the corresponding axis to reference point.(2) The axes return to reference point simultaneously

Press the mode switch key [归零 HOME] to select home operation;Press the [启动 START] key to return Z axis to reference point, and other axes return to reference point simultaneously. The automatic home sequence can be configured in the parameters. (3) Reset machine tool position

Press the mode switch key [归零 HOME] to select home operation;In [Absolute Position] and [Coordinate System] screen, press [X], [Y], [Z], [A] respectively to show the value of corresponding axis position, and then press the [Cancel] key to reset the machine tool position of current axis, i.e. current point is used as machine tool origin. After this operation, the system considers it as a home action. Therefore, when the program is running, the alarm of not home won't occur. If you press by mistake, it will switch the screen and cancel selection automatically.(4) Reset relative position manually

Press the mode switch key [手动 JOG] to select manual operation;

In [Relative Position] and [Coordinate System] screen, press [X I], [Y J], [Z K], [A U] respectively to show the value of corresponding axis, and then press the [取消 CAN] key to reset the relative position of current axis.**Note**

The tool also can return to reference point according to program instruction, i.e. returning to reference point automatically.

Caution:

Generally, the system will perform home operation after connecting the power supply. If the power fails while the machine tool is moving, the system also will return to reference point when the power supply is connected again. Return to Z axis to prevent tool and workpiece from colliding, and damaging tool, workpiece and clamp.

1.4.2 Continuous feeding manually

Press the keys on the operation panel or handwheel to move the tool along every axis.

The operation follows:

Press the mode switch key [手动 JOG] to select manual operation;

(2) Press composite keys [], []; [], []; [], []; [], [] in numbers area to move the tool along selected axis.

In manual mode,  key can be used to switch the manual speed and rapid traverse speed. The rapid traverse speed of every axis depends on comprehensive parameter 009-012 (rapid traverse speed setting). After switching to rapid traverse speed, the manual speed of the position interface will be highlighted, while the actual speed of the position interface is sampled from the moving speed of current axis. This value can truly reflect the moving speed of current axis (unit: mm/min);

☞ **Note:**

Only single axis motion is available in manual mode.

1.4.3 Single step feeding

Single step mode is similar to manual mode, the operations are same, but only moves a specified pulse increment every time press the key.

The specific operation follows:

(1) Press the mode switch key [] (this key is composite, and you can press it repeatedly to switch the modes) to select the single step operation;

(1) Press composite keys [], []; [], []; [], []; [], [] in numbers area to move the tool for a fixed distance along the selected axis. This distance is controlled by four rates (1.000, 0.100, 0.010, 0.001) (unit: mm). To select pulse increment, press  and  key in the [Position] interface.

1.4.4 Handwheel feeding

In handwheel mode, rotate the handwheel to make the machine perform single step or continuous motion. Determine the feed by testing the handwheel signal of the handheld box. In handwheel mode, the feeding axis and feeding unit are determined by the axis selection signal of the handheld box.

The handwheel feeding step follows:

Press the mode switch key [

] to select handwheel operation;

(2) Rotate the dip switch on the handwheel to select handwheel axis (X, Y, Z, A);

(3) Rotate the increment dip switch on the handwheel to select the moving amount (0.1, 0.01, 0.001);

(4) Rotate the handwheel to move the machine tool. The tool moves certain distance every time you rotate the handwheel for a scale. (For example, if you select X axis in step (2) and select 0.01 in step (3), the tool moves 0.01mm every scale). Rotate the handle continuously to move the machine tool on this axis continuously.

☞ **Note:**

The handwheel feeding mode controls only one coordinate axis every time; the faster the handwheel rotates, the faster the machine tool moves.

1.4.5 Manual auxiliary function operation

Coolant on/off

 In handwheel/single step/manual mode, press this key to switch on/off the coolant.

Key indicator: No matter in what mode, the key indicator is on if only the coolant is on, or else the indicator is off.

Lubricant on/off

 In handwheel/single step/manual mode, press this key to switch on/off the lubricant.

Key indicator: No matter in what mode, the key indicator is on if only the lubricant is on, or else the indicator is off.

Principal axis positive rotation/stop

 In handwheel/single step/manual mode, press this key to rotate the principal axis positively and press it again to stop the axis.

Key indicator: No matter in what mode, the key indicator is on if only the principal axis is positive rotating, or else the indicator is off.

Principal axis reverse rotation/stop

 In handwheel/single step/manual mode, press this key to rotate the principal axis reversely and press it again to stop the axis.

Key indicator: No matter in what mode, the key indicator is on if only the principal axis is reverse rotating, or else the indicator is off.

General instructions for manual operation keys

 ,  ,  and  are available in handwheel, single step and manual mode;

When the principal axis is rotating, press the reverse rotation key, the principal axis will stop first, and rotate in reverse direction after pressing it again.

When auxiliary output is on, if the system is switched to other modes, the output is unchanged; you need to press "Reset" key to switch it off, execute the corresponding M code in automatic mode or execute the corresponding M code in MDI interface to turn off the output;

When the principal axis is positive/reverse rotating and execute M04/M03 directly, the system first stops positive/reverse rotating and then execute M04/M03 instruction;

Positive/reverse rotating of principal is stopped while emergency stop, and other outputs can be set according to system parameters.

1.4.6 Working coordinate system settings (tool setting)

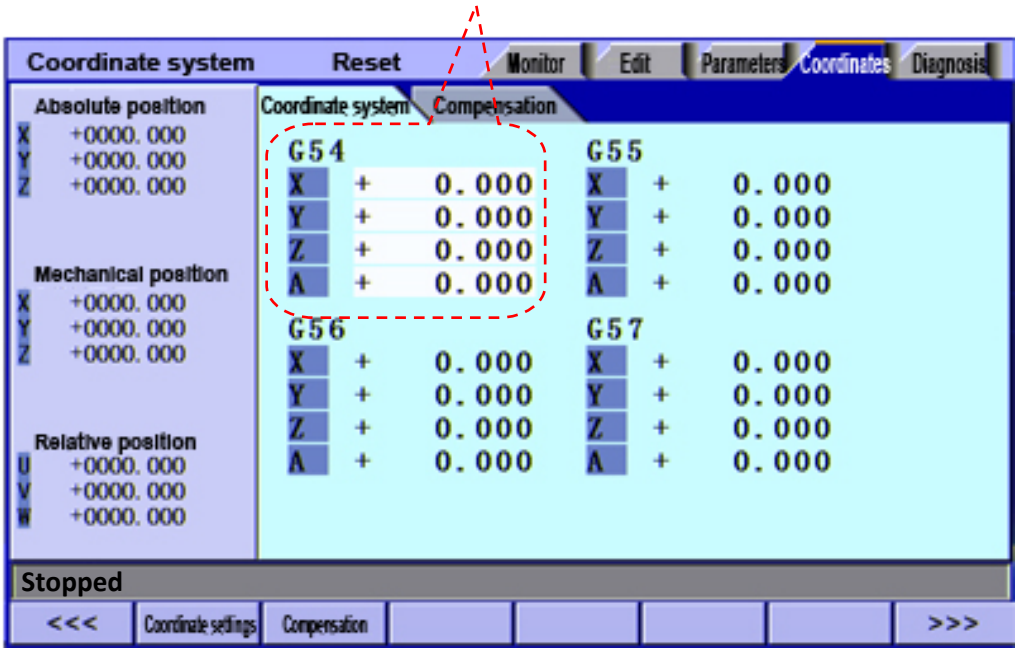
Tool setting is the main operation and important skill during CNC processing. Under certain conditions, tool setting precision can determine the processing precision of parts, and the tool setting efficiency also affects the CNC processing efficiency directly. DK300A and DK400A realize tool setting through G54-G59 coordinate system.

Workpiece coordinate system setting method

1.4.6.1 Enter current position to specified coordinate system directly (all axes)

1. Press 

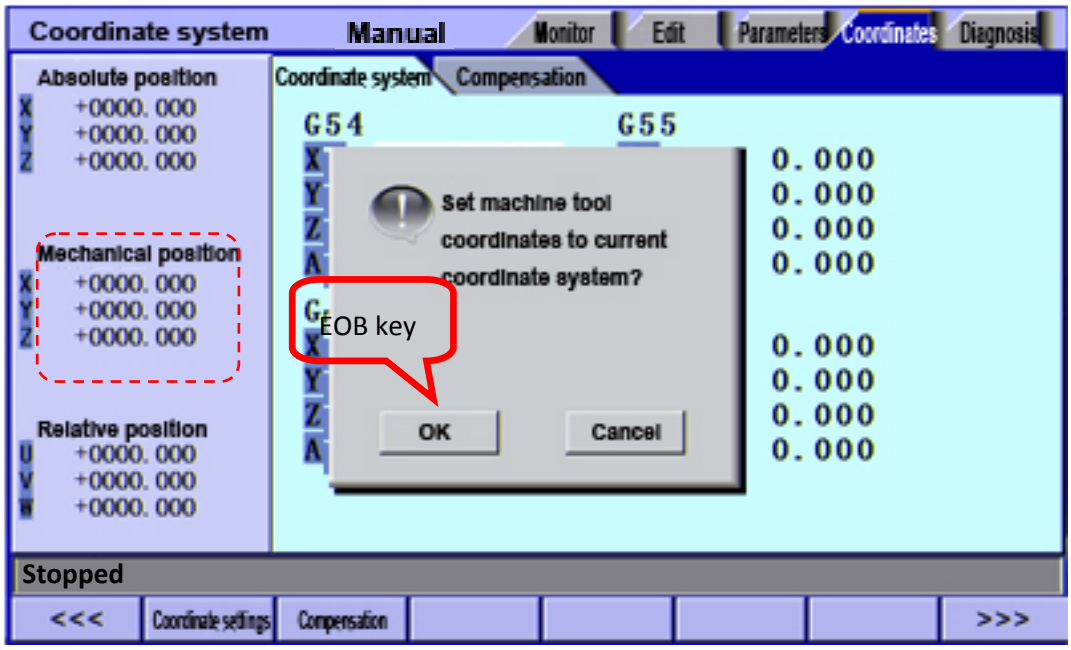
to enter the coordinate system interface. The background of select coordinate system is white.



