SAFETY NOTICE

Before using this control system, please read this manual carefully before operating. Please check whether the wiring is correct before power on!

The operation and use of the product are described in this manual as much as possible. However, due to too many possibilities involved, it is impossible to explain all the allowed and disallowed operations. Therefore, in order to ensure the normal use of the product and the safety of personnel and equipment, the operation not stated in the instruction manual shall be deemed as not allowed.

● **Working environment and protection:**

1. The working environment temperature of the control system is – 10 ℃ ˜ 50 ℃. When the temperature is exceeded, the system may work abnormally or even crash. When the temperature is too low, the LCD will display abnormally.

2. The relative humidity should be controlled at 0–85%.

3. When working in the environment of high temperature, high humidity and corrosive gas, special protective measures must be taken.

4. Prevent dust, dust, metal dust and other debris into the control system.

5. Its screen protection should be kept away from sharp objects;Prevent objects in the air from hitting the screen;When there is dust on the screen that needs to be cleaned, wipe it gently with a soft paper towel or cotton cloth.

● **System operation:**

When the system is operating, you need to press the corresponding operation button. When pressing the button, you need to press it with the belly of the index finger or middle finger. Do not press the button with your fingernail, otherwise the mask of the key will be damaged and your use will be affected.

The operator for the first time should understand the correct use method of the corresponding function before carrying out the corresponding operation. For unfamiliar functions or parameters, it is strictly forbidden to operate or change the system parameters at will.

For problems in operation, we provide telephone consultation service.

● **System maintenance:**

Operators without strict training or units or individuals not authorized by the company shall not open the control system for maintenance operation, otherwise the consequences shall be borne by themselves.

● **System warranty Description:**

Warranty period: within 24 months from the date of delivery.

Warranty scope: during the warranty period, any failure occurred under the condition of operation according to the use requirements.

During the warranty period, the fault beyond the warranty scope is charged service.

Outside the warranty period, all troubleshooting services are charged.

◆ **The following conditions are not covered by the warranty:**

1. Any human failure or accidental failure that violates the use requirements;Any human failure or accidental failure that violates the use requirements;

2. Damage caused by incorrect wiring without referring to the manual and connecting socket of hot plug system;

3. Damage caused by incorrect wiring without referring to the manual and connecting socket of hot

plug system;

4.   Damage caused by natural disasters, etc;

5.   Damage caused by unauthorized dismantling, refitting, repairing, etc.

● **Other matters:**

In case of any inconsistency and incompleteness between the manual and the system function, the system software function shall prevail.

The control system is subject to change.

Only one copy of "operation manual" is provided free of charge. If you need the latest "operation manual", you can obtain the electronic version of the manual (PDF format) for free, and inform your e-mail mailbox to send it in the form of e-mail.

The product functions described in this manual are only for this product. The actual function configuration and technical performance are determined by the design of the machine tool manufacturer. The functional configuration and technical indicators of the CNC machine tool are subject to the instructions of the machine tool manufacturer.

CATALOG

PART 1 SYSTEM OVERVIEW

1.1 System Introduction

XC609D engraving and milling CNC system is a new generation of engraving CNC system developed by our company. Support boring, drilling, milling and high-speed engraving. It adopts 32-bit high-performance microprocessor, real-time multi task control technology and hardware interpolation technology, full linkage, prospective small line segment algorithm, interpolation accuracy of 0.001mm, and maximum speed of 12m / min.

Software and hardware characteristics of XC609D CNC system:

Based on 32-bit microprocessor, full linkage, 0.001mm interpolation accuracy and maximum speed of 12m / min.

It adopts 3.5 inch color wide screen LCD with resolution of 480x320 and windows interface style. Equipped with 5 soft function keys, easy to operate and learn. Provide parameter classification, alarm log, system diagnosis and other rich display interface to facilitate debugging and maintenance.

It is compatible with FANUC system instruction.

There are 40 kinds of G commands, supporting drilling cycle.

The teaching function is supported.

The tool setting device is supported.

Full screen editing of part program, built-in 512M mass program space, can store n part programs.

With USB interface, it supports file reading and writing, data backup and direct processing (DNC) of U disk.

Input 24 (expandable to 96) points, output 24 (expandable to 96) points (customized), flexible and convenient.

The Chinese / English operation interface, complete help information, more convenient operation.

The system adopts forward-looking small line segment algorithm, and the processing is stable.

It supports fast operation of specified operation position and multi coordinate system (automatic saving after power failure).

The function of saving breakpoints is supported (press pause key to save automatically or power off to save automatically).

It supports multi-level operation authority, facilitates equipment management, and has time limited system locking function.

Support G code files of third-party software such as UG, Mastercam, PowerMILL, featurecam, ArtCAM, jdpaint, Wentai, etc.

1.2 technical specifications

basic function	
Number of control axes	3 ~ 4 axis (x, y, Z, a)
Number of linkage axes	Full linkage
Simulation spindle	1
Spindle ratio	10%~150%
Minimum instruction unit	0.001 mm
Maximum instruction value	±99999999×Minimum instruction unit
Rapid feed rate	12000 mm / min
Rapid feed rate	F0, 25%, 50%, 100%
feed rate	12000 mm / min
Rate feed rate	10%～150%

Electronic gear ratio	1～65535: 1～65535
Automatic acceleration and deceleration	yes
location	G00 (linear interpolation positioning)
interpolation	Linear (G01), arc (G02 / G03 / G12), spiral interpolation
Return to reference point	Automatic return to reference point (G28)
LCD	3.5-inch TFT LCD with resolution of 480x320
MDI software key	Five
Single step feed	x1，x10，x100
communication interface	U disk interface
External handwheel interface	yes
I / O interface	24 / 24 (expandable to 96 / 96)
Pause (SEC)	yes
Quasi stop state	yes
Accurate stop	yes
Memory trip check	yes
MDI operation	Yes, it supports multi segment operation
reset	yes
Selective stop switch	yes
Single section operation	yes
Program protection switch	yes
Self diagnosis function	yes
Emergency stop	yes
Power Supply	Single phase AC220 V + 10% - 15%, 50 Hz ± 1 Hz.DC24V/1A
Coordinate system	Machine coordinate system (g53), workpiece coordinate system (G92, g54 ~ G59), local coordinate system (G52), coordinate system plane designation
Automatic coordinate system setting	yes
Decimal point input	yes
Auxiliary function	
Auxiliary function	M2 digit, M code user-defined, manual / MDI / automatic control spindle forward, reverse, stop;Control the start and stop of coolant.
Spindle function	
Spindle function	Analog spindle (0-10V)
Edit operation	
Editing function	Parameters, diagnosis bit input, program editing, MDI multi program segment execution
storage capacity	512M
Number of stored programs	N
Display of program name	Chinese, English, numbers, combinations
Program line lookup	yes
Optional program skip	yes

Program switch	yes
display	
display	Chinese, English
Display of processing time and number of parts	yes
Spindle speed, M / s command	yes

Application fields:

Used for processing molds: embossing board, sole mold, button mold, zipper mold, pattern and character impression mold and hot gold mold, instrument mold, glass mold, etc.

Used in advertising industry: company board, sign, building model, badge, badge, signboard, display board, logo, door plate, indicator board, craft decoration, furniture decoration, etc.

For portrait, landscape, calligraphy, seals and other arts plane carving, Yin, Yang outline, relief, three-dimensional carving, etc.

It is used for various small and medium-sized automation equipment and instruments.

Used for welding manipulator, coordinate feeding manipulator, spraying manipulator, etc.

wait

PART 2 OPERATION INSTRUCTIONS

Chapter I description of operation authority

1.1 permission level

XC609D uses a classified and hierarchical authority structure, and various permissions are for different user groups. Among them, class B is the permission of the machine tool manufacturer, while the permission categories used by machine tool users are class C and class F. their permissions are described as follows. As shown in the table below:

Authority classification	User range
Class C	Machine tool operator
Class F	Unskilled workers with limited operation

The authorities at all levels are as follows:

Permission level	Permission description	Operation password
C operation level	Programmable and optional program processing; It can edit cutter compensation; Parameters can be modified;	The password can be changed
F restricted class	All operations to modify data and configuration are invalid	There is no operation password

Note: for the modification method of permission level and permission password, please refer to "permission settings"

1.2 operation authority

The operation instructions subject to permission restrictions are shown in the table below. Those not listed in the table are deemed to have no permission restrictions or functions are temporarily closed to users.

Page operation	Required permissions	Other conditions

Data setting and backup	Parameter modification	Permission above C	Edit mode, stop state, parameter switch on
	Parameter backup and recovery	Permission above C	Edit mode, stop state, parameter switch on
	Parameter backup and recovery	Permission above C	Edit mode, stop state, parameter switch on
Part program	open	Grade F	Edit mode or automatic mode, stop state
	New, edit, delete	Grade C	Edit mode, stop state, program switch on
	Program import	C	Edit mode, stop state, program switch on
	Program export	C	Edit mode, stop state, program switch on
System settings	System locking time setting and system locking function opening	Grade B	
	Import boot screen	Grade B	

Note: the default authority, program switch and parameter switch can be set in the comprehensive parameters.

Chapter 2 interface display and setting
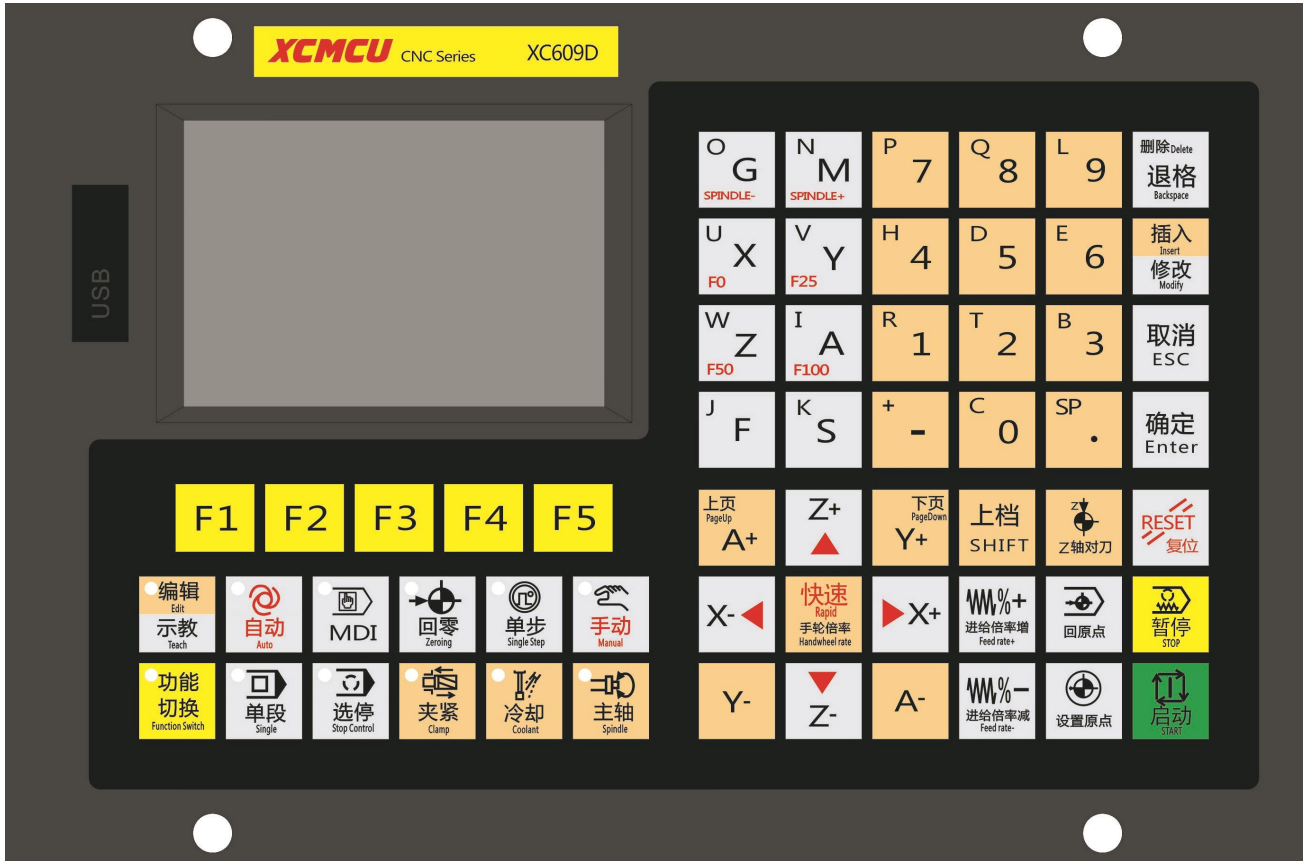
2.1 panel
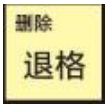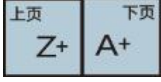
2.1.1 description

The upper part is the character number editing key.

The lower part is the editing and axis selection key, which is invalid in the mode of editing, inputting and hand-held handwheel. In zero return, single handwheel, manual mode, the edit key is invalid.

2.1.2 character number editing key

Character number editing includes all numbers and letters, as well as [reset], [backspace], [delete], [Cancel], [Enter], [upload], [insert], [modify], as well as cursor keys and page turning keys.

Key Icon	Key name	Functional use
RESET 复位	Reset key	CNC reset, program end processing, release alarm, terminate input and output.
删除 退格	Backspace / delete key	Delete the character after the cursor position, delete key to delete the character in front of the cursor. (backspace and delete share a common entity key, which can be selected with the up file key)
插入 修改	Insert / modify key	Modify edit mode, switch between insert and modify mode, or modify data by parameter
取消 ESC	escape key	Cancel the input and close the pop-up dialog box to return to the program content screen.
确定 Enter	enter key	Confirm the input and close the pop-up dialog box to return to the program content screen. At the same time, the specific program section terminator function, as well as line feed function.
上档 SHIFT	shift	Select or cancel the upshift function.
Z 轴对刀	Z axis tool setting key	Automatic tool setting of Z axis with tool setter in manual mode
回原点	Go back to the origin	In manual mode, each axis returns to absolute coordinate zero and Z axis returns to safe height. Strategy: if z-axis is higher than safe altitude, xya will return to absolute zero first, and z-axis will return to safe altitude; If the z-axis is lower than the safe height, z-axis returns to safe height first, and xya returns to absolute zero.
设置原点	Set Origin	In manual mode, the workpiece origin can be set, axis selection and all selection.
上页 下页 Z+ A+	Page up and page down	Edit or input mode. Scroll up or down 10 lines at a time. (shared with axis function key, page turning in editing and entry mode, and axis selection key in other modes)
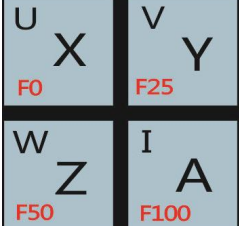
Y+ ▼ ▲ Y- ▼ ▶X+ Y-	Up, down, left and right Cursor keys	Move the cursor up, down, left and right.(shared with axis function key, page turning in editing and entry mode, and axis selection key in other modes)
(keypad)	Character key	Character input.Double function key, you need to press the up shift key to input the second function character. (the number key can be defined as the port direct control key, which works in manual mode)
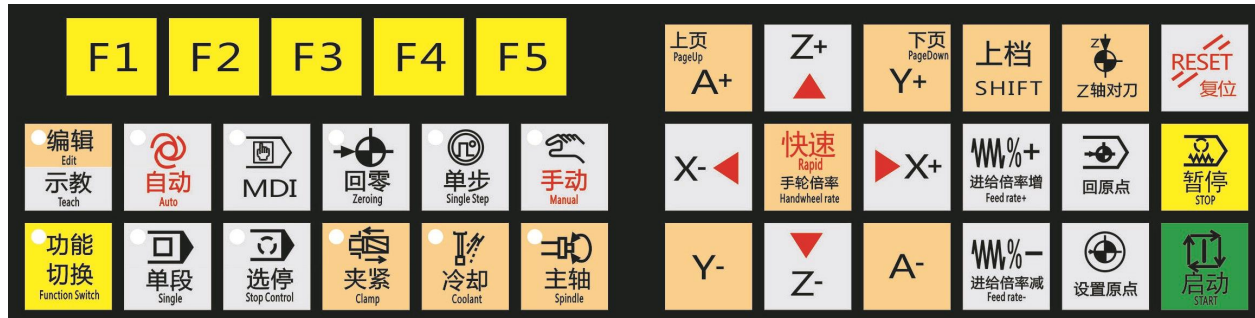
2.1.3 function operation key of machine tool

Used to select various display screens.

Key Icon	Key name	Functional use
编辑 示教	Edit key	Enter editing mode.Press again and the teaching function is switched, and the indicator light flashes as the teaching function
自动	Auto Key	Enter the automatic operation mode.
MDI	Input key	Enter the operation mode of input (MDI).
回零	Zero return key	Enter the return to reference point (machine zero) mode.
单步	Single step key or Handwheel mode key	Enter single step mode or handwheel operation mode.
手动	Manual key	Enter manual operation mode.
功能 切换	Function switching	Switch page, reverse switch page when upshift
单段	Single segment bond	When the program is running automatically, it can switch between single section operation and continuous operation, and the indicator light will be on when the single section

		operation is effective.
选停	Select stop key	In the automatic and input mode, when the stop is effective, the indicator light will be on and M01 stop will be executed.
夹紧	Clamping key	Loosening and clamping fixture
冷却	Cooling key	Turn on and off the coolant
主轴	Spindle key	Manual, handwheel, return to zero effective, switch between spindle forward transmission and stop
进给倍率增 进给倍率减	Feed rate key	Manual rate or feed rate key. Press the key once to increase or decrease the feed rate by 10%. 10%--150%。
U X F0 / V Y F25 / W Z F50 / I A F100	Fast rate selection	Fast rate f0-100%.
O G SPINDLE- / N M SPINDLE+	Spindle ratio	Spindle ratio 10% ~ 150%
快速 手轮倍率	Quick start key Handwheel ratio switching	Manual speed, speed switch. In single handwheel and single step mode, it is used for x1, X10, X100 magnification switching
启动	Start key	Cycle start key. Automatic or MDI program running start.
暂停	Pause key	Feed hold key. Automatic or MDI program running is suspended.
X- ◄ ► X+	X-axis movement or X-axis handwheel	In the zero return, manual and single step mode, the x-axis moves in the positive or negative direction. In the single handwheel mode, the x-axis handwheel axis is selected.

	axis selection key	
	Y-axis movement or y-axis handwheel axis selection key	In the zero return, manual and single step mode, Y-axis moves in positive or negative direction. In single handwheel mode, Y-axis handwheel axis is selected.
	Z-axis movement or z-axis handwheel axis selection key	In the zero return, manual and single step mode, the Z axis moves in the positive or negative direction. In single handwheel mode, the handwheel axis of Z axis is selected.
	A-axis movement or a-axis handwheel axis selection key	In the zero return, manual and single step mode, axis a moves in positive or negative direction. In single handwheel mode, select the handwheel shaft of axis A.

2.2 page display

2.2.1 page layout

| EDIT | STOP | USB | 100% F100 | POS | PRG | PARA | IFO | CHEK |

	Absolute	Machine	F:200.000	100%
X	0.000	0.000	S1:500	100%
Y	0.000	0.000	609MT.NC	L:0
Z	0.000	0.000	M81 Y1	
A	0.000	0.000	M71X1 M70X2 M70X3	
TIME:	0:00:00		M70X4	
CONT:0			M70X5	
G00G17G49G54G80G90G98			M70X6 M70X7	
M Speed:200			M70X8	
			M70X9	
Brief		MDI CLR	G54-G59	DNC CANCEL

project	explain
(1) U disk status and up shift key	U disk status, u appears, indicating that the U disk has been inserted, and the shift up button is on
(2) Magnification axis	Display magnification, display axis selection and magnification in

selection display	handwheel mode
(3) How it works	Editor: program editor Teaching: edit program with teaching function Automatic: automatic operation (program running) Input: manual data input, MDI operation Return to zero: return to the reference point manually Single step: manual single step feeding. If the handwheel is enabled, it is handwheel operation Manual: manual continuous feed
(4) Operation status	In the automatic / input mode, the current program instruction execution status is displayed; Single step / step display mode. Alarm status, (flashing display) prompt status.
(5) Page name	The currently selected master page label is displayed
(6) Soft function key	The operation menu of the currently displayed page or pop-up window

2.2.2 page display content

The system is divided into five display pages, which are [position], [program], [parameter], [information] and [diagnosis], which can be switched by pressing the [function switch] key. If the up shift key is enabled, it will switch in the reverse direction.

The display contents and related operations of each page are as follows:

page name	Screen display content	Related contents and operation
position	The position of the tool in each coordinate system Current set spindle speed and magnification, and actual speed ● current set feed / fast speed and rate, and actual speed ● modal value of the current system ● processing time and parts counting Program information during automatic operation	MDI program editing
program	● CNC machining program currently open Program directory	Process editing Copy and delete machining program files in program directory (including local and USB flash disk) Input / output of processing program files between different memories
parameter	Comprehensive parameters ● IO parameters ● shaft parameters	Parameter setting ● parameter setting of each axis
information	● CNC alarm in progress System information ● breakpoint information Import boot screen	Check and clear the alarm ● authority setting System lock setting Parameter switch and program switch
diagnosis	Input and output diagnosis information	Operation according to serial number

3.3 screen position

2.3.1 picture composition

The position main interface displays coordinates, processing time, number of pieces processed, manual speed, spindle speed, and operation information of each program.

S1: spindle setting speed

F: Machining feed rate

2.3.2 coordinate system setting

Position main interface, press [G54-G59] to enter this page.As shown in the figure:

This page sets G54-G59 working coordinate system.

2.3.3 setting coordinates

Press [set coordinates] to enter this page.As shown in the figure:

```
EDIT  STOP  USB              100%       POS  PRG  PARA  IFO  CHEK
                             F100
          Absolute    Machine
      X      0.000       0.000
      Y      0.000       0.000
      Z      0.000       0.000
      A      0.000       0.000


                  G54
                   X      0.000
                   Y      0.000
                   Z      0.000
                   A      0.000



       <<       Input     +Input     Measure     AXIS RD
```

Absolute input: input the offset of coordinate system in mechanical coordinates.

Incremental input: incremental input offset of coordinate system in mechanical coordinates.

Measurement input: set the current absolute coordinates directly.

Coordinate input: read the current axis machine tool coordinate, convenient to set workpiece coordinate 0 point.

2.3.4 line segmentation

```
EDIT  STOP  USB              100%       POS  PRG  PARA  IFO  CHEK
                             F100
          Absolute    Machine
      X      0.000       0.000
      Y      0.000       0.000
      Z      0.000       0.000
      A      0.000       0.000


          G54
          X      0.000   P1X:
          Y      0.000   P1Y:
          Z      0.000
          A      0.000   P2X:
                         P2Y:

       <<       X P1       X P2       Y P1       Y P2
```

2.3.5 center coordinates

Press [center coordinates] to enter this page.For centering the circle.As shown in the figure:

```
EDIT  STOP  USB              100%       POS  PRG  PARA  IFO  CHEK
                             F100
          Absolute    Machine
      X      0.000       0.000
      Y      0.000       0.000
      Z      0.000       0.000
      A      0.000       0.000


          G54
          X      0.000   P1X:
          Y      0.000   P1Y:
          Z      0.000   P2X:
          A      0.000   P2Y:
                         P3X:
                         P3Y:

       <<      Input P1   Input P2   Input P3
```

This page calculates the center of the circle according to the three non collinear points in the plane and sets it as the offset of the workpiece coordinate system.

2.3.6 brief display

Press [brief display] to enter this page.As shown in the figure:

Brief display: display absolute coordinates in large font, main program feed speed, spindle 1 speed, manual speed.

2.4 program screen

The program screen includes three sub screens: program content, local directory, and U disk directory.You can switch to these sub screens through the soft keys [local directory] and [U disk directory].

2.4.1 program content screen

This interface displays the current part program. Move the cursor up and down to preview the program content line by line, and press [pageup] and [pagedown] to preview the program content in ten lines.

2.4.2 local directory screen

The local catalog lists all part programs stored in CNC and shows the status of storage space.

Note oO0O0NC is a space occupying file. This file cannot be edited or deleted.

2.4.3 USB flash disk directory screen

Press [USB DNC] to directly run the program in USB flash disk.

Press [program import] to import the selected file into the local directory.

2.5 parameter screen

The parameter screen includes comprehensive parameters, input port configuration, output port configuration, direct control function configuration and axis parameters. You can switch to these sub screens through the soft key.

2.5.1 comprehensive parameter screen

Comprehensive parameters are used to set system related parameters. And with notes, it can be easily and quickly modified.

| EDIT STOP USB | 100% F100 | POS | PRG | PARA | IFO | CHEK |

```
P0001 Buzzer  0:Disable 1:Enable
1              0~1
P0002 Language 0:中文 1:ENGLISH(需要重启)
1              0~1
P0003 Counter save 0:Disable 1:Enable
1              0~1
P0004 Counter Mode 0:Auto 1:Instruction
0              0~1
P0005 Timer  Mode  0:Accumulation 1:Single
0              0~1
P0006 Reset OUT Pin  0:Disable 1:Enable
1              0~1
```

| GE PARA | Pin IN | Pin OUT | P2P CTRL | >> |

Note: the position units in the parameters are all 0.001, for example, G73 tool withdrawal 1000 represents 1mm.

In the editing mode, use the [modify] key to set.

2.5.2 input port parameter screen

Input port parameters are used to set input port function parameters. And with notes, it can be easily and quickly modified.

| EDIT STOP USB | 100% F100 | POS | PRG | PARA | IFO | CHEK |

```
P_IN 1     General              NOPEN
P_IN 2     General              NOPEN
P_IN 3     General              NOPEN
P_IN 4     General              NOPEN
P_IN 5     General              NOPEN
P_IN 6     General              NOPEN
P_IN 7     General              NOPEN
P_IN 8     General              NOPEN
P_IN 9     General              NOPEN
P_IN10     General              NOPEN
P_IN11     General              NOPEN
P_IN12     General              NOPEN
```

| GE PARA | Pin IN | Pin OUT | P2P CTRL | >> |

1 ~ 24 input ports can be configured as buttons, limit, alarm, handwheel axis selection and other functions, 25 ~ 48 are expansion input ports, the speed of extended input ports is slow, please do not configure limit, alarm and other functions that need to be detected in time. In the editing mode, use the [modify] key to set.

The input port can also be configured with the [modify] key in the diagnosis function, and the test port is more intuitive after the line is connected.

2.5.3 output parameter screen

The output parameter is used to set the function parameters of the output port. And with notes, it can be easily and quickly modified.

EDIT	STOP	USB	100% F100		POS	PRG	PARA	IFO	CHEK

P_OUT01	General
P_OUT02	General
P_OUT03	General
P_OUT04	General
P_OUT05	General
P_OUT06	General
P_OUT07	General
P_OUT08	General
P_OUT09	General
P_OUT10	General
P_OUT11	General
P_OUT12	General

| GE PARA | Pin IN | Pin OUT | P2P CTRL | >> |

The output port can be configured as spindle cooling, clamping, spindle forward rotation and reverse rotation signal, state tricolor lamp, which is convenient for programming.

In the editing mode, use the [modify] key to set.

The output port can also be configured with the [modify] key in the diagnosis function, and the test port is more intuitive after the line is connected.

2.5.4 direct control parameter screen

Direct control configuration can be configured to directly control the output port in manual mode.

EDIT	STOP	USB	100% F100		POS	PRG	PARA	IFO	CHEK

P_OUT01	NO CONTROL	Self-locking
P_OUT02	NO CONTROL	Self-locking
P_OUT03	NO CONTROL	Self-locking
P_OUT04	NO CONTROL	Self-locking
P_OUT05	NO CONTROL	Self-locking
P_OUT06	NO CONTROL	Self-locking
P_OUT07	NO CONTROL	Self-locking
P_OUT08	NO CONTROL	Self-locking
P_OUT09	NO CONTROL	Self-locking
P_OUT10	NO CONTROL	Self-locking
P_OUT11	NO CONTROL	Self-locking
P_OUT12	NO CONTROL	Self-locking

| GE PARA | Pin IN | Pin OUT | P2P CTRL | >> |

The trigger source can be a key or an input port. The trigger mode can be self-locking and inching.

In the editing mode, use the [modify] key to set.

2.5.5 axis screen

Axis parameters configure the electronic gear of each shaft, the speed of each shaft and other information.

EDIT	STOP	USB	100% F100		POS	PRG	PARA	IFO	CHEK

P0185 X axis motor pulses per rotation circle	
1	1~999999999
P0186 X axis length per rotation circle(0.001)	
1	1~999999999
P0187 X axis rapid(G0) speed	
8000	1~999999
P0188 X axis Feedrate max	
8000	1~999999
P0189 X axis JOG fast speed	
8000	1~999999
P0190 X axis fast speed of return Reference Point	
1000	1~999999

| << | X PARA | Y PARA | Z PARA | >> |

If there is a gear ratio per motor, the number of gears per gear is multiplied.

Denominator, the distance of each turn. If it is a screw, it is the screw pitch of the screw, in micrometer (0.001 mm).

Example: stepper motor subdivision 16, reducer 5:1, screw pitch 5mm

Molecular = 200 * 16 * 5 = 16000

Denominator = 5000

The numerator and denominator can be reduced or not.

If the motor has no speed limit, the maximum shaft speed can be obtained, and the maximum speed of the controller is 200kHz.

Maximum speed = 200000 / 16000 * 5 mm / S = 62.5mm/s = 62.5 * 60MM / min = 3750mm / min

Then the shaft speed parameters can not be greater than 3750In fact, the motor speed exceeds the maximum effective speed of the stepper motor, which should be subject to the maximum effective speed of the stepper motor.Assuming that the maximum effective speed of stepper motor is 800 rpm, the maximum speed is 800 / 5 * 5 = 800 mm / min

2.6 information screen

The parameter screen includes three sub screens: alarm information, system information and breakpoint information.[information] and [information] can be switched to the sub-system through [information] and [information].

2.6.1 alarm information screen

The alarm information screen displays the list of current alarms, and each alarm message contains the alarm number and alarm content.The alarm content is a brief description of the alarm and its release method.

There are two types of alarm: when CNC alarm occurs, program operation is stopped;Different alarm clearing methods are different. Some alarms are cleared by pressing [reset];Some alarms need to be restarted to clear them.Please refer to the instruction document of alarm information for details.

Boot screen import: B-level permission is required to make a true color bmp format screen with size less than or equal to 480 * 320, and save it as startBMP, store in the root directory of U disk, insert the controller, press [import boot screen], wait for the prompt to import successfully, if the prompt fails, please confirm whether the name of boot screen is correct.

[breakpoint information] you can view the number of lines last run and the basis for continuing to execute the program in case of unexpected power failure.

2.6.2 system information screen

Press the switch status and switch.Set the permissions and set the corresponding permissions. There is no password required for level F, and 888888 is the default password for Level C and level B. Password is required from low to high level, but no password is required from high level to high level.

　　If level c password is forgotten, you can go to level B permission first and then lower to level C, and then modify level c password.

Permissions from low to high are f level, C level and B level

2.6.3 time limit and password modification screen

Set the total power on running time of the system according to the trial time, and set 0 to cancel the limit.The contact number is used to contact the seller after the time has expired.

The above level B authority is required. After setting the trial time, you must change the password of level B permission, and remember that the factory can not unlock the level B password.

Password modification is used to modify the authority password of this level. To modify the password, you need to input the same password twice. The password can be composed of numbers and letters, with a maximum of 11 digits.

2.7 Diagnosis Screen

　　The diagnosis screen includes two sub screens: input diagnosis and output diagnosis.You can switch to these sub screens by soft keys [input diagnosis] and [output diagnosis].

2.7.1 Input Diagnosis Screen

As shown in the figure below, when the external input signal is valid, the circle of corresponding input point (in01-in96) will be filled and displayed, and the port function name is below the port number.

In editing mode, press modify to set the function of input port.