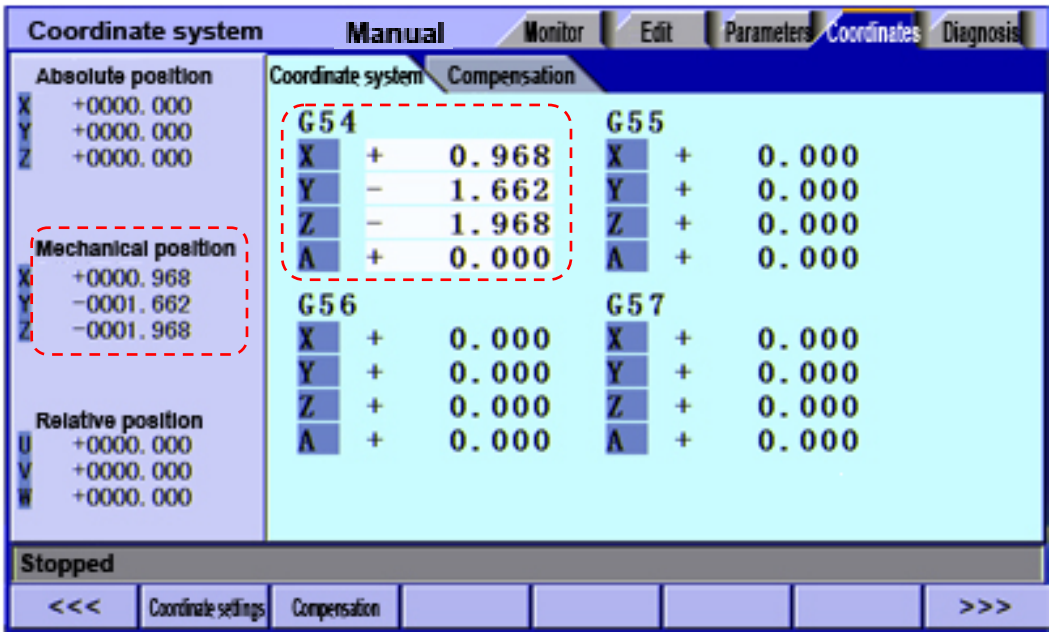
2. Then, press , to select the coordinate system, e.g. G54.

3. Press 手动 JOG to select manual mode, and press , , , , , , , to move the axis to the origin of the coordinate system.

4. Press EOB to show a dialog box, and press "EOB" to set current coordinates as the zero point of the program.



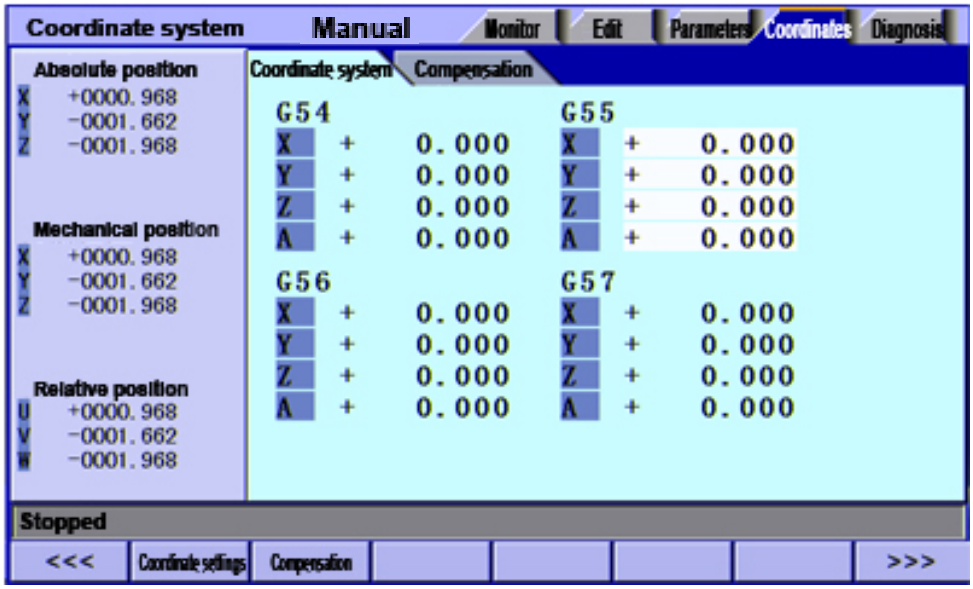
5. After setting the coordinate system, copy the mechanical position values to the axis value in the dotted box as shown in the figure below.



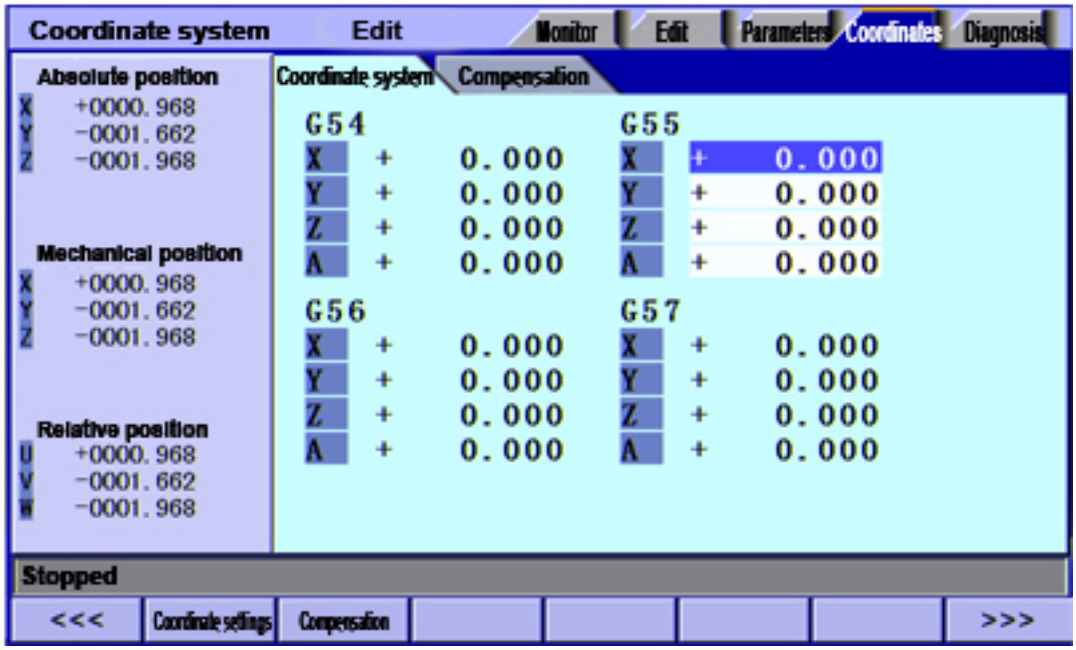
1.4.6.2 Coordinate axis settings

1. Press 坐标 COORD to enter the coordinate system interface.

2. Then, press  ,  to select the coordinate system, e.g. G55.

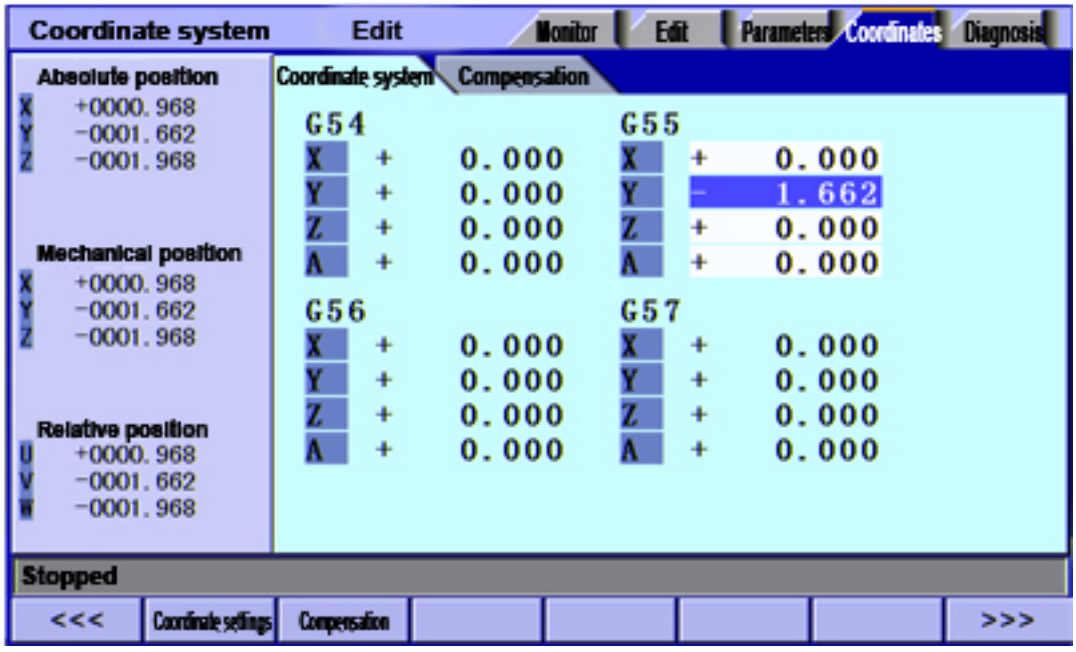


3. Press 录入 EDIT to select the edit mode.



4. Select an axis number; you can press +% , -% to move the cursor and select X I, Y J, Z K, A U; the selected coordinate parameters are shown on blue background; for example, to select axis X, press X I, the coordinates of corresponding axis number will be shown, and prompt the values to be inserted.

5. Press "EOB" key to finish the setting of currently selected coordinate system, as axis Y in the figure below



1.4.6.3 Modify coordinate value

In addition to above two methods, you can also press the keypad to set the coordinate system in the following steps:

1. Press 坐标 COORD to enter the coordinate system interface.

2. Then, press ⊕, ⊖ to select the coordinate system, e.g. G56

3. Press 录入 EDIT to select edit mode.

4. Press +%, -% to select a desired axis; to set Z axis of G56 to -10.5, press +%/-% to select axis Z, as shown in Fig. 4A, then, press —/+, 1 A-, 0 *, .><, 5 ~], in sequence, and then press EOB to enter -10.5 to Z axis coordinate system, as shown in Fig. 4B.