2.7.2 Output Diagnosis Screen

[output switch] and [all off], the corresponding output points (y01-y96) can be tested. When opened, the circle will be filled with the display, and the load of the corresponding port will be turned on. The port function name is below. In editing mode, press modify to set the function of output port.

Chapter 3 Manual Operation

Manual operation refers to the manual operation under each manual mode, and the manual mode includes: return to zero mode, manual mode, hand wheel mode and single step mode.

3.1 Return To Mechanical Zero Operation

Manual mechanical return to zero points in 6 cases.Reference axis parameter setting for specific setting.

Operation steps

1. Press [return to zero] on the system panel, and it is in the return to zero operation mode, and the indicator light on the key is on.

2. Press the manual axial movement switch ([x +], [y +],.) on the system panel. When returning to the reference point, the return to zero indicator will flash.

3 return to zero at the same time, up to 3 axes.

4 return to zero mode, 0: zero switch + Z pulse

1: Zero switch

2: Limit switch

3: Limit switch + Z pulse

4: Z pulse

5: No return to zero

Related treatment

After returning to mechanical zero, the parameters can be set: after returning to mechanical zero, the machine tool coordinates of each axis are set to 0. The offset of each axis after returning to mechanical zero.Set in axis parameters.

If it is higher than 1 mm, it is better to use the limit switch to return to zero.

3.2 Tool Setting (Set Workpiece Origin)

Operation steps

1 press the manual axial movement switch ([x +], [y +],.) on the system panel in manual or single step state to move to the required workpiece surface.

2 press [set origin] to pop up the menu, select the required axis or all axes, and then press [OK] to clear the corresponding coordinates or all axes.

3.3 Tool Setting Device Z-Axis Tool Setting

Long time use will make the tool wear, or change the position relationship between the tool tip and the workpiece origin after tool change, which directly affects the machining quality.In this case, the tool setting instrument is needed to determine the new tool tip offset.

Operation steps

When the thickness of the tool setting device is not known, set the thickness of the tool setting block to 0 in the parameter.

2 move the tool tip to the top of the tool setter, press [manual] on the system panel, and then press [z-axis tool setting] to start z-axis tool setting.

3 and then move to the workpiece origin. If the z-axis coordinate is displayed as - 5, set the tool setting block thickness to 5000 in the parameter.

4 at this time, you can press [set origin] on the system panel, select the desired axis or all axes in the pop-up menu, and then press [OK] to reset the coordinates of corresponding axes or all axes.Finish the tool setting of workpiece origin.

5 select the program to process.

Press [6] to reset the tool tip on the tool setting panel, if it is necessary to press the tool setting tool to the top of the tool setting tool, if it is necessary to press the tool setting tool to replace the tool.

7 in the [information] screen, check the line number of the breakpoint, and then in the [program] screen, if hundreds of lines (as the case may be) are selected before the program breakpoint, manually open the spindle, and then press [auto] and [start] to continue to run the processing program.

If the tool is changed again after 8, repeat steps 6 and 7.

3.4 Manual Feed

In manual mode, press the manual axial movement switch on the panel of the machine tool to make the tool move continuously along the selected axis and direction.

Operation steps

1. Press the mode key [manual] to select the manual operation mode, and the indicator on the key is on.

2. Select the movement key of the axis to be moved, press and hold it, so that the machine tool moves along the selected axis direction.

3. Release the key of axis movement, and the machine tool will decelerate and stop immediately.

Relevant explanation

Manual feed rate

By default, the manual low speed is based on comprehensive parameters and shared by all axes.

Manual rapid feed

In manual mode, pressing 快速 key can control the manual movement to manual rapid feed. 快速 is a key with self-locking. When pressed several times, it will switch in the switch state. When the indicator light on the key is on, it means that the manual fast switch is on; when the indicator light on the key is off, it means that the manual fast switch is closed.
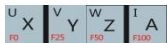
1. Set the fast rate to the lowest level F0.

2. Set the fast rate to 25%.

3. Set the fast rate to 50%.

4. Set the fast rate to 100%.

3.5 Single Step Feed

In the single step feed mode, select the movement key of the axis to be moved. Each press can make the selected axis move step by step in the selected direction. The minimum unit of movement is the minimum programming unit of the system. The input multiple of each step can be 10 times, 100 times and 1000 times, corresponding to x1, X10 and X100.

Operation steps

1 press the mode key [single step], when the parameter p0026 = 1, the system enters the single step feeding mode, and the indicator light on the key is on.

2 press XTI23523 to switch between x1, X10 and X100.

3. Press the axis movement key to select the axis to be moved and the moving direction. Each time you press a motion key, the corresponding axis will move one step in the specified direction, and the moving rate is the same as that of manual feed rate.

3.6 Hand Wheel Feed

In the handwheel mode, the micro feed of the machine tool can be realized by rotating the manual pulse generator on the operation panel or external. The user can select the axis to be moved through the axis selection key.

The minimum unit of the movement of each scale of the manual pulse generator is the minimum programming unit, and the optional magnification is 1 times, 10 times and 100 times.

Operation steps

1 press the mode key [single step], when the parameter p0026 = 0, the system will enter the handwheel feed mode, and the indicator light on the key will be on.

2. Press the rate switch key XTI23785 by single handwheel to switch between x1, X10 and X100 to select the corresponding movement of each scale of the hand remote pulse generator. In handwheel mode, it means that the movement corresponding to each scale is the minimum programming unit×1、×10、×100。

3 press the handwheel axis selection key to select the axis to be moved.

4 turn the hand pulse generator, clockwise rotation of the selected axis positive motion, counterclockwise rotation of the selected axis negative motion.

3.7 manual auxiliary operation

3.7.1 Manual Coolant Switch

In manual / handwheel / single step / return to zero mode, press the [cooling] key to change the state of the coolant switch, that is, when the coolant is output, press this key to turn off the output; when the coolant is not output, press this key to turn on the output. The [cooling] key is a button with self-locking, which will switch from "on → off → on" when pressed several times.

3.7.2 Manual Clamping Switch

In the manual / handwheel / single step / zero return mode, press the [clamp release] key to change the clamping switch state, that is, when clamping output, press this key to turn off the output; when the clamping output is not, press this key to open the output.The [clamp release] button is a button with self-locking. When pressed repeatedly, it will switch between "clamp → loosen → clamp".

3.7.3 Manual Spindle Control

In manual / handwheel / single step / zero return mode, press the [spindle] key, if the spindle runs, it will stop;If the spindle stops, it will forward.

3.8 Return To Origin (Return To Workpiece Origin)

Operation steps

1. In manual mode, press [return to home] on the system panel.
2. The tool tip moves to the workpiece origin, which is also the program origin.

Chapter 4 Automatic Operation

Machine tool running under program control is called automatic operation. Automatic operation is divided into several types, including program running and MDI running.

4.1 Program Operation

In automatic mode, running a program stored in memory in advance is called program running.

Operation steps

　1 Store the program in memory (it can be edited directly in the system or exported from U) or directly to usbdnc.

　2 press the [auto] key to enter the automatic mode.

　3 Press [Function Switch] to enter the program screen, then press the [Local Directory] soft key, and then press [↑], [↓] to retrieve the program to be executed.

　4 Press the soft key 【« 】 again to return to the program interface.

　5 【Start】key to start the automatic running program. Relevant explanation

Related explanation

　Automatic operation stop method

　(1) the programmer enters the stop command at the place to stop in advance, including M00 (program pause), M01 (optional stop), M02 / M30 (end of program).

　M00 (program pause)

　　① After M00, all the program segments containing mode information are automatically saved. Press [start] to start automatic operation again. If the motor is moved manually, the non-z will return to the position when it is suspended again, and then the z-axis will continue to execute when the

motor is suspended. Therefore, it is better to manually return to the position above the original position before resuming the operation.

M01 (optional stop)

② When the program select stop switch is on, the function of M01 is equivalent to M00;When the program select stop switch is off, M01 has no effect.

M02 (end of program)

③ After the program segment containing M02 is executed, it means that the main program ends and the automatic operation stops. The program cursor is in the current program segment.

M30 (end of program)

④ After the program segment containing M30 is executed, it means that the main program ends and the automatic operation stops, and the program cursor returns to the beginning of the program.

(2) use the buttons on the operation panel to stop the program, including [pause] and [reset]

Feed hold (pause)

① Press the [pause] key, the system will stop the movement of the machine tool as fast as possible and safely, stop the execution pause, and enter the feed holding state.

reset

② Press the [reset] key to immediately end the automatic operation and stop the tool deceleration.

The M99 in the main program

When the program is running, if there is M99 in the main program, after the execution of M99, it will jump to the beginning of the program to continue to execute, count plus one, and repeatedly execute the main program. If l is specified, press l times to stop

4.2 Multi Segment Operation Of MDI

In MDI mode, multi line program can be compiled and executed through MDI panel, which is called MDI multi segment running or MDI running. The program format of MDI is similar to that of normal program. MDI runs are suitable for simple test operations.

Operation Steps

1 press the [function switch] key to enter the program screen, as shown in the figure below.
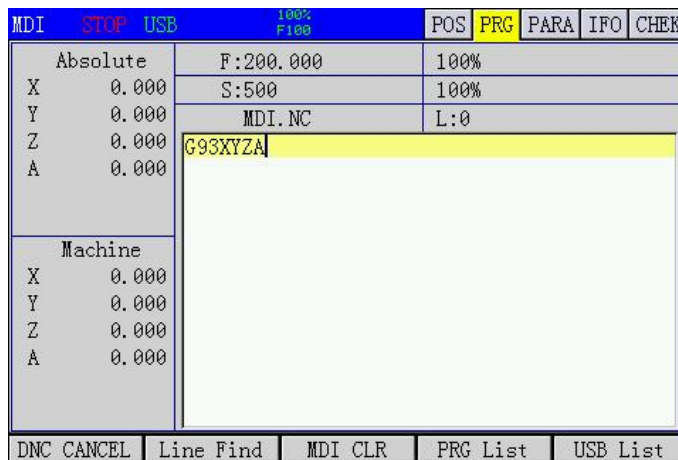
2 Press the cursor key 【↑】 or 【Reset】 to move the cursor to the beginning of the program.

3 Press the [Start] key to start the MDI operation.

Relevant explanation

Stop MDI operation

It is the same as when the program is running, but the difference is that when MDI runs to M02, M30 will respond differently (see above).

The M99 in MDI temporary program

If M99 is specified in MDI temporary program, then after M99 is executed, it will jump to the beginning of the program to continue execution and repeat the main program. After M99 is executed, the program will stop executing and jump to the beginning of the program.

Restart

After MDI is running, the operation can be stopped. When the cursor is in any position in the program, you can press the start key to start MDI operation again, and the system will re execute the program from the beginning of the program segment where the current cursor is located.

4.3 USB DNC Operation

Operation steps

1 press the [function switch] key to enter the program screen.

2 press F5 [U disk directory] key to enter the U disk directory screen.

3. Select the program you need to execute in the USB flash disk directory, and press F2 [USB DNC], DNC will be displayed in the prompt bar, and DNC status will be entered at this time.

4 press the [start] key to start the USB DNC program.

Chapter 5 Trial Operation

Before actual processing, in order to test the correctness of the processing program, the functions described in this chapter can be used to debug the processing program.

5.1 Feed Rate Override

By setting the feed rate (feed rate), the user can change the feed rate specified by the program by percentage, so as to achieve the purpose of verifying the program.

The feed rate can be controlled by the feed rate key on the system operation panel. The feed rate can be changed in the range of 10% ~ 150%, and the difference of each gear is 10%. The final value of actual feed rate = feed rate / feed rate specified by the program.

5.2 Rapid Feed Rate

The user can set the rapid feed rate (rapid rate) to temporarily reduce the rapid feed speed, which will affect the speed of all types of rapid movement in the program. include:

1 G00 fast feed;

2. Fast feed in fixed cycle;

3. Manual rapid feed;

Operation steps

Before or during automatic operation, press the fast rate key on the system operation panel to adjust the fast rate to the required percentage.

5.3 One Way Sequence Section

When the single program segment switch is on, the system will stop after one program segment is executed, and the system will stop again after the next program segment is executed. Users can execute the whole program segment by segment, which is often used to check whether the execution results of multiple program segments meet the expectations.

The switch of one-way sequence section is controlled by the [single section] key on the machine tool. This key is like a button with self-locking. When the indicator light on the key is on, it means that the one-way sequence section switch is on, and when the key is off, it means that the one-way sequence section switch is closed.

Operation steps

Before the automatic mode, press the [single segment] key to make the indicator light on the key on. After the program starts to start, the first program segment will stop.

During automatic operation, press the [single segment] key to make the indicator light on the key on. After the execution of the currently executing program segment, it will stop.

5.5 Selective Stop

With M01 instruction in the program, if the select stop function is enabled, the program will end when the program runs to M01 segment.

This function can be stopped from a complete process in a circular program.

Chapter 6 Safe Operation

6.1 power on

Before the system is powered on, confirm that:

1. The machine is in normal condition.

2. The power supply voltage meets the requirements.

Firm and correct wiring.

After the system is powered on, the boot screen will be displayed (the user can modify it by himself). At this time, the system self checks and initializes. After the self-test and initialization, the integrated position page is displayed.

6.2 shutdown

Before shutdown, confirm that:

1 The feed axis of the CNC is stopped.

2 auxiliary function is off.

3. Cut off the CNC power supply first, and then cut off the power supply of the machine tool.

6.3 over program protection

In order to avoid damage to the machine tool caused by the over travel of each feed shaft, over travel protection measures must be taken.

6.3.1 hardware over range protection

Generally, limit switches (travel switches) are installed in the positive and negative directions of each axis. The tool can only move within the range limited by the positive and negative limit switches of each axis. When the tool tries to pass the limit switch, the limit signal is valid, the system immediately stops the tool movement and displays the over travel alarm information.

In case of overtravel, the reverse moving tool (such as positive overtravel, negative moving; negative overtravel, positive moving) will leave the limit switch. The alarm can be reset and cleared after the limit is broken.

6.3.2 software over range protection

Software over range protection is similar to hardware over range protection. The positive and negative limit coordinates of software overtravel correspond to the limit switch of hardware overtravel. The positive and negative limit coordinates of each axis are respectively set in the parameters, and the range they limit is called soft limit.

When the machine tool coordinates will exceed the soft limit, the system will immediately stop the tool movement and display the over travel alarm. Manually move the tool reversely to make the machine tool of each axis sit

The alarm can be reset and the alarm can be relieved when the mark enters the limited range.

6.4 emergency operation

In the processing process, due to user programming, improper operation or product failure, some unexpected results may appear. At this time, the system must be stopped immediately. This section describes the treatment that the system can carry out in case of emergency. Please refer to the relevant

instructions of the machine tool manufacturer for the treatment of the machine tool in case of emergency.

6.4.1 reset

When the machine tool is abnormal, press the [reset] key to reset the machine tool

    1. All axis movement stops;

    2. Cooling and spindle rotation stop output;

    3 automatic operation ends.

6.4.2 emergency stop

    During the operation of the machine tool, in case of danger or emergency, the emergency stop button should be pressed. The system will immediately control the machine tool to stop moving, stop the output cooling, stop the spindle rotation, and display the emergency stop alarm.

    After releasing the emergency stop button, the emergency stop alarm is relieved and the system enters the reset state. In order to ensure the correctness of the coordinate position, after the emergency stop alarm is removed, the mechanical zero returning operation should be performed again (the machine tool without mechanical zero point shall not return to zero).

6.4.3 cut off the power supply

    During the operation of the machine tool, in case of danger or emergency, the power supply of the machine tool can be cut off immediately to prevent accidents.

    After cutting off the power supply, the coordinate displayed by the system may deviate greatly from the actual position, so it is necessary to re set the tool or return to zero.

Chapter 7 Program Editing

7.1 general

Users can edit the program directly in the system.

step

The general steps of the editing program are as follows:

1 turn on the program protection switch

2 switch to the program area of position screen or program screen

3 switch to edit mode, teaching mode or MDI (input) mode

4 use the address keys, number keys, backspace and function keys of the keyboard to insert and delete programs

explain

Program area

The program and the program display area are the edit windows. As shown in the figure below.

The steps to enter the program area of the program screen are as follows:

1 press the [program] key to enter the program screen

2 press the [edit / teach] key to enter the program area on the right. At this time, you can edit. Press [edit / teach] again and switch to teaching

Operation mode and program protection switch

The system must be switched to edit mode or MDI mode before program editing can be carried out. It is not necessary to turn on the program protection switch when editing MDI temporary programs, but it must be turned on when editing ordinary programs.

Program switch can protect program from accidental modification.

> notes
>
> In MDI mode, ordinary programs can be edited, but this is not recommended. MDI mode is generally only used for editing and performing some simple operations.
>
> 2 super large program (more than 200KB), can not be edited, can only be copied to the personal computer through U disk, edit and modify with Notepad. Then download it to the CNC system through U disk.
>
> There is no limit to the number of programs in this CNC system, which can be n programs.
>
> The storage capacity of 4 CNC system standard configuration is 512M.

7.2 Teaching Procedure

Teaching programming can easily write some programs that do not require very accurate, and can automatically input absolute coordinate values.

[edit / teach] is switched to teaching programming, and the teaching light flashes. At this time, it is turned on manually by default, or you can press [single step] to switch to the handwheel.

The axis and edit mixed key is axis selection when the manual light is on, and it is also axis selection in case of single handwheel mode, and editing key if it is hand-held handwheel.

Teaching programming must be absolute programming.

If you press the I, Z keys, the current coordinates will be automatically added.

If [ESC] is pressed, all valid axis positions will be input at one time.

When teaching programming and arc programming, only 3-point circle interpolation instruction G12 can be used. Because the relative coordinates cannot be entered.

During teaching, there is no difference between the invalid axis key and the editing mode, which can be input normally.

7.3 new program

EDIT STOP USB	100% F100	POS	PRG	PARA	IFO	CHEK

NC Size
Left Size: 240M   PRG: 609D测试.NC
0M/240M

1	00000.nc	0B
2	609MT.NC	5KB
3	000 New File	
4	111	
5	222 FXX	
6	333	
7	609 OK ESC	
8	609D测试.NC	5KB
9	G02圆.TAP	46B
10	G03圆.TAP	46B

| << | New | DEL | SAVE AS | Export |

EDIT STOP USB	100% F100	POS	PRG	PARA	IFO	CHEK

NC Size
Left Size: 240M   PRG: FXX.NC
0M/240M

1	00000.nc	0B
2	609MT.NC	5KB
3	0000.NC	102B
4	1111.NC	343B
5	2222.NC	7B
6	3333.NC	334B
7	609MT^1.NC	5KB
8	609D测试.NC	5KB
9	FXX.NC	0B
10	G02圆.TAP	46B

| << | New | DEL | SAVE AS | Export |

step

1 press the [program] key to enter the program screen.

2 press the [local directory] soft function key to enter the local directory screen.

3. Press the [new program] soft function key to open the dialog box of new processing program, and input the file name FXX of processing program. As shown in Figure a.

4 press enter to confirm. As shown in Figure B, FXX appears in the directory on the leftNC file name. If it is a subroutine, please name it oxxxxx, which means pure number.

5 press the left most soft function key to return to the main program screen.

7.4 Insert One Line Program Segment
step

1 in the program editing state, press the cursor keys [←] and [→] to move the cursor to the front of the program line to be inserted.

2 press the [Enter] key to change the line. The original program is moved down and a new line is edited up

3 use the address key, number key, backspace and other function keys of the keyboard to edit the program.

explain

1 [backspace] key is to delete the previous character.

2 the [delete] key is to delete the next character.

3 the [insert] key is used to switch between the insert and edit modes.

PART 3 PROGRAMMING INSTRUCTIONS

Introduction To The First Chapter Of Programming

There are two ways to command axis movement: absolute value command and increment value command.

1.1 absolute value instruction

The absolute value command is programmed using the coordinate value of the end position of the axis movement. That is, the coordinate position of the tool moving to the end point. As shown in Figure 2-1
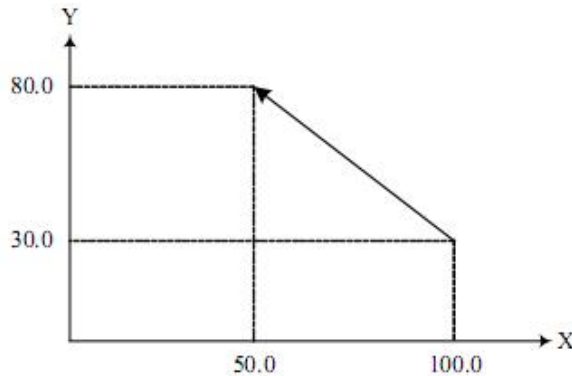
The tool moves quickly from the starting point to the end point, and is programmed as G90 x50.0 y80.0 with the absolute value command;.

1.2 Increment Value Instruction

The increment command is programmed directly using the amount of axis movement. The coordinate value of the tool movement position is relative to the previous position, not to the fixed origin of the machining coordinate system, that is, the actual moving distance of the tool. As shown in Figure 2-1: the tool moves quickly from the starting point to the end point, and is programmed with increment value instruction as: G91 x-50.0 y50.0;

Note:

> In the same processing program, users can use G90 or G91 to switch the instruction mode according to their needs. G90 / G91 is a group of mode g codes. After one instruction is given, the mode is valid until another G code in the same group is instructed.

1.3 decimal point programming

Values can be entered with a decimal point. Decimal points can be used for instruction values that represent units of distance, time, and speed, as shown below.

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F, E, H

Depending on the address and command, the decimal point can be in millimeters, degrees or seconds. Note that x1.0.000 is different from other controllers.

Chapter 2 Composition Of Procedure

2.1 procedure

The program is composed of several program segments, which are composed of words, and each program segment is separated by a segment end code.

2.1.1 program number

N programs can be stored in the memory of the system, which can be distinguished by the program number composed of address O and the following four digits (the program name imported from U disk can be Chinese). The program starts with a program number and ends with M30 or M02.

format

```
OXXXX
   O : Program number address character.
   XXX: program number (1 ~ 9999, leading zeros can be omitted)
```

2.1.3 program number and program segment

A program is composed of multiple program segments. Segment Terminator (;) between segmentsseparate. At the beginning of the program segment, the address N and the following five digits can be used to form the sequence number, and the leading zero can be omitted.

format

```
NXXXXX
   N : Program number address character.
   Xxxxx: sequence number (leading zeros can be omitted)
```

explain

The sequence number can be arbitrary and the interval can be unequal. Sequence numbers can be inserted in all program segments or only in important segments. It is convenient to carry the sequence number in the important place of the program.

2.1.4 skip optional segments

In automatic operation, the program section with a slash (/) at the beginning is skipped by the system when the skip switch is on. If the trip switch is off, the block will not be skipped.

example

```
N100 X100.0 ;
/N101 Z100.0 ;
N102 X200 ;
```

2.1.5 Words and Addresses

The numeric elements of the program can be composed of the numeric and numeric elements.

format

```
X1000
   X: address
   1000: value
```

explain

The address is a letter in the English letters (A-Z), which specifies the meaning of the values after it. According to different preparation functions, sometimes the same address has different meanings. In this system, the address that can be used and its meaning are shown in the table below.

function	address	significance
程序号	O	program number
顺序号	N	Sequence number
准备功能	G	Specify the action state (line, arc, etc.)
尺寸字	XYZABUVW	Coordinate axis movement command

	R	Arc radius
	I J K	The coordinate of the arc center, the middle point of the G12 arc.
进给速度	F	Feed speed designation
主轴功能	S	Spindle speed specification, S spindle 1
刀具功能	T	Designation of tool number
辅助功能	M	Machine tool auxiliary function assignment
暂停	P/X	Designation of Pause Time
参数	P/Q/R	Canned cycle parameters

2.1.6 base address and instruction value range

The base address and instruction value ranges are shown in the table below

function	address	Mm input
program number	O	1～9999
Sequence number	N	unlimited
prepare function	G	0～99
size word	XYZABCUVWI J K Q R	±999999.999
feed per minute	F	0.001～15000.0
Spindle function	S	0～9999
Accessibility	M	0～99
pause	X P	0～999999.999S

2.2 end of procedure

The program ends with M30 or M02.

format

```
M30; End of procedure
```

explain

In the execution program, if the above program code is encountered, the system will end the program execution and enter the reset state.

At the end of M30, whether the program cursor returns to the beginning of the program is controlled by the bit parameter M30.

Chapter 3 Preparation Function (G Code)

The preparation function is represented by G code, including G address and its subsequent value.G code includes modal and non modal.

type

Type 1: modeless G code
Only valid in the program segment being instructed
Type 2: modal G code
Valid until other G code instructions in the same group

example
G00 and G01 are the same group of modal g codes.The processing procedure is as follows:
G00X__; (G00 is valid)
Y__;   (G00 is valid)
G01Z__; (G01 is valid)
X__;   (G01 is valid)

3.1 G code list

G code	level	function
G00 *	01	Fast positioning, the speed runs according to the speed parameter G0 speed, linear and non-linear can be selected in the
G01		comprehensive parameters
G02		Clockwise circular interpolation, the speed is given by F, if there is a non-planar axis, it will run according to helical interpolation
G03		Counterclockwise circular interpolation, others are the same as above
G12		Circular interpolation through intermediate point
G04	00	Delay waiting, parameter X is in seconds, parameter P is in milliseconds, resolution is 5 milliseconds
G17 *	02	Circular interpolation plane selection XY
G18		Circular interpolation plane selection ZX
G19		Circular interpolation plane selection YZ
G28	00	Circular interpolation plane selection YZ
G52	00	local coordinate function
G53		Machine tool coordinate positioning
G54 *	06	Workpiece coordinate system 1
G55		Workpiece coordinate system 2
G56		Workpiece coordinate system 3
G57		Workpiece coordinate system 4
G58		Workpiece coordinate system 5
G59		Workpiece coordinate system 6
G73	07	High-speed deep hole machining cycle
G80 *		Canned cycle cancellation
G81		Drilling cycle (spot drilling cycle)
G82		Drilling cycle (boring step empty cycle)
G83		Deep hole drilling cycle

G85		boring cycle
G86		Drilling cycle
G89		boring cycle
G90 *	08	Absolute value programming
G91		Incremental programming
G92	00	Set the workpiece coordinates. When there is no mechanical zero return, all workpiece coordinates will be offset.
G93		Set the machine coordinates, if there is a software limit, please use it with caution.
G98 *	9	Canned cycle returns to the initial plane
G99		Canned cycle returns to point R

Note:

1 The G code marked with * is the default G code of the system. When the power is turned on, the modal G code will be in the default state.

2 The G codes in group 00 are non-modal G codes and are only valid in the current block.

3 An alarm occurs if a G code not listed in the G code list or not enabled is used.

4 Several different groups of G codes can be commanded in the same block. If multiple G codes of the same group are commanded in the same block, the last G code is valid.

3.2 G00 Quick Positioning

G00 is a fast positioning command. It starts from the current point and moves to the specified position according to the speed parameter G0.

Instruction format

G00IP__;

Semicolon (;):Indicates the end of the segment.

Instructions

Non linear interpolation positioning

Optional and non-linear parameters are synthesized in straight line.

When G00 is executed, each axis is positioned at an independent fast moving speed. If x and Z axes are specified at the same time, the tool path is usually not a straight line. In other words, in the process of G00, moving the axis at the same time does not do linear interpolation, but selects the optimal path. It is usually used in stepping motor system.

Linear interpolation positioning

Linear and non-linear can be selected in the comprehensive parameters

When G00 is executed, the tool path is the same as that of G01, and the tool is positioned in the shortest time with the speed not greater than the rapid movement of each axis.

The two methods are defined in the parameter (P0028), and the subroutine can only be non-linear.

The schematic diagram of the two positioning modes is as follows:
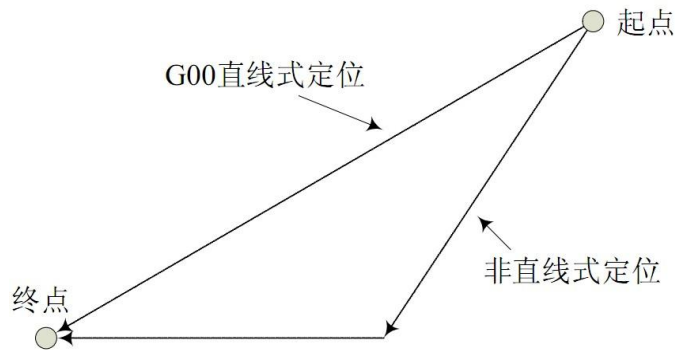
1. The fast moving speed of each axis of G00 is set by parameter, and the feed speed specified by F is invalid. The speed of G00 can be divided into 100%, 50%, 25% and F0.

2. When G00 is a modal instruction and the next instruction is also G00, it can be omitted. G00 can be written as G0.

3. Pay attention to the safe position of the tool when G00 is ordered to avoid hitting the tool.

3.3 G01 Linear Interpolation

G01 is a linear interpolation command. It takes the current point as the starting point, uses IP to specify the end point and F to specify the speed.

In the subroutine, it moves approximately in a straight line.

Instruction format

G01 IP__ F__ ;
   F: tool feed rate.

Instructions

The feed rate specified by F is always valid until a new value is specified, so it is not necessary to specify each block one by one.

The speed specified by F is the resultant speed of the tool moving along a straight line.

give an example
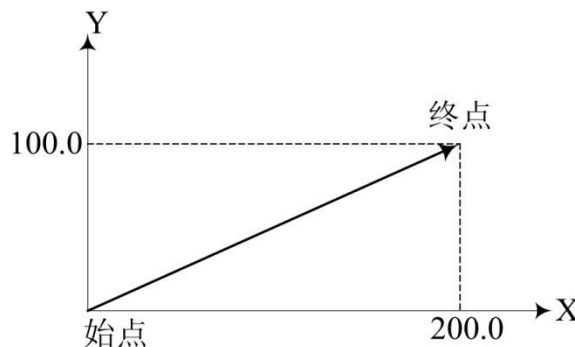
G91 G01 X200.0 Y100.0 F200.0

The tool moves from the start point (0, 0) to the end point (200.0, 100.0) at a speed of 200mm/min.

3.4 G02 / G03 — circular interpolation

G02 / G03 are arc interpolation commands. They control the cutting motion of the tool along the arc on the specified plane.

The subroutine cannot run this instruction.

Instruction format

Arc of XY plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_ Y\_\_ \begin{Bmatrix} R\_\_ \\ I\_\_ J\_\_ \end{Bmatrix} F\_\_$$

Arc of XZ plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_ Z\_\_ \begin{Bmatrix} R\_\_ \\ I\_\_ K\_\_ \end{Bmatrix} F\_\_$$

Arc ofYZ plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\_ Z\_\_ \begin{Bmatrix} R\_\_ \\ J\_\_ K\_\_ \end{Bmatrix} F\_\_$$

project	specified content	Order	describe
1	Plane designation	G17	XY plane arc designation
		G18	ZX plane arc designation
		G19	YZ plane arc designation
2	Interpolation direction	G02	Clockwise circular interpolation (CW)
		G03	Counterclockwise circular interpolation (CW)
3	End position or distance	Two axes in X Y Z	End position in absolute coordinate system
		Two axes in XYZ under U V W or G91	The distance from the start point coordinates to the end point coordinates
4	Center position or radius	Two axes in I J K	The distance from the coordinates of the starting point to the coordinates of the center of the circle (I J K is also calculated incrementally in absolute mode)
		R	Arc radius
5	Feed rate	F	Tangent speed of arc feed

Instructions

Direction of arc interpolation

The so-called clockwise (G02) and counter clockwise (G03) means that in the right-hand rectangular coordinate system, for X_Y_The plane (Z - x - plane, y - Z - plane) from Z_The positive direction of the axis (Y axis, X axis) is shown in the following figure
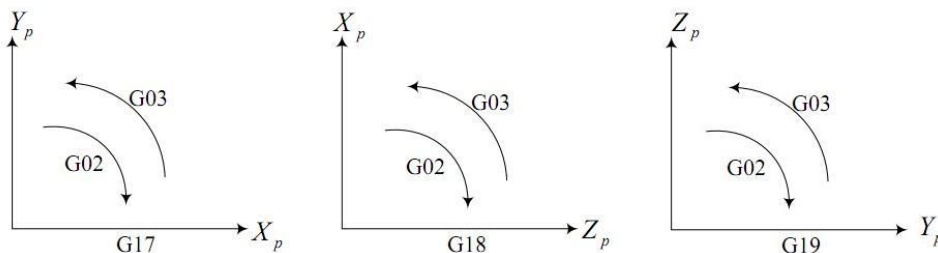
The amount of movement on the arc

With the address x \Y_Or Z_Specify the end point of the arc. The absolute value is indicated under the G90 command, and the incremental value is indicated under the G91 command.The increment value is the distance from the start point to the end point of the arc.