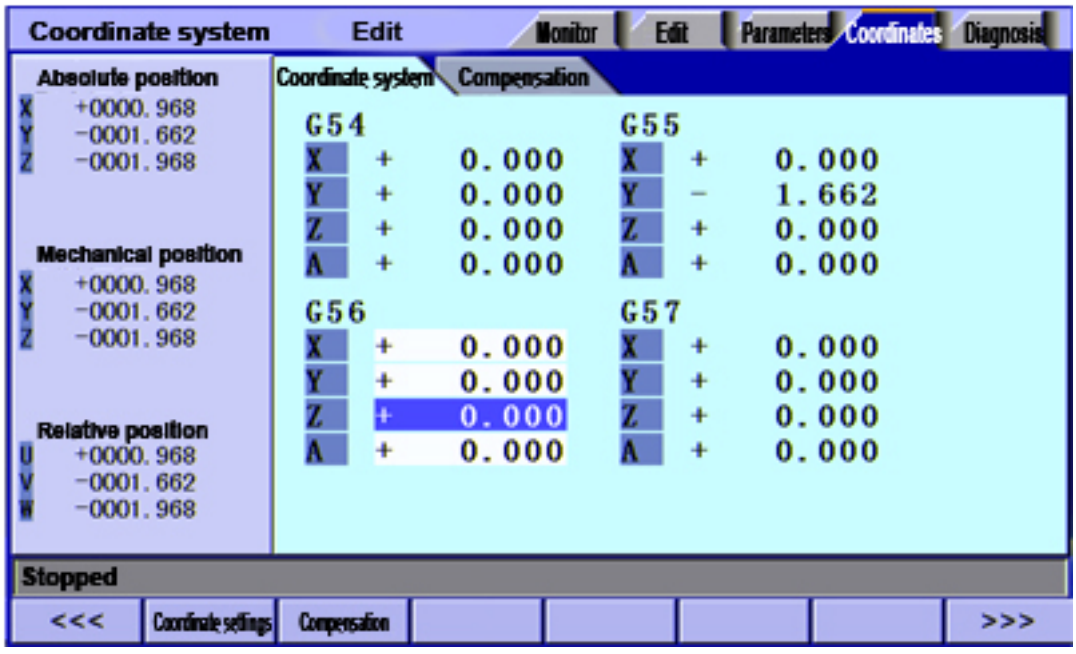


Fig. 4A

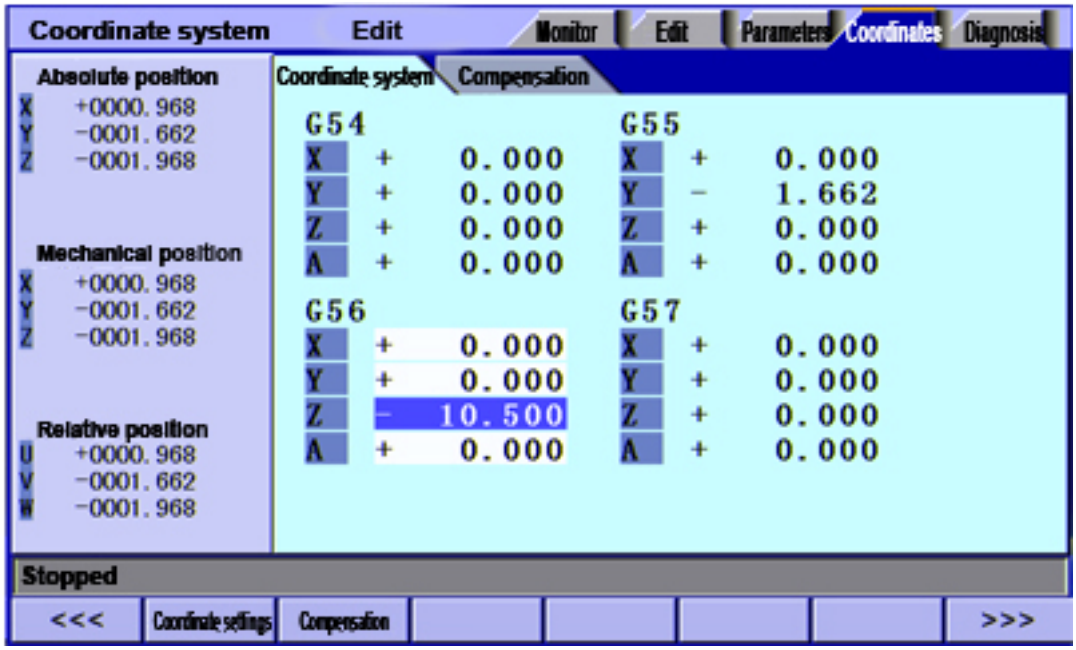


Fig. 4B

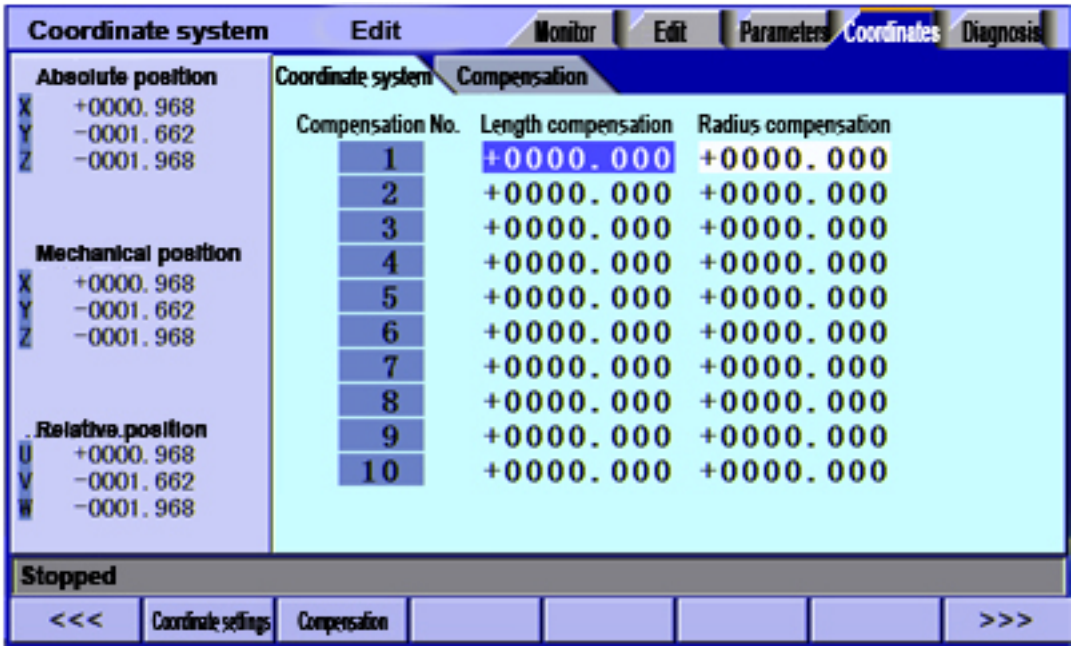
Edit other axes in the same method.

1.5 Data settings

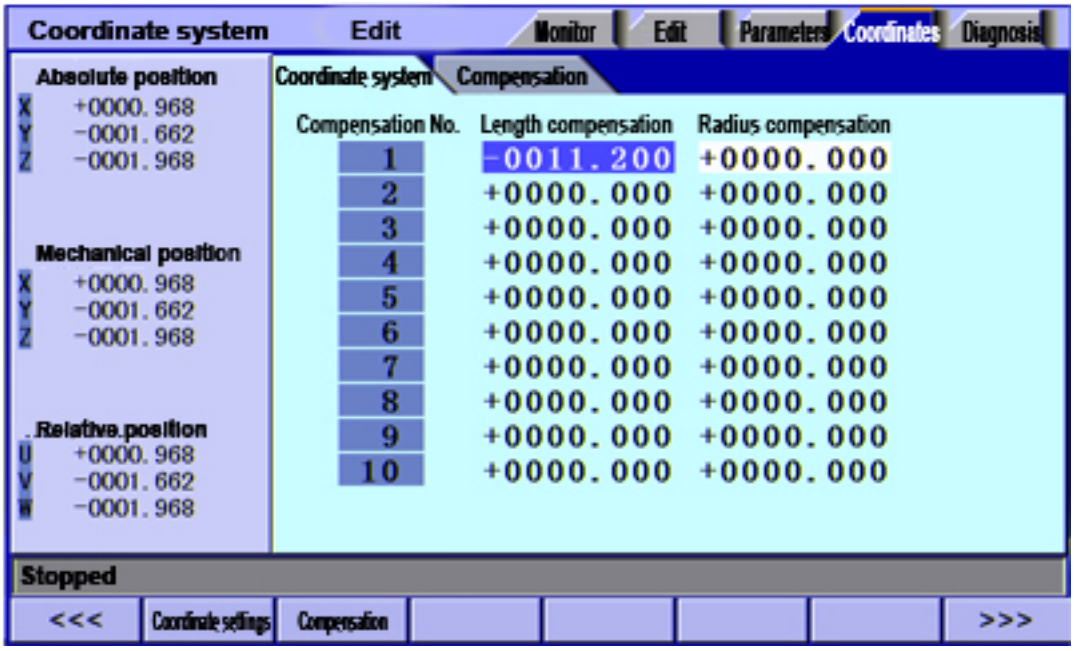
1.5.1 Tool compensation data setting

The tool compensation parameters can be set as follow:

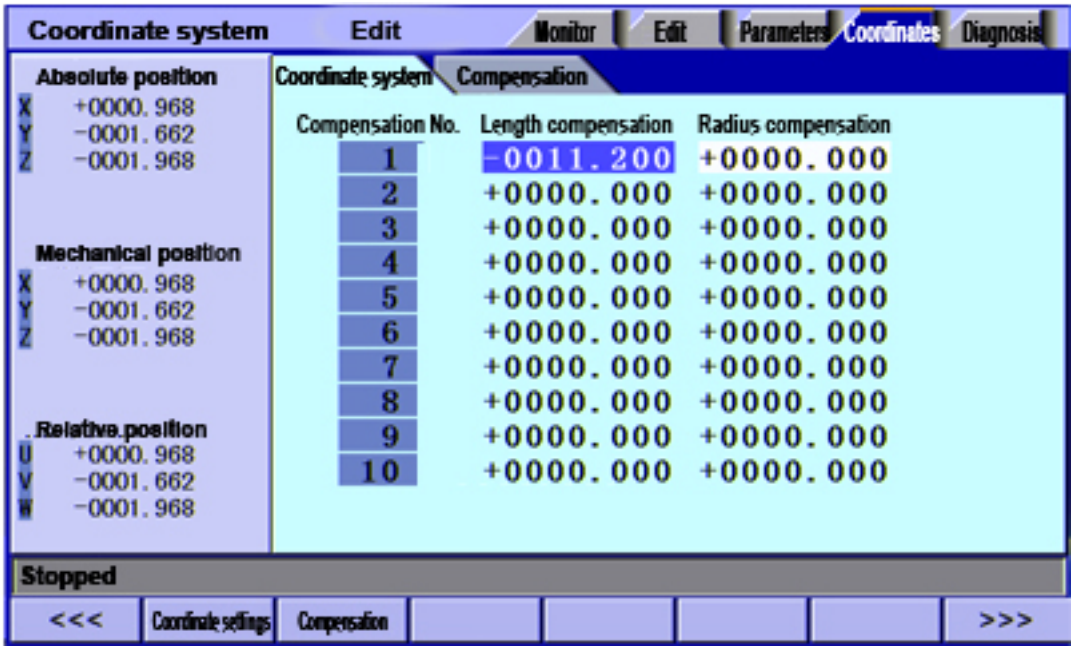
- Press 录入 EDIT to select the edit mode;
- In the main menu, press [坐标 COORD], and then press submenu [Compensation] to enter tool compensation parameter setting interface;



Press +% / -% to select the parameter, e.g. compensation No. 1, enter -11.2 and press EOB to modify the parameters where the cursor is located, as shown in the figure below



Enter tool length compensation quickly: select compensation number, press Z K and then press EOB to insert Z axis coordinates into length compensation, as shown in the figure below