Center of arc

The center of the arc is specified by the addresses I, J, and K, which correspond to X - u, X - u, and K, respectivelyY_、Z_、。1. The value after J and K is the vector component from the starting point of the arc to the center of the circle, and is the increment value with sign.As shown in the figure below:
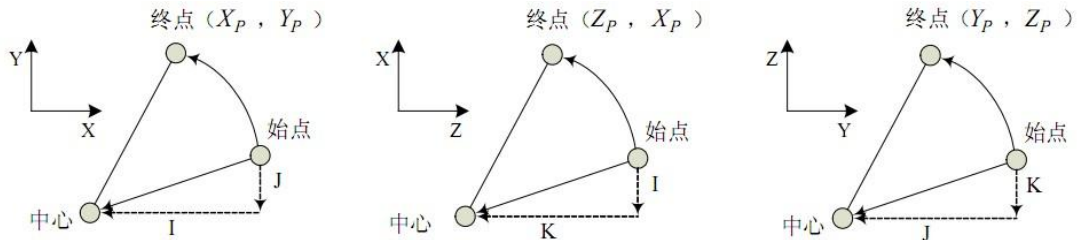
Arc radius
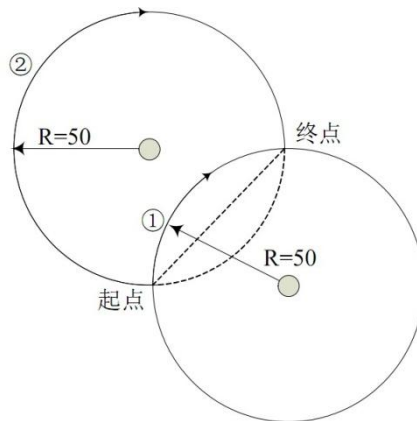
When the arc of ① is less than 180º, execute G code G91 G02 X60 Y50 R50 F300;

When the arc of ② is greater than 180º, execute G code G91 G02 X60 Y50 R-50 F300;

The feed rate

The feed rate of circular interpolation is specified by F, which is the speed of the tool along the tangent direction of the arc.

notes

When I, J and K are 0, they can be omitted.

2. If the movement of all arcs (x, y, z) is ignored, the end point is the same as the starting point. If I, J, K are used to specify the center of the circle, then a whole circle is specified. At the same time, the position of the starting point of R is not generated, that is, the position of the starting point is not generated.

4. The error of the actual moving speed of the tool relative to the specified speed is within ± 2%, and the specified speed is the speed of the tool moving along the arc after radius compensation.

5 when I, J, K and R are specified at the same time, R is valid, I, J, K are invalid.

give an example
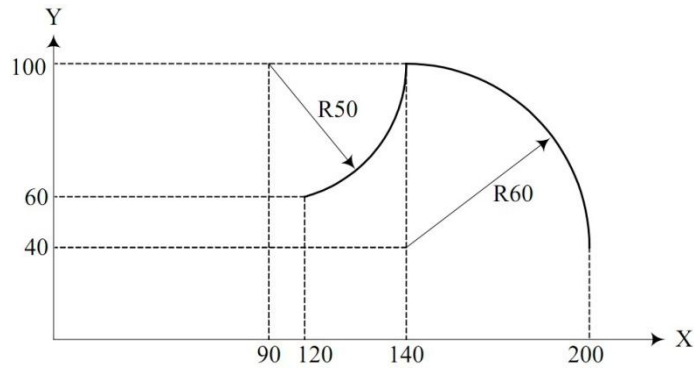
The traces on the graph are programmed in absolute value mode and incremental value mode respectively.

```
Absolute way
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 I-60.0 F300.0 ;
G02 X120.0 Y60.0 I-50.0 ;
或
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300.0 ;
G02 X120.0 Y60.0 R50.0 ;
Incremental way
G91 G03 X-60.0 Y60.0 I-60.0 F300.0 ;
G02 X-20.0 Y-40.0 I-50.0 ;
或
G91 G03 X-60.0 Y60.0 R60.0 F300.0 ;
G02 X-20.0 Y-40.0 R50.0 ;
```

Helical interpolation

If the axis out of the specified plane is commanded while the arc interpolation is specified, the tool spiral motion.

Instruction format

Arc of XY plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_ Y\_ Z\_ \begin{Bmatrix} R\_ \\ I\_ J\_ \end{Bmatrix} F\_$$

Arc of ZX plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_ Z\_ Y\_ \begin{Bmatrix} R\_ \\ I\_ K\_ \end{Bmatrix} F\_$$

Arc of YZ plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_ Z\_ X\_ \begin{Bmatrix} R\_ \\ J\_ K\_ \end{Bmatrix} F\_$$

explain

The f command specifies the feed rate of the circle projected along the arc, as shown in the figure below. The feed rate of the linear axis (Z axis) is:

$$F \times \frac{\text{Length of linear axis}}{\text{arc length of arc projection}}$$
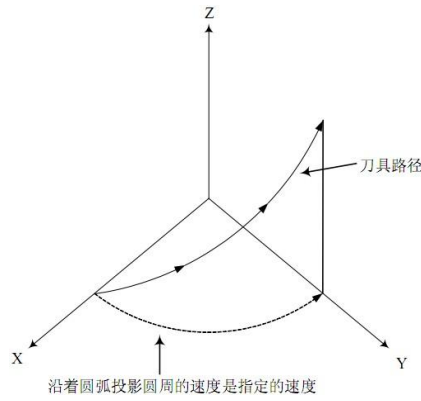
When the feed rate f is specified, the speed of the linear axis should not exceed any limit value.

3.5 g12-3 point circular interpolation

G12 through the middle of the arc interpolation command.They control the cutting motion of the tool along the arc on the specified plane.

The subroutine cannot run this instruction.

Instruction format

Arc in the XY plane
G12 I J X Y
Arc in the ZX plane
G12 I K X Z
Arc in the YZ plane
G12 J K Y Z

I, J, K are the middle point of the arc, under G90 is absolute, under G91 is relative.

X, Y, Z are the end points of the arc, absolute under G90, relative under G91.

This instruction is mainly used for teaching programming.

The middle point is best to take the middle point of the arc to reduce the calculation error. Note that the starting point, the middle point and the end point cannot be collinear.

3.6 G04 - delay waiting

The execution of the next program segment can be delayed by using the delay wait instruction, and the delay time is the instruction time.

Instruction format

G04 X__ ;
or
G04 P__ ;
X: Delay wait time setting (decimals can be used).
P: Delay waiting time setting (decimal not allowed).

Instructions

With the pause instruction, the execution of the next program segment can be delayed for a specified period of time.

Instruction word	Scope of instruction	Command unit
X	0.001～99999.999	second
P	1～99999999	0.001second

Note

1 The X/P command unit has nothing to do with the smallest unit of the linear axis or rotary axis.

2 If the P and X commands are omitted, it can be regarded as an exact stop.

3 Executing G04 command will automatically disable pre-reading and buffering.

3.7 reference point function

The so-called reference point is a specific position on the machine.When there is a mechanical zero point, the mechanical zero point is the reference point of the machine tool;When there is no mechanical zero point, the set floating zero point can also be regarded as the reference point of the machine tool. It can return to the reference point under the manual and mechanical return to zero mode, or use G28 command to make the tool return to the reference point automatically.

G28 - automatic return to reference point

The reference point can be automatically returned by G28 to the reference point.After returning to the reference point, the return to zero light is on.As shown in the figure below.

A：返回参考点的起点
B：中间点
C：从参考点移动的目标点
R：参考点
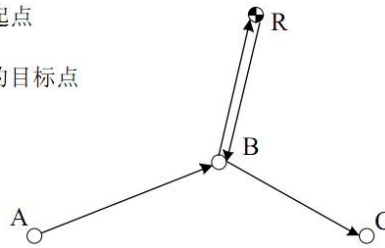
G28自动返回参考点：A—>B—>R
G29从参考点移动：R—>B—>C

command format

G28IP__;

IP__:The coordinate of the intermediate point passed by when automatically returning to the reference point, specified by absolute or incremental value.

Instructions

1. During G28 execution, the intermediate point and reference point are located at the speed of program return to zero.

2.When the machine tool is locked, G28 cannot locate from the middle point to the reference point, and the zero return lamp will not be on.

3. G28 is usually used in automatic tool change, so in principle, the compensation such as tool radius and length should be cancelled in advance.

4. in G28, there are several axes in the order of Z > XY > ABC

example

N1 G28 X40.0 ;   Midpoint (40.0)

N2 G28 Y60.0 ;   Midpoint (40.0, 60.0)

notes

1 After the power is turned on, if the manual reference point return is not performed once, when G28 is commanded, the movement from the intermediate point to the reference point is the same as that of manual reference point return.

2 After passing through the intermediate point to the reference point with the G28 command, when the workpiece coordinate system is changed, the intermediate point also moves to the new coordinate system.

3 If the Z axis is specified, the Z axis returns first.

3.8 coordinate system function

When the machine tool is working, the tool moves to the specified position according to the coordinate specified by the machining program, and the coordinate value is specified by the axis components of the coordinate axis.As shown in the figure below is the tool position specified by x40.0 y50.0 z30.0.
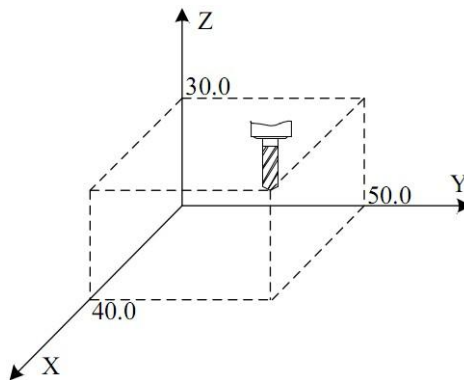
This system uses one of the machine tool coordinate system, workpiece coordinate system and local coordinate system to specify the coordinate position.

Machine zero is a fixed reference point set by the machine tool manufacturer for the machine. The coordinate system with the machine zero as the coordinate origin is called the machine coordinate system.

1 The machine tool coordinate system takes the fixed point on the machine tool as the origin and is the datum of other coordinate systems. Once established, the system will save it and it will be valid until it is reset.

2 The workpiece coordinate system is based on the sub-coordinate system in the machine tool coordinate system, and its position in the machine tool coordinate system can be set and changed.

3 The local coordinate system is based on the sub-coordinate system in the workpiece coordinate system, and its position in the workpiece coordinate system can be set and changed.

The relationship of each coordinate system is as

follows: 参考点
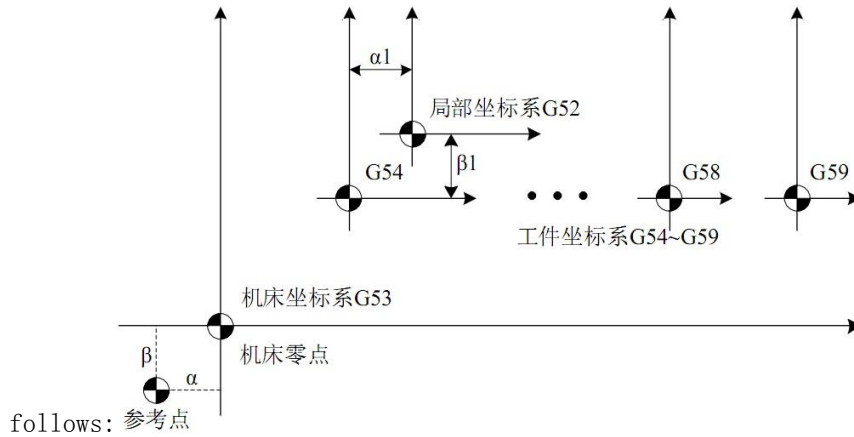
After manually returning to zero for each coordinate axis or G28 command to return to the reference point, the system can establish the machine coordinate system according to the zero point of the machine tool. This coordinate system will be saved in the system until you reset it.

3.8.1 G53 positioning of machine tool coordinate system

According to the specified machine coordinate, quickly move the tool to the target position.

Instruction format

> G53IP__;
>
> IP__: The absolute coordinates of the target point in the machine coordinate system

Instruction description

 1 Since the general positioning command (G00) can only specify the target point in the workpiece coordinate system, if the user wants to move the tool to a special position of the machine tool (such as the tool change position), it is more convenient to use the G53 command.

 2 G53 is a non-modal G code, which is only valid in the current block.

 3 The G53 command must be an absolute command. If it is an incremental command, an alarm will be generated.

 notes

> 1 When the G53 command is specified, the compensation amount such as tool radius and length will be automatically cleared.
>
> 2 G53 command suppresses G code pre-reading.

3.8.2 G92, G54-G59 - workpiece coordinate system setting

 The coordinate system used in machining parts is called workpiece coordinate system. The workpiece coordinate system needs to be set in advance before machining. It can also be changed by moving the origin.

 There are three ways to set the workpiece coordinate system

 1 G92 sets the workpiece coordinate system;

 2. Set workpiece coordinate system automatically;

 3 select G54-G59 workpiece coordinate system.

3.8.2.1 G92 - set workpiece coordinate system

Instruction format

> G92IP__ ;
>
> IP__: Specify the coordinates of the current point in the set workpiece coordinate system.

The way of establishing the workpiece coordinate system is to make the specified coordinate value (IP__) become the absolute coordinate value of the point (such as the tool nose) on the current tool in the set workpiece coordinate system.

1 Under the condition of tool length compensation, when the coordinate system is set with G92, the specified coordinate value (IP__) is the position before tool compensation.

2 For tool radius compensation, when G92 is commanded, the compensation temporarily disappears.

Normally, please set the workpiece coordinate system before specifying tool compensation.

give an example

以刀尖为指定点设定工件坐标系
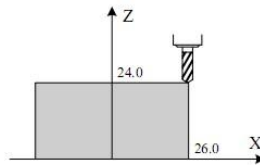
G92 X26.0 Z24.0 ;

以刀架基准点为指定点设定工件坐标系
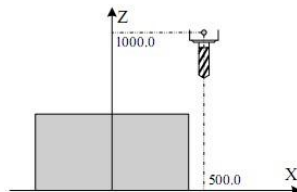
G92 X500.0 Z1000.0 ;

After the coordinate system is established, in absolute mode, when the commanded reference point moves to the specified position, tool length compensation must be added, and the compensation value is the difference between the reference point and the tool nose.

3.8.2.2 automatic setting of workpiece coordinate system

If the coordinate system automatic setting function is selected, the system will automatically set the workpiece coordinate system after returning to the reference point manually or automatically. If α 、 β 、 γ When the reference point is returned, the absolute coordinate value of tool rest reference point or tool tip position is X=α,Y=β,Z=γ 。This sets the workpiece coordinate system. This method is equivalent to executing the following instruction setting at the reference point.

G92 Xα Yβ Zγ ;

3.8.2.3 Select workpiece coordinate system (G54-G59)

The system provides six workpiece coordinate systems G54-G59. The user can set the workpiece zero offset data of each coordinate system through the system MDI panel, and then select any workpiece coordinate system. When the machine is turned on and the reference point is returned, g54 coordinate system is selected by default.

G54 Workpiece coordinate system 1
G55 Workpiece coordinate system 2
G56 Workpiece coordinate system 3
G57 Workpiece coordinate system 4
G58 Workpiece coordinate system 5
G59 Workpiece coordinate system 6

Instructions

The six workpiece coordinate systems are set according to the distance (workpiece zero offset) from the machine zero point to the respective coordinate system zero point, as shown in the figure below.
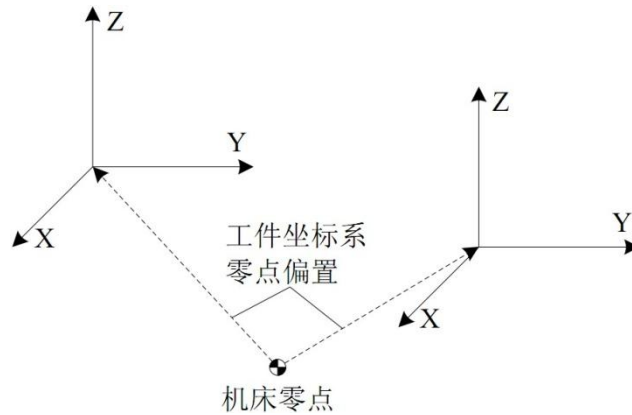
2 After returning to the reference point, the absolute position is the negative value of the workpiece zero offset, as shown in the figure below.
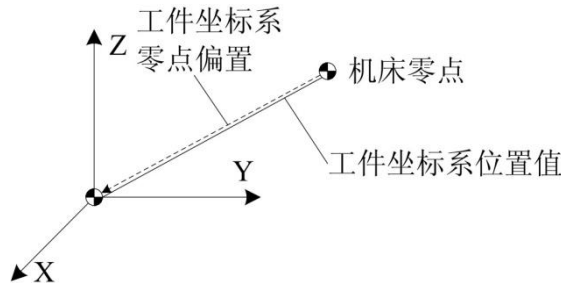
3 If the work coordinate system is selected, it is generally set to zero in the parameter. If the set value is not zero, after returning to the parameter reference point, all workpiece coordinate systems will offset the parameter set value.

Note
1 When the workpiece coordinate system is selected, it is generally not necessary to set the coordinate system with G92. If G92 is used to set, the workpiece coordinate system 1~6 will be moved. Therefore, do not mix G92 with G54~G59 unless you want to move the workpiece coordinate system 1~6.
2 Whether the relative position changes with the setting of the workpiece coordinate system depends on the corresponding setting in the parameters.

give an example
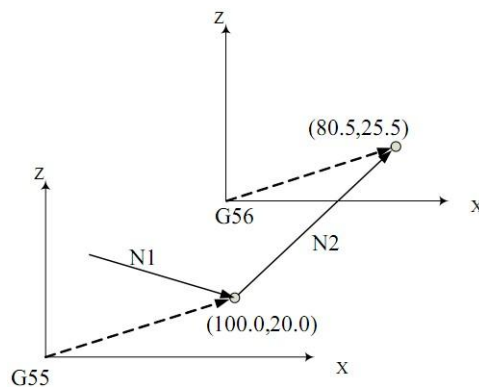
```
Machining program
   N10 G55 G00 X100.0 Z20.0 ;
   N20 G56 X80.5 Z25.5 ;
```

3.8.3 Move the workpiece coordinate system with G92

Instruction format

```
G92 IP_ ;
   IP__: Specify the coordinates of the current point in the set workpiece coordinate system.
```

Instruction interpretation

When G92 instruction is executed in the selected workpiece coordinate system (G54-G59), all the original workpiece coordinate systems can be offset synchronously to generate a new coordinate system. All workpiece coordinate systems have the same offset.

3.8.4 setting machine coordinates (G93)

Set the coordinates of the current machine tool, please use it with caution when there is a software limit. The workpiece coordinate system will be completely offset.

G93 IP_

Example: G93Z0; set the current machine coordinate of Z axis to zero.

3.8.5 G52 local coordinate system

When programming in the workpiece coordinate system, another sub coordinate system can be set in the workpiece coordinate system for convenience. This sub coordinate system is called a local coordinate system.

Instruction format

```
   G52 IP_ ;Set the local coordinate system
   ...
   G52 IP0; Cancel the local coordinate system
   IP__: Specifies the absolute coordinates of the origin of the local coordinate system in the
workpiece coordinate system.
```

explain

When setting the local coordinate system, the specified IP__Whether it is absolute or relative, its value represents the absolute coordinate of the origin of the local coordinate system in the workpiece coordinate system. At the same time, the absolute coordinates displayed in the system interface are also the coordinates in the local coordinate system. In the workpiece coordinate system, the local coordinate system can be changed by specifying the new zero point of the local coordinate system with G52.

Once the local coordinate system is specified with G52, the local coordinate system will remain valid in its corresponding workpiece coordinate system until the command "G52 IP" makes the zero point of local coordinate system consistent with the zero point of workpiece coordinate system.

Unlike G92 instruction, G52 only works in its corresponding workpiece coordinate system, as shown in the figure below.
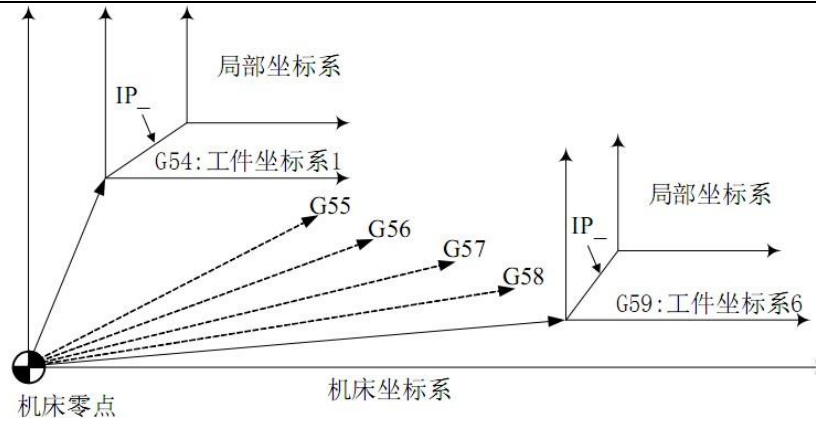
> Note
>
>   1 When an axis returns to the reference point automatically or manually, the zero point of the local coordinate system of this axis is consistent with the zero point of the workpiece coordinate system, that is, the local coordinate system is cancelled. This has the same effect as the command G52 α; (α: axis that returns to the reference point).
>
>   2 The setting of the local coordinate system does not change the workpiece coordinate system and the machine tool coordinate system.
>
>   3 Whether to clear the local coordinate system during reset depends on the parameter setting.
>
>   4 When the workpiece coordinate system is set with G92, the local coordinate system is canceled. If the coordinate values of all axes are not specified, the local coordinate system of the axis for which the coordinate value is not specified is not canceled, but remains unchanged.
>
>   5 G52 temporarily cancels tool radius compensation.
>
>   6 After the G52 block is executed, the absolute coordinates display the coordinates in the local coordinate system immediately.

3.8.6 G17 / G18 / G19 - plane selection

Use G code to select the plane of arc interpolation and the plane of tool radius compensation.

Instruction format

G17. XY plane
G18. ZX plane
G19. YZ plane

G17, G18, G19 in the program section without command, the plane does not change.

example
G18 X_ Z_ ; ZX plane
X_ Y_ ; plane unchanged (ZX plane)

Also, the move command is independent of plane selection. For example, in the case of the following command, the Z axis is not on the XY plane, so the Z axis movement is independent of the XY plane.

G17 Z_   ;

3.9 Simplified programming functions

3.9.1 general

In drilling process, it is usually necessary to use multiple program segments to specify several processing actions with high frequency. The fixed cycle introduced in this chapter can be used to complete various ways of drilling with one-way sequence segment containing a G code, which simplifies the programming operation.

List of drilling preparation functions

code	Opening action	Hole bottom action	Retraction action	purpose
G73	intermittent feed	－－	Rapid feed	High-speed deep hole machining cycle
G80	－－	－－	－－	Cancel a canned cycle
G81	cutting feed	－－	Rapid feed	drill, drill
G82	cutting feed	－－	Rapid feed	Drilling and boring stepped holes
G83	intermittent feed	－－	Rapid feed	Deep hole machining cycle
G85	cutting feed	－－	cutting feed	boring
G86	cutting feed	Spindle stop	Spindle stop	boring
G89	cutting feed	－－	cutting feed	boring

Instruction format

G__ IP__ R__ Q__ P__ F__J__;		
Specified content	address	explain
Hole processing method	G	Select canned cycles G73, G74, G80 to G89.
Hole machining direction		Fixed Z
Hole position data	Non-hole machining direction axis address in IP	Specify the hole position by absolute value or incremental value, and the control is the same as that of G00 positioning.
Hole machining data	IP center hole machining direction address	As shown in the figure below, specify the distance from point R to the bottom of the hole with an incremental value or specify the coordinate value of the bottom of the hole with an absolute value. The feed speed is the speed specified by F in action 3, and in action 5, it is the speed specified by the rapid feed or F code according to the hole machining method.
	R	As shown in the figure below, specify the distance from the initial point plane to point R with an incremental value, or specify the coordinate value of point R with an absolute value. The feed rate is all rapid traverse in actions 2 and 6.

Q	Specify the cutting amount for each time in G73 and G83 or the translation amount (incremental value) in G76 and G87.
P	Specifies the pause time at the bottom of the hole. The relationship between the time and the specified value is the same as that specified by G04.
F	Specifies the cutting feed rate.
J	Machining axis specification, J0: X, J1: Y, J2: Z, J3: A other values or do not specify the default Z axis

Instruction interpretation

Absolute programming and relative programming

Use G90 and G91 to specify absolute programming and relative programming.

G90 (absolute value instruction)	G91 (increment value instruction)

return point plane

① The instruction g98 returns to the initial point plane.

② The instruction G99 returns the r-point plane.

Usually, G99 is used for initial hole processing and g98 is used for final machining. When the hole is machined with G99 state, the plane of the initial point does not change

G98 (return to initial point plane)	G99 (return to r-point plane)

Hole processing method

The optional fixed cycle instructions for hole machining include: G73, G74, G76, G80 ~ G89, all of which are modal g codes.

The data, the data and the data of a hole are fixed, and the data and program are fixed.

Once the hole processing method and data are instructed, they remain valid until the G codes (g80 and 01 group G codes) for canceling the fixed cycle are specified. Therefore, when the same hole processing is carried out continuously, it is not necessary to specify the hole processing method and data in each

program segment. At the beginning of the fixed cycle, all the necessary hole machining data are assigned, and in the subsequent fixed cycle, only the changed data need to be specified.

notes

The cutting speed (F command) of the fixed cycle is still maintained after the fixed cycle is cancelled.

The cancellation of fixed cycle

Fixed loop can be cancelled by using group 01 code or G80 in the same group as fixed loop. 01 group G code includes: G00, G01, G02, G03.

3.9.2 G73 - high speed deep hole processing cycle

G73 cycle is a high-speed deep hole drilling cycle, performing intermittent feed until the bottom of the hole.

Instruction format

G73 IP_ R_ Q_ F_ J_;
IP_ (non-hole machining axis): hole position data
IP_ (hole machining axis): distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
Q_:Feed rate per cutting feed
F_:feed rate
J_:Machining shaft

Instruction interpretation

High speed deep hole drilling cycle along the drilling axis intermittent feed, to the bottom of the hole, fast return. This cycle is beneficial to chip removal and improve drilling speed and accuracy.

notes

1. The tool withdrawal amount D can be set by parameters, and the drilling axis direction can be fed intermittently. In order to make deep hole machining easy to chip removal. In this way, the work efficiency can be increased. The tool retraction movement adopts fast movement.
2 start spindle rotation before G73 is specified.
3 In canned cycle state, if any one or more of X, Y, Z, R data is commanded, the system will perform hole machining. But when X and G04 are specified at the same time, hole machining is not performed.
4. In the program section which can carry out the hole machining action, we can command the hole processing data Q and P. In the program section that can not be processed, the data Q and P of the instruction hole processing can not be stored as modal data.
5 in the fixed cycle mode, if the tool length offset has been instructed, the offset will be performed when the initial point plane is positioned.
Tool offset command is invalid in 6 fixed cycle.
7 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.3 g81 - drilling cycle, point drilling cycle

G81 is a general drilling cycle instruction.

Instruction format

G81 IP_ R_ F_ J_;
IP_ (hole machining axis): distance from R point to hole bottom (increment value) or coordinate

of hole bottom (absolute value)

R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)

F_:feed rate

J_:Machining shaft

Instruction interpretation

After positioning, the tool quickly moves to the R point, drills to the bottom of the hole along the drilling axis direction, and then the tool quickly returns.

Note

1 Before specifying G81, start the spindle rotation first. If G81 and M code are specified in the same block, the M code is sent at the initial positioning, and the next cycle operation is performed after the execution of the M code is completed.

2 In the canned cycle state, if any one or more of X, Y, Z, R data is commanded, the system will perform hole machining. But when X and G04 are specified at the same time, hole machining is not performed.

3. In the fixed cycle mode, if the tool length offset has been instructed, the offset will be carried out when the initial point plane is positioned.

4 tool offset command is invalid in fixed cycle.

5 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

example

M04 S800 ;Spindle rotation

M04 S800 ;   Spindle rotation

G90 G99 G81 X100. Y100. Z-20. R50. F100. ;   Locate, drill 1 hole and return to R

X150. ;   Locate, drill 2 holes and return to R

G98 Y150. ;   Positioning, return to the initial plane after drilling 3
holes

G80 G00 X0. Y0. Z0. ;   Cancel the canned cycle and return to the starting point of
processing

M05 ;   Spindle stopped

3.9.4 G82 - drilling cycle, boring step hole cycle

G82 is a general drilling cycle instruction. The tool returns after the hole bottom is suspended. Because the hole bottom is suspended, the accuracy of hole depth can be improved in the process of blind hole processing.

Instruction format

G82 IP_ R_ P_ F_ J_;

P_ (non-hole machining axis) hole position data

IP_ (hole machining axis) the distance from point R to the bottom of the hole (incremental value) or the coordinates of the bottom of the hole (absolute value)

R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)

P_:Hole bottom pause time (unit: 0.001 s)

F_:feed rate

J_:Machining shaft

G82(G98)	G82(G99)

Instruction explanation

After positioning, the tool quickly moves to the R point, drills to the bottom of the hole along the drilling axis direction, pauses time p, and then the tool quickly returns.

Note
1 Before specifying G82, start the spindle rotation first. If G82 and M code are specified in the same block, the M code is sent at the initial positioning, and the next cycle operation is performed after the execution of the M code is completed.
2 In the canned cycle state, if any one or more of X, Y, Z, R data is commanded, the system will perform hole machining. But when X and G04 are specified at the same time, hole machining is not performed.
3 The hole machining data Q and P can be commanded in the block that can perform hole machining. The commanded hole machining data Q and P cannot be stored as modal data in a block where hole machining cannot be performed.
4 In the canned cycle mode, if the tool length offset has been commanded, the offset will be performed when the initial point plane is positioned.
5 The tool offset command in the canned cycle is invalid.
6The canned cycle must be canceled before changing the drilling axis or machining plane.

G3.9.5 G83-deep hole machining

G83 is a deep hole machining cycle instruction, which performs intermittent tapping to the bottom of the hole, and then exits quickly.