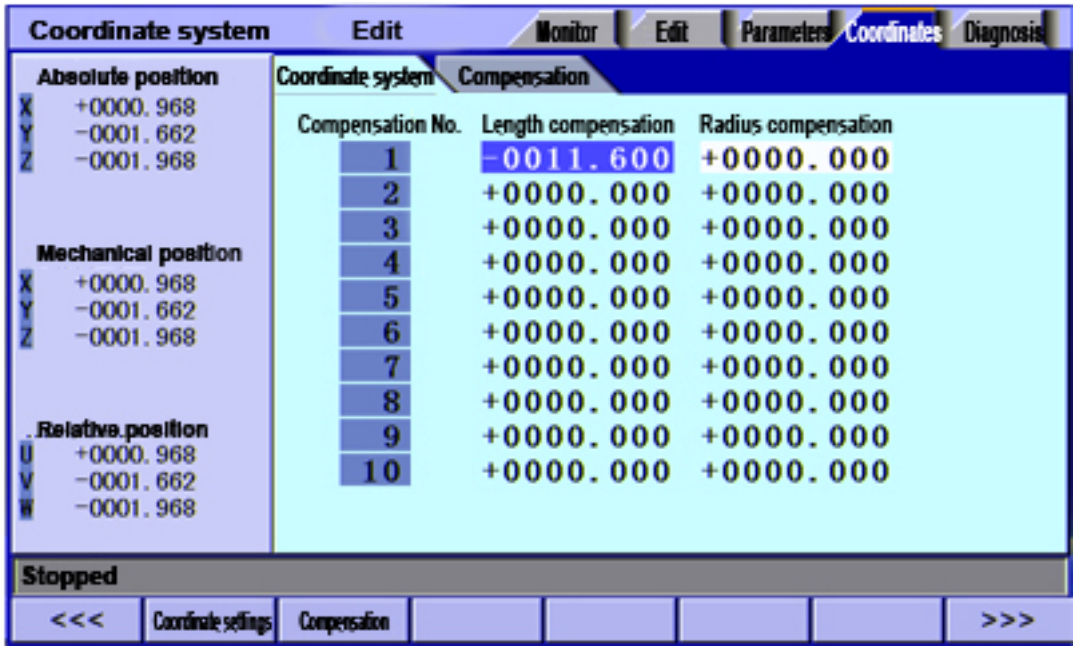


Caution

1. The value entry dialog box has two input methods, i.e. direct assignment and increment assignment. For direct assignment, the entered value is assigned to specified parameter directly; for increment assignment, the entered value is added to specified parameter and then assigned to the parameter;

2. The increment input box and direct input box have a prompt symbol in the left respectively, '=' indicates direct input, '+' indicates increment input, and the default mode is direct assignment. To change to increment assignment, press the "UP/DOWN" key

3. The increment input is shown above, 1# compensation is -11.2, to add -0.4, the operation follows: select 1# compensation, enter -0.4, and then press  **to select the mode, "+" is to add the new value to original value, and "=" is to replace with the new value directly.** Then, press  to finish the modification of the compensation, as shown below.



1.5.2 System parameter setting

The system parameters can be modified as follow:

- Press 录入 EDIT to select the edit mode;
- In the main menu, press [参数 SYSTEM] to enter parameter setting interface;
- Then, press the submenu key to select the parameter type (comprehensive, management …);
- Press ◀, ▶, +%, -% to move cursor to select the parameter, enter the value and then press [EOB] to modify the parameter where the cursor locates.

To change the comprehensive parameter 1 to 10, select edit mode, press the direction keys to select 1# parameter, and press 1 A-, 0 *, EOB in sequence to finish the modification. Fast system operation

The value entry dialog box has two input methods, i.e. direct assignment and increment assignment. For direct assignment, the entered value is assigned to specified parameter directly; for increment assignment, the entered value is added to specified parameter and then assigned to the parameter; the increment input box and direct input box have a prompt symbol in the left respectively, '=' indicates direct input, '+' indicates

increment input, and the default mode is direct assignment. To change to increment assignment, press the "  " key.

In [Monitor] interface, press  in the [Position] menu to call out G code O program number box to switch to O program block directly. Press  to confirm, or press  to return.

In [Monitor] interface, press  in the [Position] menu in manual mode to trim the main axis rotation; press and hold the key accumulate the rotation value directly. Note: This function can be operated only after the main axis is opened, and the current speed of the main axis shouldn't be 0; if the speed is 0, set a non-zero rotation in MDI mode, and then perform the operation. In [Monitor] interface, press  in the [Position] menu in manual mode to adjust the manual magnification. In automatic mode, the automatic magnification is adjusted. Note: if additional panel is enabled, this operation will be covered by the magnification of the additional panel automatically, that is, this operation is invalid.

2. Automatic operation

The machine tool moving according to prepared program is called as automatic operation. The automatic operation modes of DK300A system follow:

Memory operation, MDI operation, USB disk DNC operation

2.1 Automatic operation

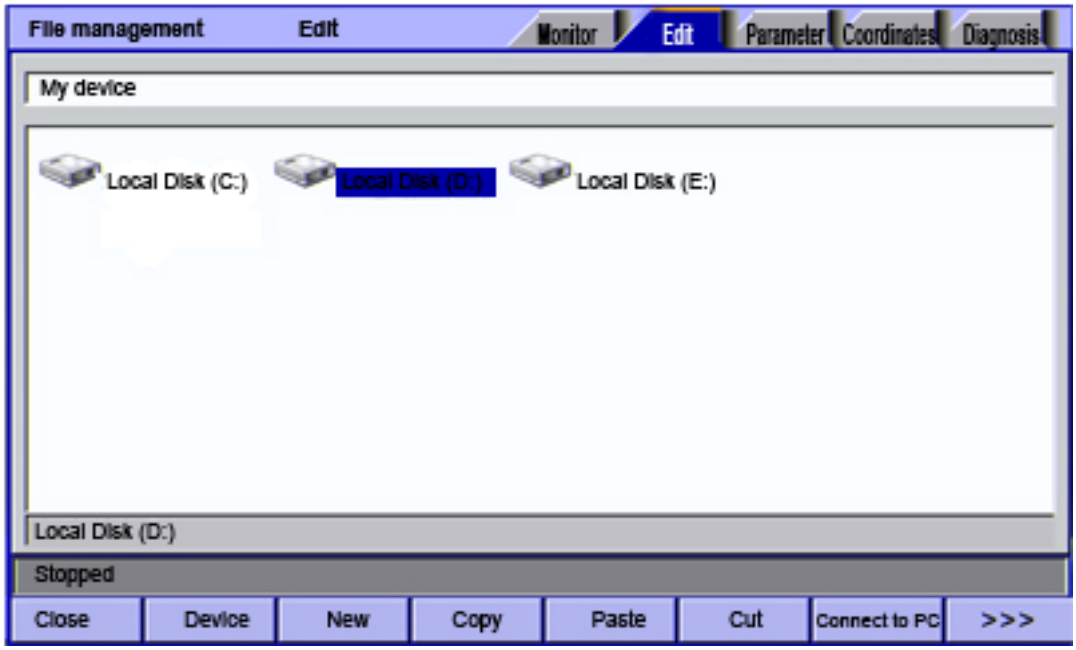
The machine tool can operate according to the program in DK300A memory, which is called as memory operation.

The program is prestored in the memory. Select and load a program with the operation panel and press the "启动 START" key to start the automatic operation. Then, press "暂停 PAUSE" key to pause, press "启动 START" key again to resume the operation, and press "复位 RESET" during operation to stop the program immediately.

The step of memory operation follows:

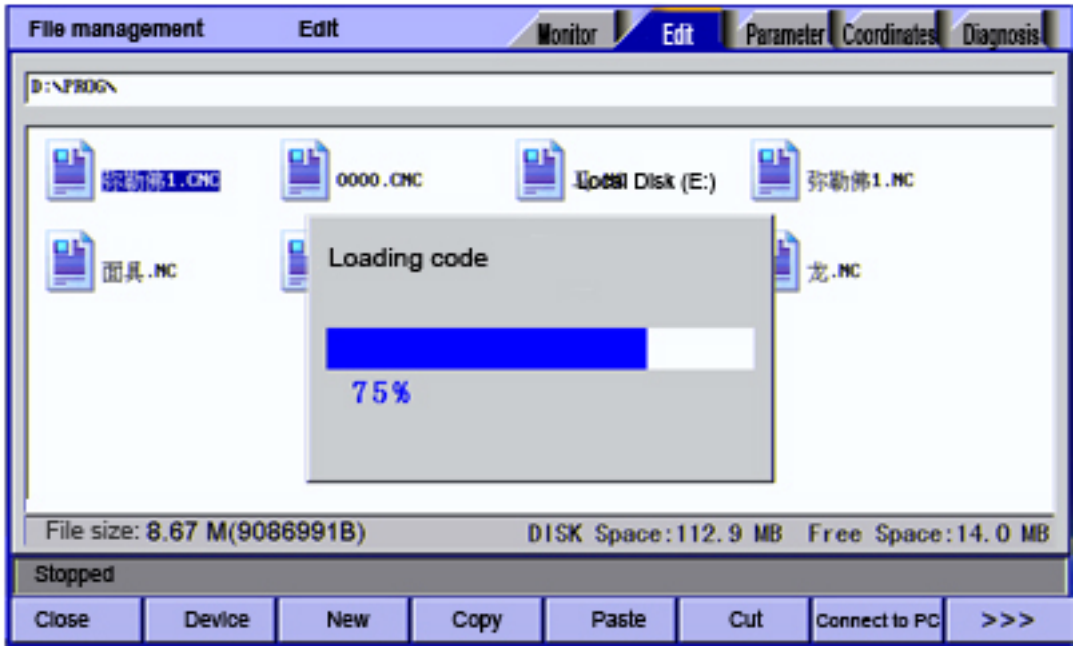
Save the program in the memory (see 4. Program saving & editing for details);

Select [Edit], [File] in the menu or press [File] on the panel to enter file operation interface;



Disk C is YAFS file system disk, which is dedicated system disk. Disk D is FAT data disk, which saves processing files and system settings as well as PC interactive data. Removable USB disk saves external user data.

(1) Press  , to move the cursor, press [EOB] to select a program and load the file into the work area; as shown in the figure below



(5) Press mode selection key [

] to switch to automatic mode;

(6) Press the [

] key to run the program, and the indicator is on.