Instruction format

G83 IP_ R_ Q_ F_ J_;
IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
Q_:Feed rate per cutting feed
F_:feed rate
J_:Machining shaft
J_:加工轴

G83(G98)	G83(G99)
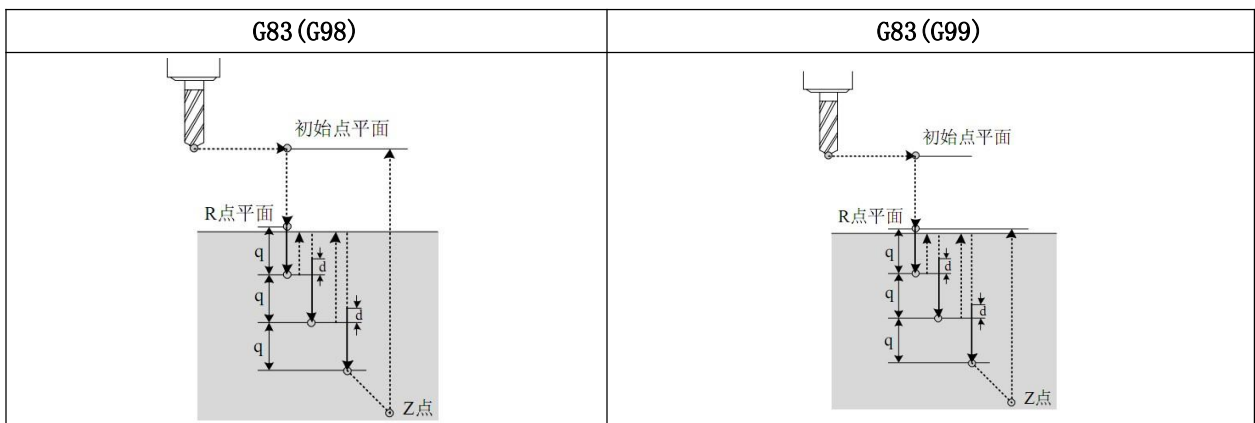
According to the above format instruction, q is the cut in quantity each time, and the increment value instruction is used.When cutting in after the second time, fast feed to the position D mm away from the

position just finished, and then change to cutting feed. Even if the value of Q is negative, the sign must be invalid. D is set with parameters.

notes

---

1 start spindle rotation before g83 is specified. If g83 and M code are specified in the same program segment, the M code is sent out at the initial positioning, and the next loop action is carried out after the M code is executed.

If any one or more of the X, y, Z, R data is instructed in the fixed cycle state, the system will process the hole. However, when x and G04 are specified at the same time, no hole machining is performed.

3. In the program section which can carry out the hole machining action, the hole processing data Q and P can be ordered. In the program section that can not be processed, the data Q and P of the instruction hole processing can not be stored as modal data.

4 in the fixed cycle mode, if the tool length offset has been instructed, the offset will be performed when the initial point plane is positioned.

5 tool offset command is invalid in fixed cycle.

6 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.6 g85 - boring cycle

G85 is used for boring. After machining, G76 can be used for fine boring. The cycle process is the same as g84, but the spindle does not reverse at the bottom of the hole, and there is no pause time.

Instruction format

指令格式

G85 IP_ R_ F_ J_;

  IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)

  R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)

  F_:feed rate

  J_:Machining shaft

Instruction interpretation

After positioning, the tool quickly moves to the R point, cuts to the bottom of the hole along the Z direction, exits at the cutting speed, and then returns to the R point or the initial plane.

notes

---

1 start spindle rotation before g85 is specified. If g85 and M code are specified in the same program segment, the M code will be sent out at the initial positioning and wait for the M code to execute before the next loop action.

If any one or more of the X, y, Z, R data is instructed in the fixed cycle state, the system will process the hole. However, when x and G04 are specified at the same time, no hole machining is performed.

3. In the fixed cycle mode, if the tool length offset has been instructed, the offset will be carried out when the initial point plane is positioned.

4 tool offset command is invalid in fixed cycle.

5 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.7 G86 - boring cycle

G86 is used for boring. After machining, G76 can be used for fine boring. The cycle process is the same

as g81, only the spindle stops at the bottom of the hole.

Instruction format

> **G86 IP_ R_ F_ J_;**
>   IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate
> of hole bottom (absolute value)
>   R_:The distance from the plane of the initial point to the R point (incremental value), or
> the coordinates of the R point (absolute value)
>   F_:feed rate
>   J_:Machining shaft

Instruction interpretation

After positioning, the tool quickly moves to R point and cuts to the bottom of the hole along the Z direction.
The spindle stops rotating. Then the tool returns to R point or initial plane quickly, and the spindle
rotates forward.

notes

> 1 start spindle rotation before g86 is specified. If g86 and M code are specified in the same
> program segment, the M code is sent out at the initial positioning, and the next loop action
> is carried out after the M code is executed.
>   If any one or more of the X, y, Z, R data is instructed in the fixed cycle state, the system
>   will process the hole. However, when x and G04 are specified at the same time, no hole machining
>   is performed.
>   3. In the fixed cycle mode, if the tool length offset has been instructed, the offset will be
>   carried out when the initial point plane is positioned.
> 4 tool offset command is invalid in fixed cycle.
> 5 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.8 G89 - boring cycle

G89 is used for boring. After g89 is finished, the cycle process is the same as g85, but there is a pause

time at the bottom of the hole.

Instruction format

> **G89 IP_ R_ P_ F_ J_;**
>   IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate
> of hole bottom (absolute value)
>   R_:The distance from the plane of the initial point to the R point (incremental value), or
> the coordinates of the R point (absolute value)
>   P_:Pause time at hole bottom (0.001 sec)
>   F_:feed rate
>   J_:Machining shaft

Instruction interpretation

G89 cycle and g85 - sample only add pause time at the bottom of the hole, which can improve the machining
accuracy of blind hole.

notes

1 start spindle rotation before g89 is specified. If g89 and M code are specified in the same program segment, the M code is sent out at the initial positioning, and the next loop action is carried out after the M code is executed.

If any one or more of the X, y, Z, R data is instructed in the fixed cycle state, the system will process the hole. However, when x and G04 are specified at the same time, no hole machining is performed.

3. In the fixed cycle mode, if the tool length offset has been instructed, the offset will be carried out when the initial point plane is positioned.

4 tool offset command is invalid in fixed cycle.

5 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.9 G80 - fixed cycle cancellation

G80 is used to cancel the fixed state.

Instruction format

指令格式

G80 ;

It is used to cancel all processing data of all fixed cycles (G73, G74, g81 ~ g89), and then process according to the normal action.

Chapter 4 auxiliary functions (M code)

If a 2-bit value is given after the address m, the corresponding signal is sent to the machine tool to control the auxiliary function switch of the machine tool. Only one valid M code is allowed in a program segment. When the position moving instruction and M instruction are in the same program segment, both start to execute at the same time.

4.1 overview

The following is a list of the codes used in this system.

name	function
M00	Program pause, press "Cycle Start" program to continue execution
M01	Select stop, if the stop light is selected, the program stops
M02	program stops
M03	Spindle 1 rotates forward
M04	Spindle 1 reverse
M05	Spindle 1 stops
M08	coolant on
M09	Coolant off
M10	clamp
M11	release
M13	Spindle 2 Forward
M14	Spindle 2 back transmission
M15	Spindle 2 stops
M30	Program ends, program returns to start
M64	Count plus one
M65	count clear
M70	Waiting for the input port, output port or auxiliary relay to be invalid Example: M70 X12 input port; M70 Y1 output port; M70 Z1 auxiliary relay;
M71	Wait for the input port, output port or auxiliary relay to be valid Example: M71 X12 input port; M71 Y1 output port; M71 Z1 auxiliary relay;
M72	Input port, output port or auxiliary relay invalid jump
M73	Input port, output port or auxiliary relay valid jump
M74	Wait for the falling edge of input port, output port or auxiliary relay
M75	Wait for the rising edge of input port, output port or auxiliary relay
M76	Absolute jump Example: M76 P1
M80	The output port or auxiliary relay is closed Example: M80 Y12
M81	Output port or auxiliary relay open Example: M81 Y12
M82	The output port or auxiliary relay output is closed for a period of time Example: M82 Y12 P1000 (milliseconds)
M83	The output port or auxiliary relay output will be closed after waiting for an input port to be valid. Example: M83 Y12 X13
M84	The output port or auxiliary relay output is closed after waiting for an input port to be invalid. Example: M84 Y12 X13
M98	Call the subroutine. Note that the subroutine name format is Oxxxx.nc, where x is a number
M99	Subprogram or macro program return. If used in the main program, the

	program loops from the beginning

4.2 M code description

4.2.1 moo program suspension

Instruction format

M00 (or M0);
    Command function
After executing the M00 command, the program stops running, and the word "pause" is displayed. After pressing the cycle start key, the program continues to run.

4.2.2 M01 program selective stop

Instruction format

M01 (or M1);
Command function
When "select Stop" is on, M01 command is valid. When M01 is encountered in the process of program execution, the system stops running after the current program segment is executed, and continues to execute when the cycle is started again.

4.2.3 M02 - end of procedure

Instruction format

M02 (or m2);
Command function
In the automatic mode, the M02 instruction is executed. After other instructions of the current program segment are executed, the automatic operation ends. The cursor stays in the program segment where the M02 instruction is located and does not return to the beginning of the program. To execute the program again, you must return the cursor to the beginning of the program.
    When the counting mode is automatic (p0003 = 0), the counter is increased by one.

4.2.4 M03 - spindle 1 forward rotation

Instruction format

M03 (or m3);
Command function
When the program executes the M03 command, first make the spindle 1 forward rotation relay close, and then control the spindle to rotate clockwise according to the speed specified by s code.

4.2.5 M04 - spindle 1 reversal

Instruction format

M04 (or M4);
Command function
Control spindle 1 to reverse.

4.2.6 M05 - spindle 1 stop

Instruction format

M05 (or M5);
Command function
Turn off the output of M03 or M04 to stop the rotation of spindle 1.

4.2.7 M08 / M09 - coolant on / off

Instruction format

M08 (or M8);

M9 or M9;

Command function

The M08 command opens the coolant.

The M09 command turns the coolant off.

4.2.8 M10 / M11 - clamping / loosening

Instruction format

M10;

M11;

Command function

The instruction of M10 is clamping.

M11 command is release.

4.2.9 M13 spindle 2 forward rotation

Instruction format

M13

Command function

When the program executes the M13 command, it first makes the spindle 2 forward rotation relay close, and then controls the spindle to rotate clockwise according to the speed specified by SS code.

4.2.10 M14 - spindle 2 reversal

Instruction format

M14

Command function

Control spindle 2 to reverse.

4.2.11 M15 - spindle 2 stop

Instruction format

M15

Command function

Turn off the output of M13 or M14 to stop the rotation of spindle 2.

4.2.12 M30 - program stop

Instruction format

M30

Command function

In the automatic mode, M30 instruction is executed. After other instructions of the current program segment are executed, the automatic operation ends and the cursor returns to the beginning of the program. To execute the program again.

When the counting mode is automatic (p0003 = 0), the counter is increased by one.

4.2.13 M64 counter plus one

Instruction format

M64

Command function

Add one to the workpiece count.

4.2.14 M65 - counter clear

Instruction format

    M65

Command function

The workpiece count value is cleared.

4.2.15 M70 - wait for input port, output port and auxiliary relay to be invalid

Instruction format

M70 Xxx; Input port

    M70 Yxx; Output port

    M70 Zxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

When the specified port is valid, the program waits.

When the specified port is invalid, the program goes down.

Only one port can be specified at a time.

4.2.16 M71 - wait for input port, output port and auxiliary relay to work

Instruction format

M71 Xxx; Input port

    M71 Yxx; Output port

    M71 Zxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

When the specified port is invalid, the program waits.

When the specified port is valid, the program goes down.

Only one port can be specified at a time.

4.2.17 M72 - invalid jump of input port, output port and auxiliary relay

Instruction format

M72 Xxx Pn; Input port

    M72 Yxx Pn; Output port

    M72 Zxx Pn; Auxiliary relay

Command function

XX: No. 01 ~ 96.

When the specified port is invalid, the program jumps to the N number specified by P. Effective downward execution.

Only one port can be specified at a time.

4.2.18 M73 - input port, output port, auxiliary relay effective jump

Instruction format

M73 Xxx Pn; Input port

    M73 Yxx Pn; Output port

    M73 Zxx Pn; Auxiliary relay

Command function

XX: No. 01 ~ 96.

When the specified port is valid, the program jumps to the N number specified by P.Invalid execution down.

Only one port can be specified at a time.

4, port M74, wait for down

Instruction format

M70 Xxx; Input port

　　M70 Yxx; Output port

　　M70 Zxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

　　The specified port waits for a valid signal and then an invalid signal.Only one port can be specified at a time.

4.2.20 M75 - wait for input port, output port and auxiliary relay to rise

Instruction format

M71 Xxx; Input port

　　M71 Yxx; Output port

　　M71 Zxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

　　The specified port waits for an invalid signal and then a valid signal.

Only one port can be specified at a time.

4.2.21 M80 output port, auxiliary relay off

Instruction format

M80 Yxx; Output port

　　M80 Zxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

　　Turn off an output port or auxiliary relay.

　　Only one port can be specified at a time.

4.2.22 M81 output port, auxiliary relay on

Instruction format

M81 Yxx; Output port

　　M81 Zxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

　　Open an output port or auxiliary relay.

　　Only one port can be specified at a time.

4.2.23 M82 - output port, auxiliary relay output is closed for a period of time

Instruction format

M82 Yxx Paaaa; Output port

　　M82 Zxx Paaaa; Auxiliary relay

Command function

XX: No. 01 ~ 96.

PAAA: delay time, in milliseconds.

Only one port can be specified at a time.

4.2.24 M83 - output port, auxiliary relay output will be closed after one input port is valid

Instruction format

M83 Yxx Xxx; Output port

    M83 Zxx Xxx; Auxiliary relay

Command function

XX: No. 01 ~ 96.

For example: M83 Y12 x13;Explanation: turn on output port 12 and judge whether input port 13 is valid. If it is, turn off output 12.

Only one port can be specified at a time.

4.2.25 M84 - output port, auxiliary relay output will be closed after one input port is invalid

Instruction format

M84 Yxx Xxx; Output port

    M84 Zxx Xxx; Auxiliary relay
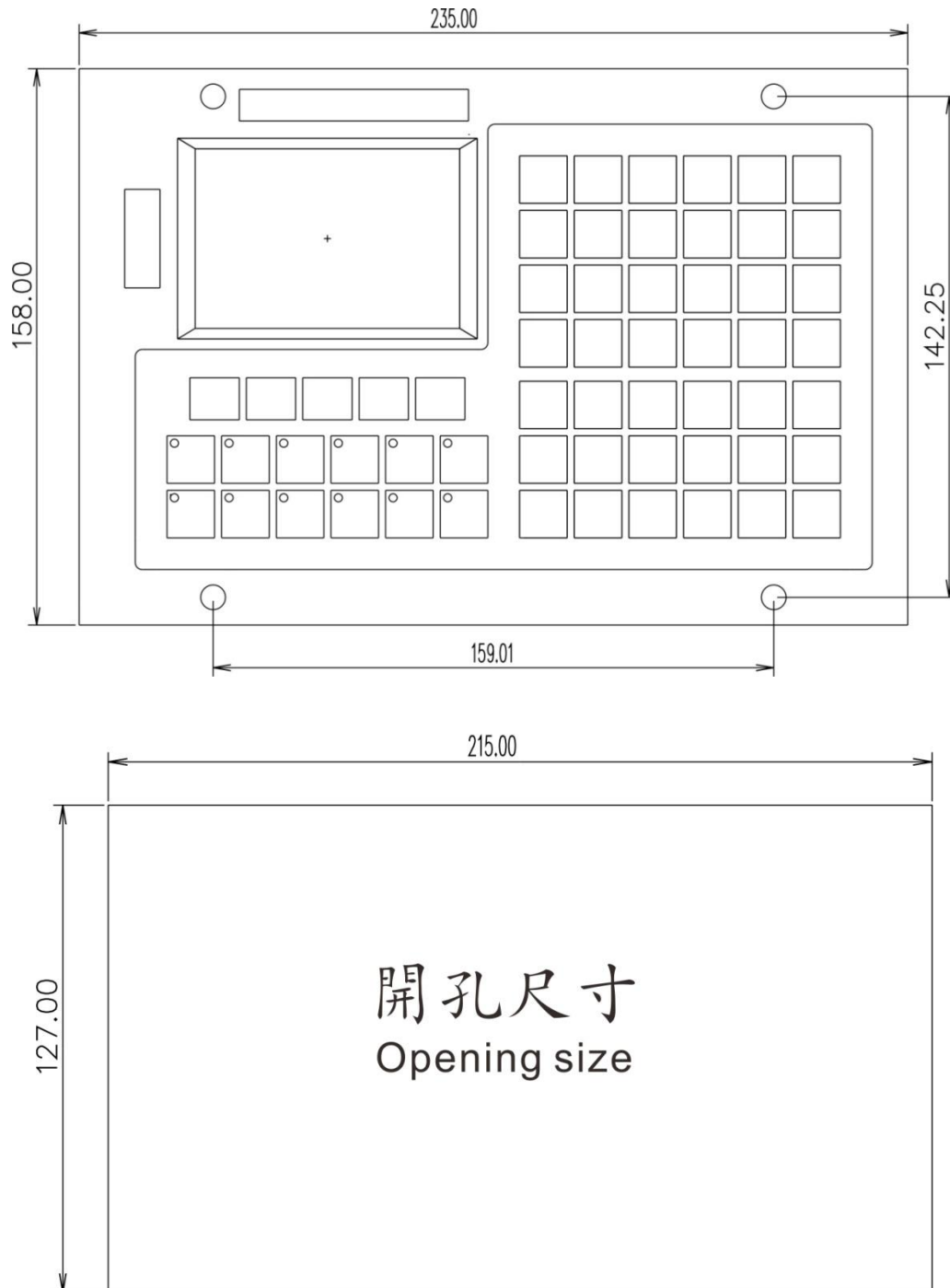
Command function

XX: No. 01 ~ 96.