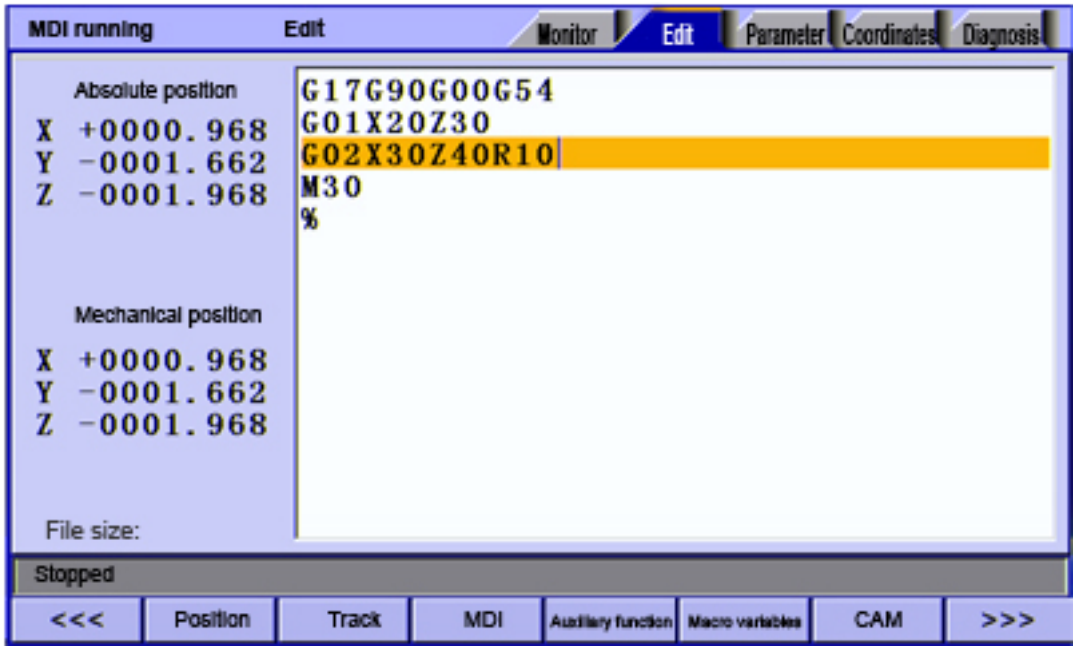
2.2 MDI operation

In [Monitor] interface, switch to [MDI], enter the program with keypad and make the machine tool operate according to the program. The program block isn't saved in system memory, and can't be preserved upon power failure. This is called as MDI operation and the step follows:

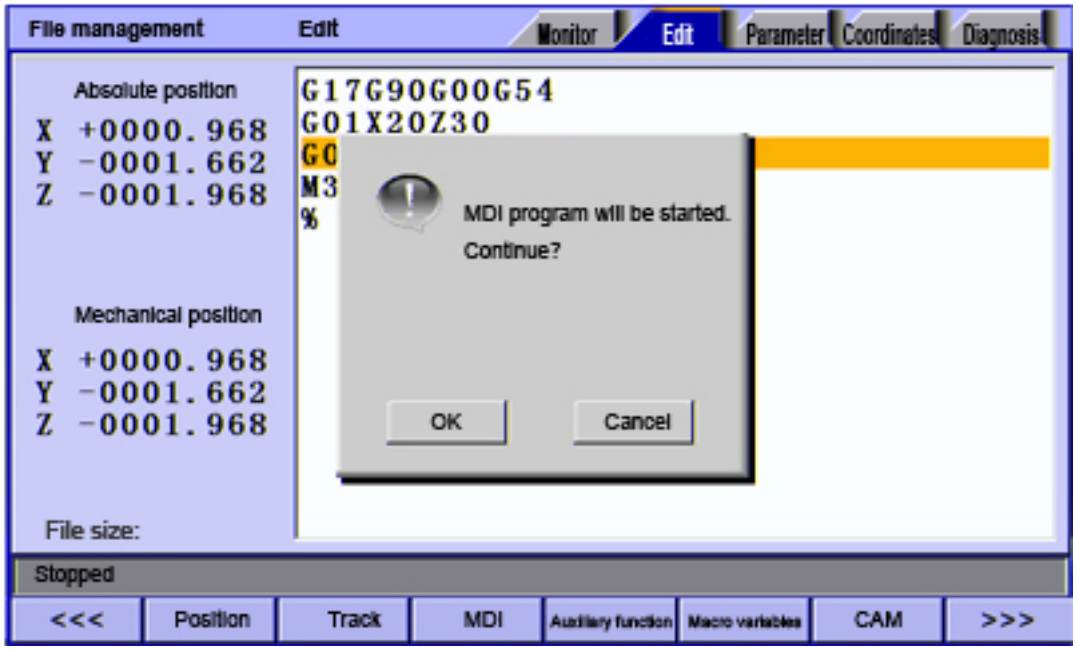
(1) Press mode selection key [Edit];

Select [Monitor], [MDI] in the menu to enter MDI interface;

Enter program block instruction manually;



(5) Press [Start], [EOB] to start executing the program block, as shown below.



2.3 USB disk DNC

The program read from external USB disk can operate the machine tool without saving in CNC memory. This operation is called as USB disk DNC operation.

The step of USB disk DNC operation follows:

(1) Insert the USB disk;

(2) Select [Monitor], [File] in the menu to enter file operation interface;

(3) Select USB disk and press [EOB] to enter;

(4) Move cursor to select a file in the disk;

(5) Press [EOB] to load the file into work area (system buffer);

(6) Press mode selection key [Auto];

(7) Press the [Start] key to run the program, and the indicator is on.

Caution

The system won't record the USD disk path. If power failure occurs during DNC processing, the program info will be lost when the power supply is connected again.

2.4 Speed rate adjustment

2.4.1 Feeding rate

In automatic mode, press 

key in [Position] interface to adjust the feeding rate; Press the key once to increase or decrease by 10% (10%-150%).

2.4.2 Manual rate

In manual mode, press 

key in [Position] interface to adjust the manual rate; Press the key to increase or decrease by 10% (10%-150%). If you press the 

key and 

key, you can adjust the fast forward rate by 10% (10%-150%).

2.4.3 Principal axis rotation

In automatic or manual mode, press the  key to adjust the principal axis rotation by 100r/min. The maximum rotation is set by the principal axis parameters in the system and the minimum rotation is 16r/min. If you press and hold the key for three seconds, the value will be increased or decreased quickly.

2.5 SBK function

In automatic mode, press [

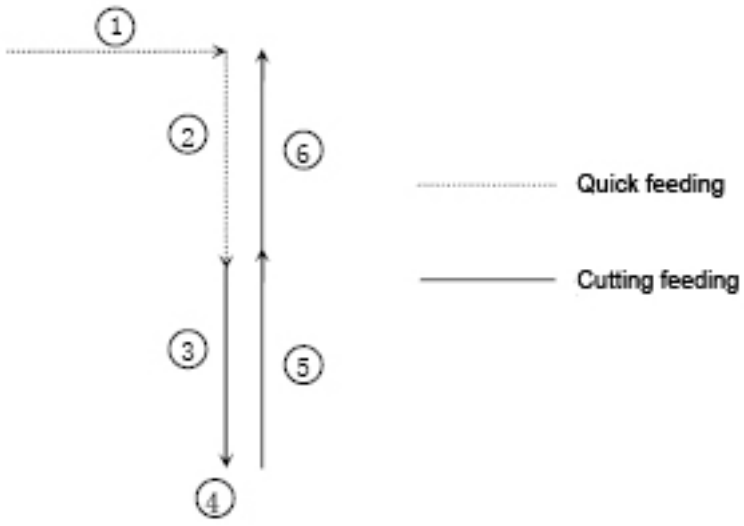
] to start the SBK function. Current program block stops after executing; press [

] again and next block stops after executing. The SBK mode allows checking the program block by block.

Caution:

① In G28-G30, single block also can be stopped at the center point;

② The stop points of single block in fixed circle are ①, ②, ⑥ in the figure below; when the single blocks of ①, ② stops, the feeding pauses and the pause indicator is on.



2.6 BDT function

In automatic mode, press [

] to start the BDT function, which will make the block instructions in the line after '/' in the program invalid.

2.7 Stopping automatic operating

Two methods are available to stop automatic operating, i.e. enter stop command where the program will stop (M00, M01) and press the 

key on the operation panel to stop the machine tool.

2.7.1 Program stops

After executing the block with M00 or M01, the automatic operating stops, which is same to single block stop, and all mode information is saved. Start with CNC and the automatic operation can be started again.

After processing a part, the automatic operation stops.

2.7.2 Program ends

After executing the block with M30, the automatic operating stops, changes into reset state, and returns to program start.

2.7.3 Feeding pause

During automatic operation, press the [

] key on the operation panel, the automatic operation pauses and the indicator is on; press [

] again to continue operating the machine tool and the pause indicator is on.

2.7.4 Reset

During automatic operation, press the [

] key on the operation panel and the system stops immediately. Here, [

] has the same function as emergency stop button.

3. Safe operation

3.1 Emergency stop

Press the emergency stop button on the machine tool, which will stop immediately, and all outputs such as principal axis rotation and coolant are turned off. Rotate the button clockwise to cancel emergency stop, but all outputs must be restarted.

Caution:

The power supply isn't always cut off upon emergency stop. Please refer to the electrical configuration description of the machine tool manufacturer for details;

Before releasing emergency stop, please eliminate the problems of the machine tool.

3.2 Hard limit over travel

The system alarms if the tool touches travel switch during operation. The axis in corresponding direction can't move, and only moves in reverse direction. Before the alarm is released, the system can't enter automatic operation normally. After investigating the alarm reason, press [

] to clear the alarm information.

3.3 Soft limit over travel

If the tool enters the restriction area regulated by the parameter (travel limit), the system alarms over travel, and the tool decelerates and stops. At this moment, you can move the tool to safe direction in manual mode, and then press [

] to release the alarm.

Caution:

During automatic operation, when the tool touches an axial travel switch, the tool decelerates and stops all axial motions, and only displays one over travel alarm.

During manual operation, when the tool touches an axial travel switch, the tool only decelerates and stops motion on current axis, and still moves along other axes.

When the tool is in safe position, press [Reset] to clear the alarm. Please refer to the manual of the machine tool for details.

Both limit alarm and soft limit alarm have a deceleration stop, and therefore the sensing range of the limit should have sufficient space, or else the limit protection will be disabled due to over travel.

Alarm processing

If alarm occurs, please refer to the alarm code to confirm the failure reason.

When alarm occurs, if the system isn't reset, the alarm will constantly prompt no matter whether the alarm still exists, so as to avoid the conditions that false alarm causes system suspended, but can't find the reason.

If the error is caused by data setting, modify the data, and then press [Reset] to clear the alarm info.

When alarm occurs, please remove the alarm reason. Please note that several alarms may occur at the same time. Please refer to the alarm info in the Diagnosis menu for details. When the alarms are eliminated, please press [Reset] to clear the alarm ring.

3.4 Self-diagnosis function

The CNC system may stop even when there is no alarm info, this may be because the system is executing certain processes. Please check with the self-diagnosis function.

The step of self-diagnosis follows:

(1) In the main menu, press [Diagnosis] to enter the diagnosis interface;

(2) Select [Input] to enter the input diagnosis interface, or select [Output] to enter the output diagnosis interface;

(3) Output diagnosis: In edit mode, press the direction keys to select the output port, and press [EOB] to switch the output level of corresponding output port;

Input diagnosis: When certain input signal is valid, the corresponding area on the screen flashes.

4. Program saving & editing

Saving the program in the memory

4.1 Keypad input (new program)

Create new program in the memory with the keypad, and the step follows:

In the main menu, press [Edit] to enter program edit interface;

Press [File] to enter file operation interface;

Select [New] to create a new file;

Enter the file name and press [EOB] to confirm and create a new program in current directory in the memory, and load into the system by default;

Select [Close] to exit [Edit] interface;

In edit mode, enter the program content;

After editing all programs, press [Reset] to save the edited programs into the system memory.

4.2 Copying processing files from USB disk

The step of copying CNC processing file to system memory through USB disk follows:

In the main menu, press [Edit] to enter program edit interface;

Press [File] to enter file operation interface;

Select USB disk and press [EOB] to enter;

Move the cursor to select a CNC file and then select [Copy];

Return to the root directory, locate the PROG directory in disk D, enter the directory, and select [Paste] to complete copying.

4.3 Reading programs into work area

4.3.1 Reading programs from controller into work area

The step of loading files from system memory into work area follows:

Press [File] to enter file operation interface;

Select desired program, which is in PROG directory in disk D by default, press [EOB] to enter subdirectory, or press [Cancel] to exit;

Move cursor to select desired program, press [EOB] to confirm and load the program.

4.3.2 Reading programs from USB disk into work area

The step of loading files from USB disk into work area follows:

Insert the USB disk;

Press [File] to enter file operation interface;

Select USB disk, move cursor to select a file in the disk, and press [EOB] to load the file.

4.4 Editing & modifying programs

The program in CNC memory can be edited with NC keypad. In the main menu, press [Edit] to enter program edit interface and edit the program in current work area (for loading program into work area, refer to section 8.2). The edit mode is similar to notepad in Windows. Move the cursor directly to locate, press keys to enter, press [EOB] to change line, and press [Delete] to delete the character where the cursor locates.

Caution

After all operations, press Reset to save the files, and the functions base on edit mode;
DK300A uses new file mapping technology, and allows loading processing files that exceed its memory. Therefore, to ensure the system efficiency, you can only search and process, but can't edit the processing files that exceed 2MB.

4.4.1 Deleting files in memory

Follow the step below to delete the programs in system memory:

Press [File] to enter file operation interface;

Follow the prompt on the screen, select the file and press [Delete] to confirm and delete the file.

Caution

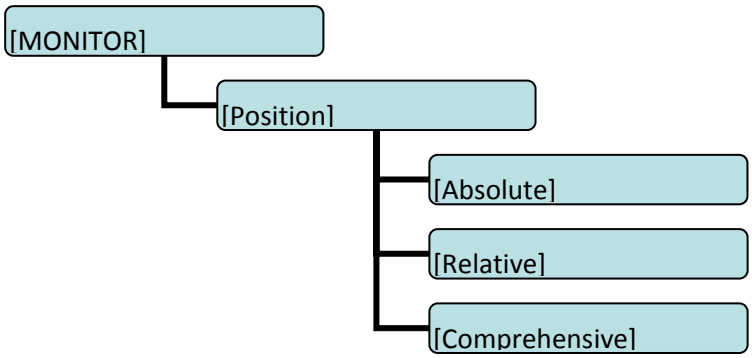
If the program has been loaded into work area, you need to restart the system to delete the program, or else the system will report error.
The programs loaded into the work area can't be deleted, or else the system will report error.

5. Main interfaces of the system

5.1 Position interface

The position interface shows current machine tool coordinates, including absolute position, relative position and comprehensive position. In the main interface, press [Monitor] to enter the position interface.

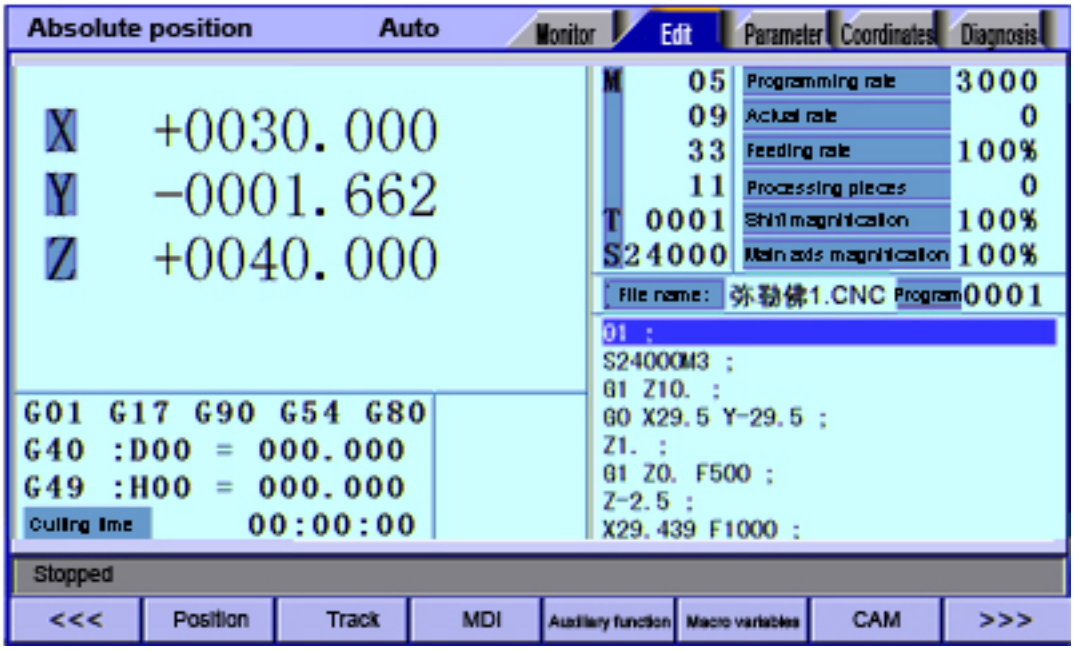
To enter position interface:



5.2 Absolute position

The position of current machine tool coordinates relative to the origin of workpiece coordinate system

The absolute position interface follows:



5.3 Relative position

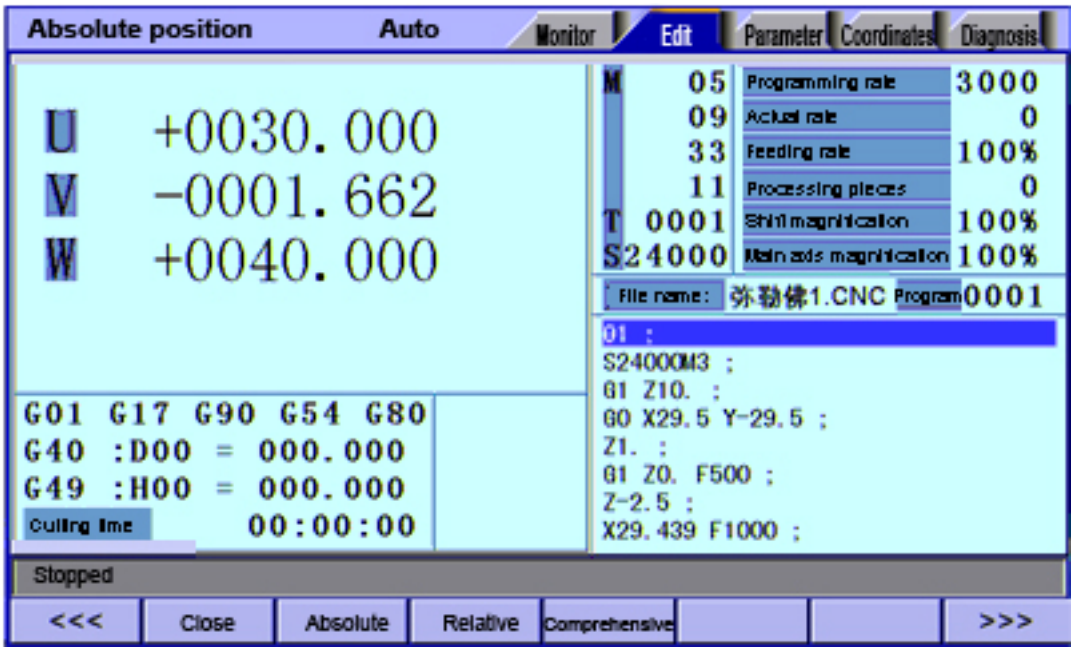
In manual mode, reset current coordinates to check the relative motion distance of any displacement, and thus it is called as relative position.

This interface is usually used for early tool setting. Considering that some operators have been used to manual calculation, this function is preserved. With the more and more powerful of automatic centered function, it is used less.

The operation follows:

- Enter [Position] interface;
- Switch to [Relative] interface;
- Then, enter manual mode;
- Press a coordinate axis No., e.g., 'X', and the X coordinate flashes;
- Press "Cancel" to reset X coordinate to 0;

The relative position interface follows:

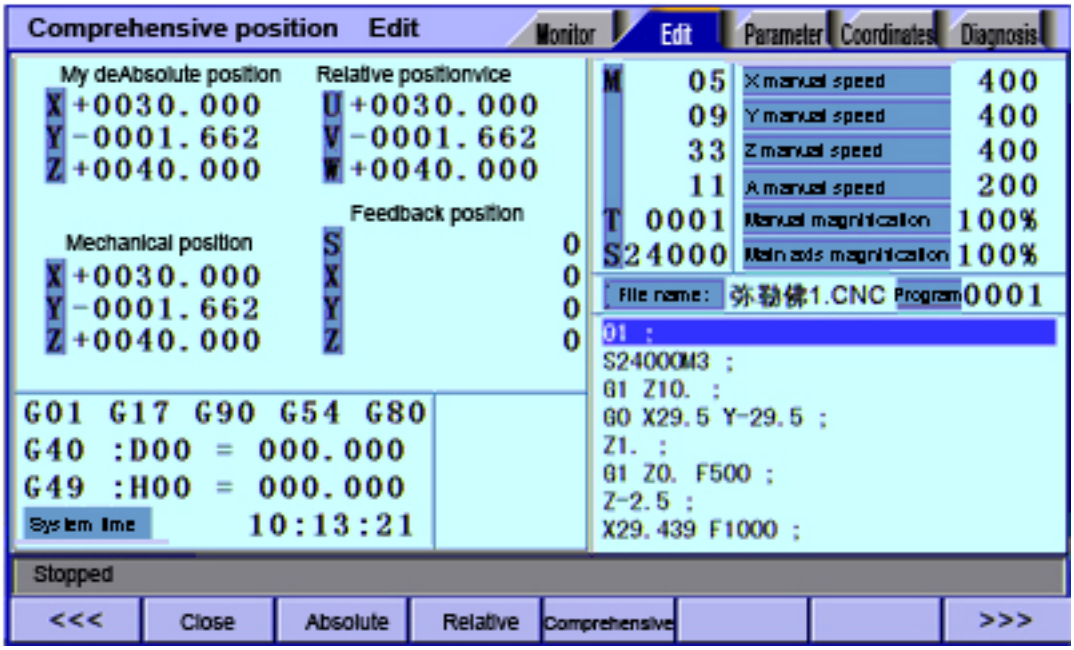


Relative Position Interface

5.4 Comprehensive coordinates

The interface displayed by absolute coordinates and machine tool coordinates.

Comprehensive position interface is shown below:

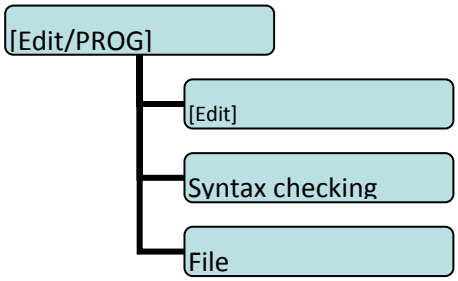


Comprehensive Position Interface

5.5 Edit interface

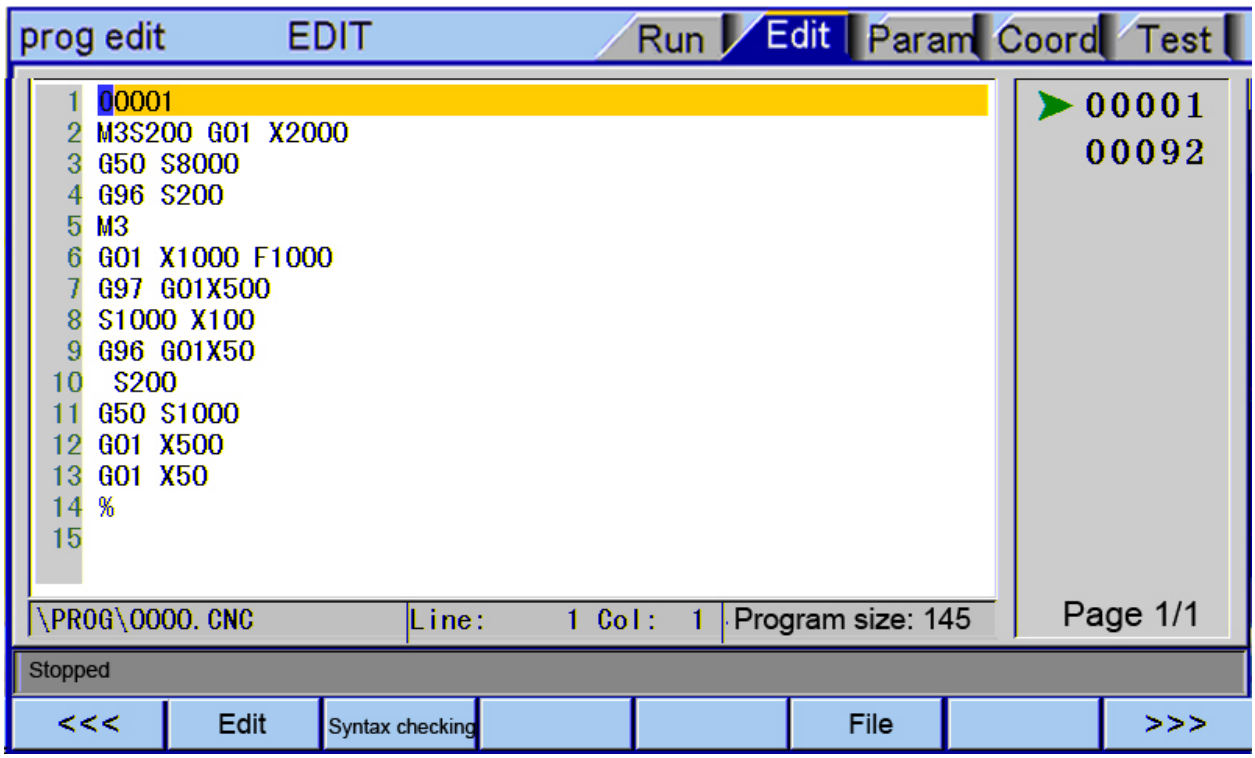
The edit interface shows the program info in current work area. In the main interface, press [编辑 PROG] to enter the program interface.

To enter program edit interface:



5.5.1 Program edit

The program edit interface shows the NC program currently processed; in edit mode, you can edit the NC program (see section 8.3 for details).

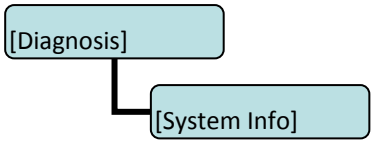


Program Edit Interface

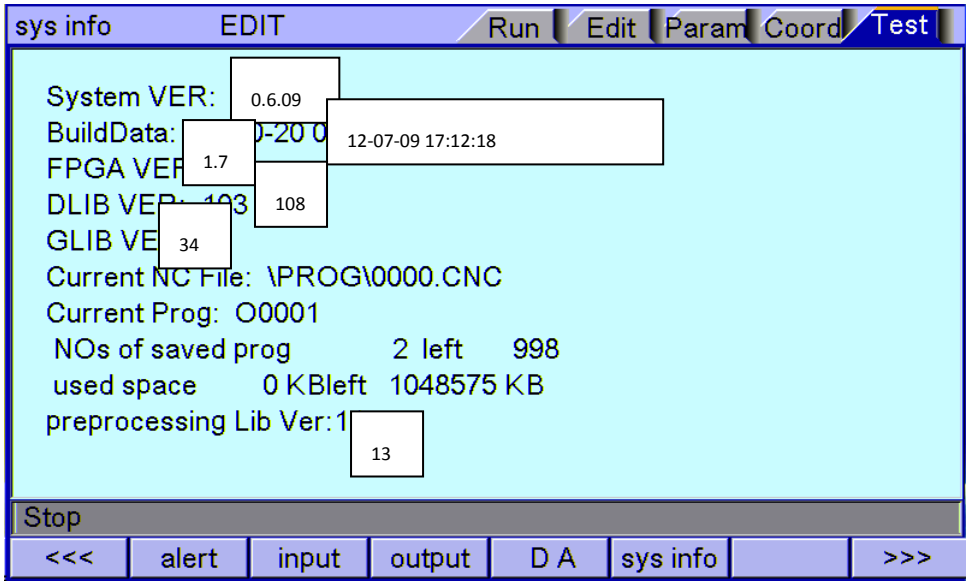
5.6 System info interface

The system info is a summary of the program blocks in current processing area, and calculates the resource usage in current work area. The upper right of the program directory interface shows the version info of current controller software. If our engineering personnel ask to confirm the software version of the controller on site, please provide this version info.

To enter system info interface:



System info interface is shown below:

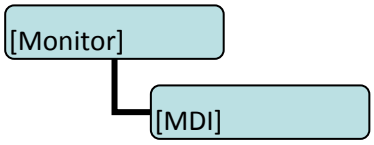


System Info Interface

5.7 MDI interface

MDI mode is mainly used for the execution of single G code in certain occasions.

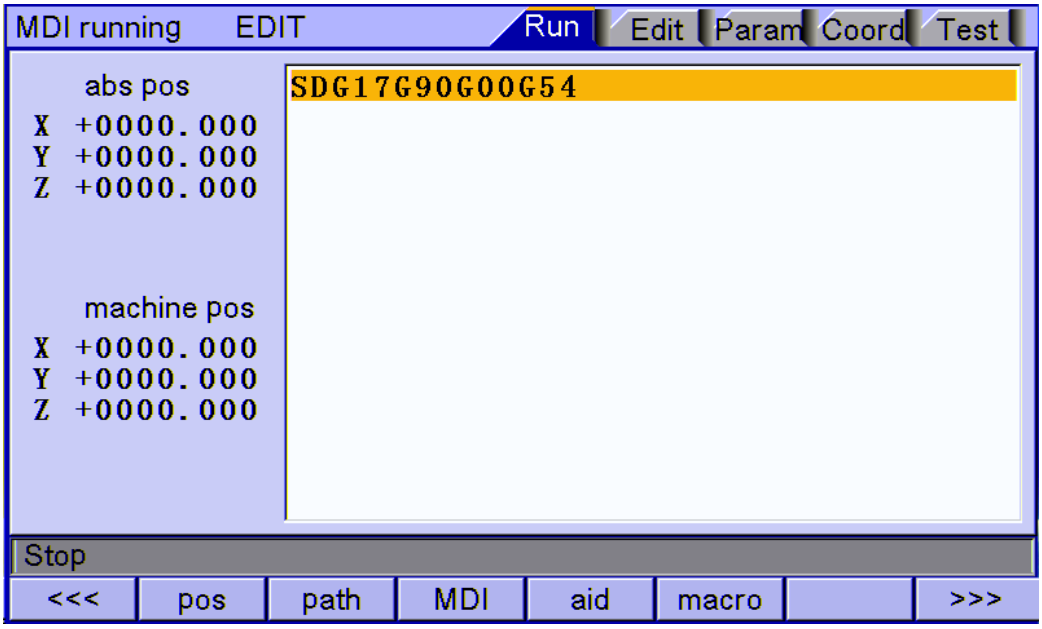
To enter MDI interface:



In MDI interface, enter complete NC code instruction in edit mode, press the [Start] key in the edit mode and confirm to execute directly.

To restore the default settings quickly, press and hold the [Reset] key for three seconds and choose to reset or not.

MDI interaction interface is shown below:

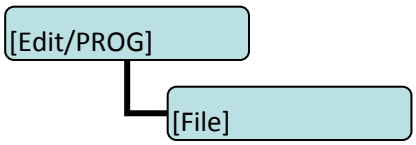


MDI Interface

5.8 File management

In the file management interface, you can manage the system files.

To enter file management interface:



File management mainly has the following functions:

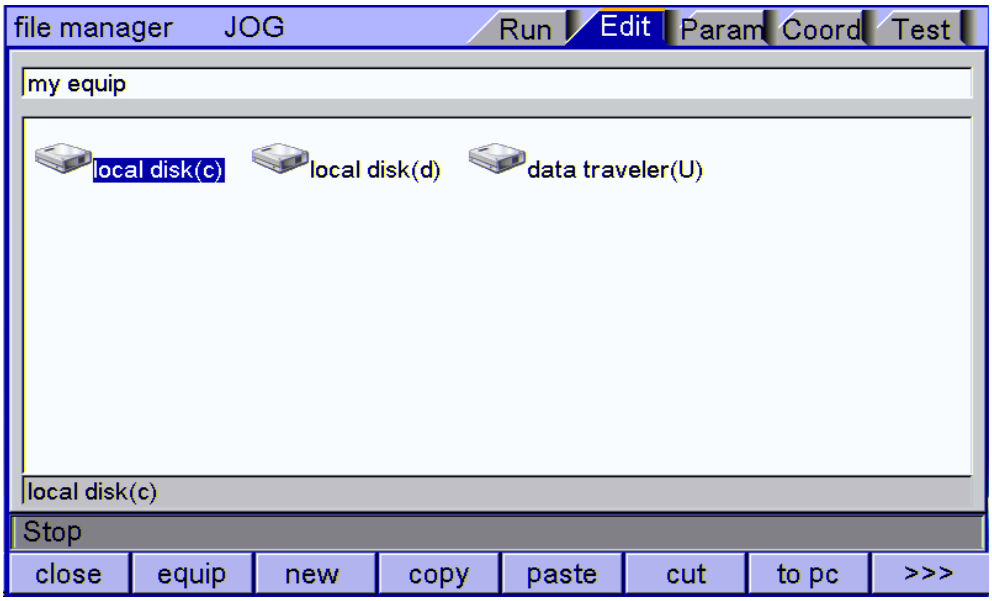
Connect the UBS disk, and copy the files between USB disk and electronic disk;

Upgrade system software: Copy the upgrade file to system memory in either method above to upgrade the software;

Restart the controller. In [File Management] interface, press the Reset key to restart the controller. This method is different from restarting due to power failure. In certain occasions, you can restart the controller quickly in this method to make certain function take effect.

Connect to PC with the USB cable, and exchange the data between USB disk and PC.

File operation interface is shown below:

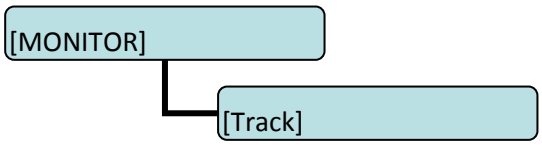


File Operation Interface

5.9 Graphic simulation

[Track] function is to simulate NC processing program.

To enter graphic simulation interface:



Enter track interface to enable real-time track display automatically. During automatic running of the system, the motion track is displayed in real-time. In standby mode, you can also press Preview to prescan the processing file.

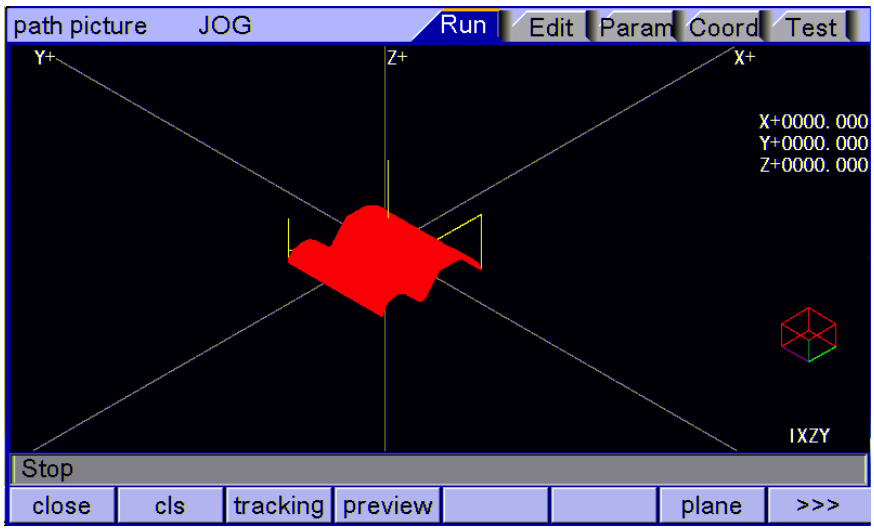
The shortcuts of adjusting position:

: Zoom in

: Zoom out


: Shift position; the shift unit is the set pixel unit

Graphic simulation interface is shown below:

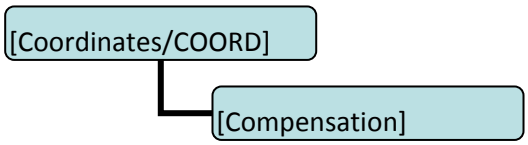


Graphic Simulation Interface

5.10 Compensation interface

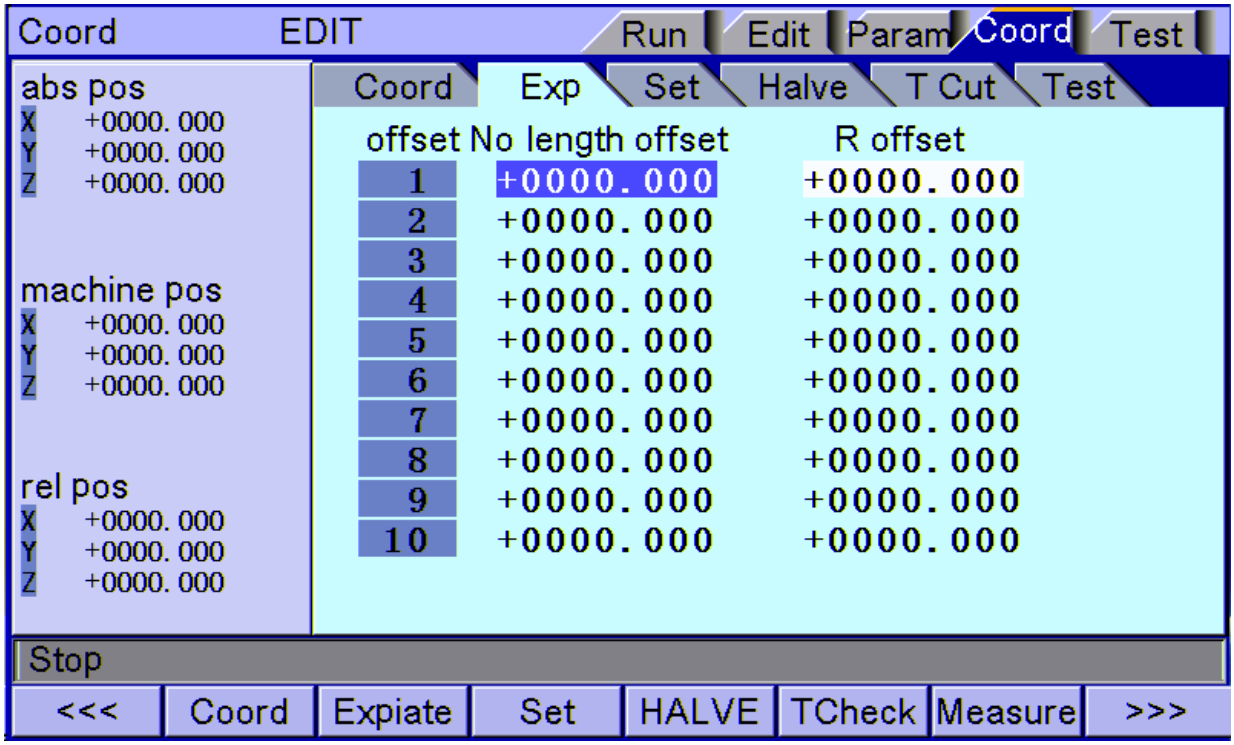
Tool compensation interface shows tool compensation info of the system, including tool length compensation, tool radius compensation and other input variables. The compensation method is described below.

To enter tool compensation interface:



M series tool compensation interface has two compensation variables, i.e. tool length compensation and tool radius compensation; corresponding to G43, G44 and G41, G42; enter compensation value to corresponding compensation number, and transfer the compensation number in NC program to realize the compensation. Tool compensation numbers have 36 variables.

Tool compensation interface is shown below:



Tool Compensation Parameter Setting Interface

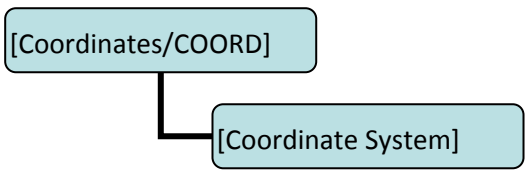
5.11 Workpiece coordinate system setting interface

The coordinates interface shows coordinate system info, including coordinate system and compensation. In the main interface, press [Coordinate] to enter coordinate system.

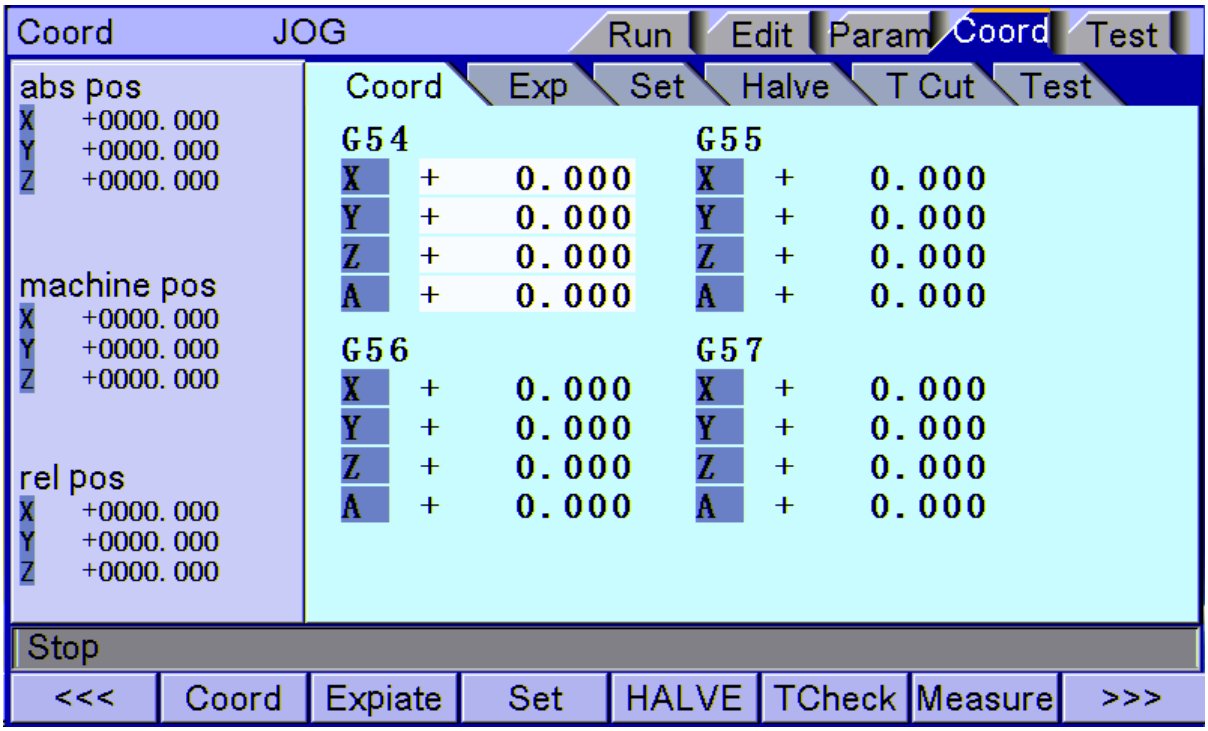
Workpiece coordinate system

Display workpiece coordinate system, i.e. the offset of workpiece home position and machine tool home position. Totally six basic workpiece coordinate systems (G54~G59) and nine extension coordinate systems (G591~G599) are available.

To enter workpiece coordinate system interface:



The workpiece coordinate system interface is shown below:



Workpiece Coordinate System Setting Interface

See 1.4.6.1~1.4.6.3 for the details of setting methods.