For example: M84 Y12 x13;Explanation: turn on output port 12, then judge whether input port 13 is invalid, if not, turn off output 12.

Only one port can be specified at a time.

PART 4 INSTALLATION AND COMMISSIONING
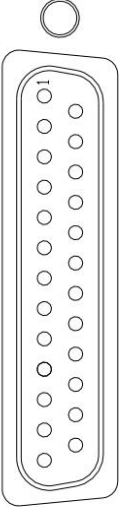
Chapter 1 installation layout

1. 1 Dimensions

Note: The above dimensions are in mm.

Chapter 2 interface signal definition and connection

2.1 driver interface

2.1.1 driver interface definition

Interface form	Pin	Signal name	Function description
	1	XCP+	X-axis command pulse signal+
	2	XCP-	X-axis command pulse signal-
	3	XDIR+	X-axis command direction signal+
	4	XDIR-	X-axis command direction signal-
	5	YCP+	Y-axis command pulse signal+
	6	YCP-	Y-axis command pulse signal-
	7	YDIR+	Y-axis command direction signal+
	8	YDIR-	Y-axis command direction signal-
	9	ZCP+	Z-axis command pulse signal+
	10	ZCP-	Z-axis command pulse signal-
	11	ZDIR+	Z-axis command direction signal+
	12	ZDIR-	Z-axis command direction signal-
	13	ACP+	A-axis command pulse signal+
	14	ACP-	A-axis command pulse signal-
	15	ADIR+	A-axis command direction signal+
DB type 25 hole socket (driver)	16	ADIR-	A-axis command direction signal-
	25	5V	Spare 5V

2.1.2 principle of command pulse signal and command direction signal

CP + and CP - are command pulse signals, and dir + and dir - are command direction signals. These two groups of signals are differential output. Am26ls31 driver chip is used inside, which meets RS422 level standard. The internal circuit is shown in the following figure:
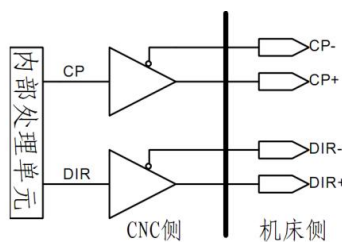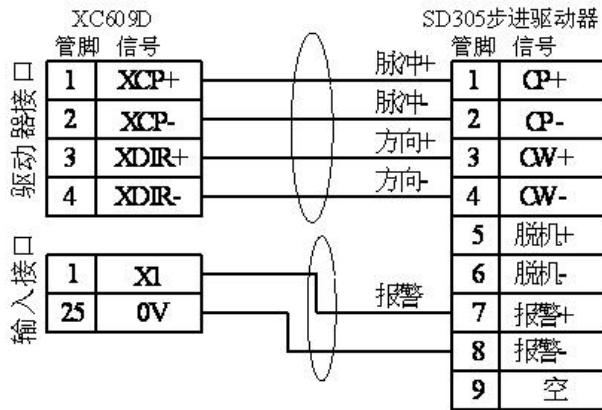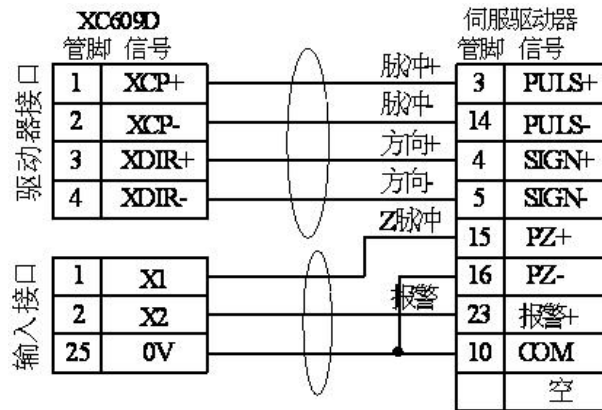

Figure 2.1.2 Internal circuit of command pulse signal and command direction signal

2.1.3 connection diagram with driving unit
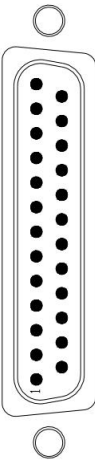
2.1.3.1 Connection with stepper driver

### 2.1.3.2 Connect with Servo Drive

2.2 Spindle and other interfaces

2.2.1 definition of spindle and other interfaces

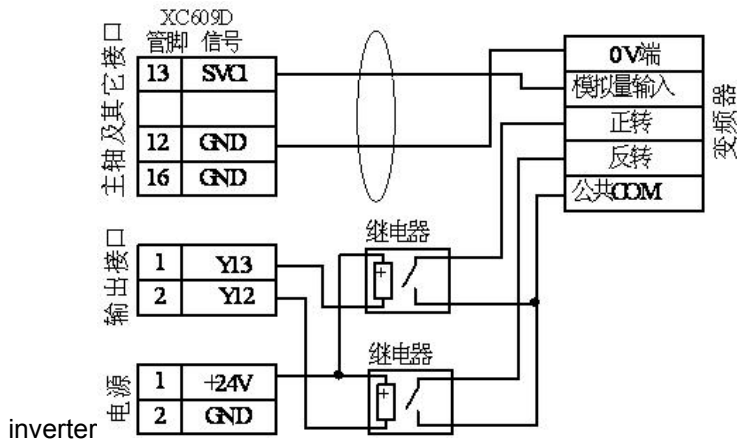| Interface form	Pin	Signal name	Function description
	1	X13	Input 13 (optional function, refer to the optional function of entry)
	8	GND	power ground
	9	SLB	Handwheel B-phase pulse input
	10	SLA	Handwheel A-phase pulse input
	11	5V	+5V power output
	12	GND	power ground
	13	SVC1	Analog voltage 1. 0~10V analog signal output, connect to the analog voltage input terminal of the inverter
	14	485A	RS485 communication interface
	15	485B	RS485 communication interface
	16	GND	power ground
(spindle and others)（主轴及其它）	17	X22	Input 22 (optional function, refer to the optional function of the entrance)
	18	X21	Input 21 (optional function, refer to the optional function of the entrance)
	19	X20	Input 20 (optional function, refer to the optional function of the entrance)

20	X19	Input 19 (optional function, refer to the optional function of the entrance)
21	X18	Input 18 (optional function, refer to the optional function of the entrance)
22	X17	Input 17 (optional function, refer to the optional function of the entrance)
23	X16	Input 16 (optional function, refer to the optional function of the entrance)
24	X15	Input 15 (optional function, refer to the optional function of the entrance)

2.2.2 interface principle of analog spindle

The analog spindle interface (SVC) can output 0 ~ 10V analog voltage signal.The internal circuit of the signal is shown in the figure below:
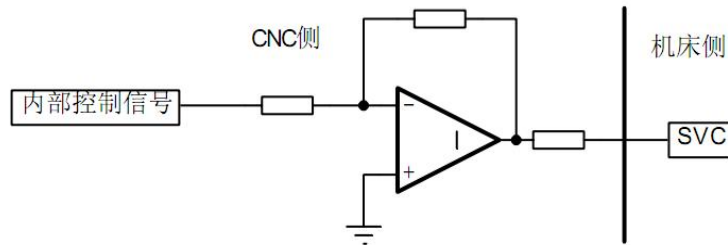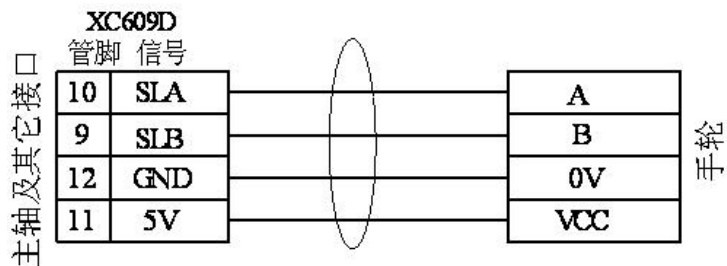

2.2.3 Description of the connection between the simulated spindle and the

inverter

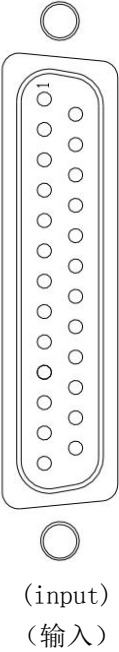2.2.4 Handwheel connection instructions

XC609D handwheel adopts single end input mode.Hand wheel connection is shown in the following figure:

2.3 Input interface

2.3.1 definition of input interface

Interface form	Pin	Signal name	Function description
	1	X01	Input 01 (optional function, refer to the optional function of entry)
	2	X02	Input 02 (optional function, refer to the optional function of entry)
	3	X03	Input 03 (optional function, refer to the optional function of entry)
	4	X04	Input 04 (optional function, refer to the optional function of entry)
	5	X05	Input 05 (optional function, refer to the optional function of entry)
	6	X06	Input 06 (optional function, refer to the optional function of entry)
	7	X07	Input 07 (optional function, refer to the optional function of entry)
	8	X08	Input 08 (optional function, refer to the optional function of entry)
	9	X09	Input 09 (optional function, refer to the optional function of entry)
	10	X10	Input 10 (optional function, refer to the optional function of entry)
	11	X11	Input 11 (optional function, refer to the optional function of entry)
	12	X12	Input 12 (optional function, refer to the optional function of entry)
	13	X13	Input 13 (optional function, refer to the optional function of entry)
(input) (输入)	14	X14	Input 14 (optional function, refer to the optional function of entry)
	15	X15	Input 15 (optional function, refer to the optional function of entry)
	16	X16	Input 16 (optional function, refer to the optional function of entry)
	17	X17	Input 17 (optional function, refer to the optional function of entry)
	18	X18	Input 18 (optional function, refer to the optional function of entry)
	19	X19	Input 19 (optional function, refer to the optional function of entry)
	20	X20	Input 20 (optional function, refer to the optional function of entry)
	21	X21	Input 21 (optional function, refer to the optional function of entry)

22	X22	Input 22 (optional function, refer to the optional function of entry)
23	X23	Input 23 (optional function, refer to the optional function of entry)
24	X24	Input 24 (optional function, refer to the optional function of entry)
25	GND	power ground

2.3.2 optional functions of input port

| Serial number	function	level	Serial number	function	level	Serial number	function	level
1	Universal	Normally open, normally closed	19	Handwheel Y	Normally open, normally closed	37	Z zero	Normally open, normally closed
2	emergency stop	Normally open, normally closed	20	Handwheel Z	Normally open, normally closed	38	Z alarm	Normally open, normally closed
3	reset	Normally open, normally closed	21	Handwheel A	Normally open, normally closed	39	A positive limit	Normally open, normally closed
4	main boot	Normally open, normally closed	22	Handwheel X1	Normally open, normally closed	40	A negative limit	Normally open, normally closed
5	pause	Normally open, normally closed	23	Handwheel X10	Normally open, normally closed	41	A zero	Normally open, normally closed
6	edit	Normally open, normally closed	24	Handwheel X100	Normally open, normally closed	42	A alarm	Normally open, normally closed
7	Teach	Normally open, normally closed	25	Knife setter protection	Normally open, normally closed	43	Tool setter input	Normally open, normally closed
8	automatic	Normally open, normally closed	26	Spindle alarm	Normally open, normally closed	44	X-axis Z-CP	Normally open, normally closed

| 9	MDI	Normally open, normally closed	27	X positive limit	Normally open, normally closed	45	Y-axis Z-CP	Normally open, normally closed
10	return to zero	Normally open, normally closed	28	X negative limit	Normally open, normally closed	46	Z-axis Z-CP	Normally open, normally closed
11	single step	Normally open, normally closed	29	X zero	Normally open, normally closed	47	A-axis Z-CP	Normally open, normally closed
12	manual	Normally open, normally closed	30	X alarm	Normally open, normally closed	48		
13	cool down	Normally open, normally closed	31	Y positive limit	Normally open, normally closed	49		
14	Spindle rotates forward	Normally open, normally closed	32	Y negative limit	Normally open, normally closed	50		
15	Spindle reverse	Normally open, normally closed	33	Y zero	Normally open, normally closed	51		
16	Spindle stopped	Normally open, normally closed	34	Y alarm	Normally open, normally closed	52		
17	Clamp release	Normally open, normally closed	35	Z positive limit	Normally open, normally closed	53		
18	HandwheelX	Normally open, normally closed	36	Z negative limit	Normally open, normally closed	54		

2.3.3 The principle of the input port circuit

Input signal refers to the signal input to CNC from machine tool or equipment.When the input signal is connected

with GND, the input is valid;When the input signal is disconnected from GND, the input is invalid.

There are two ways of external input of input signal: one is to use contact switch input. The signal in this way comes from the key, limit switch and relay contact on the machine side. The connection mode is shown in the following figure with contact type input:
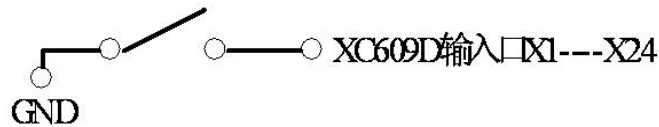
The other uses the non-contact switch (transistor) input, and the connection is as shown below for the non-contact switch (NPN transistor) input:
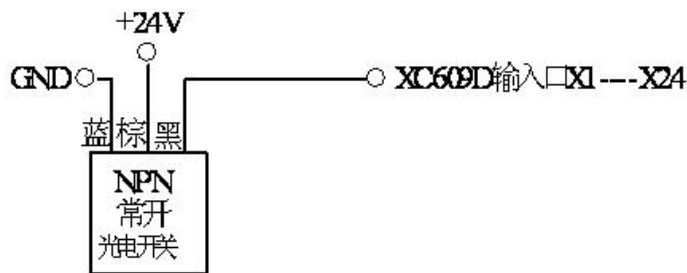
2.3.4 The principle and wiring of the tool setter
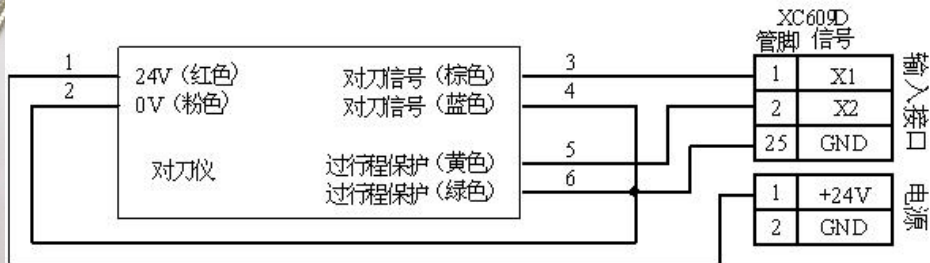
The tool setting signal (brown) of the tool setting instrument must be connected to the input X1---X8 of the XC609D system, and the over-travel protection (yellow) signal can be connected to the input X1---X24.

2.4 output interface

2.4.1 definition of output interface

Interface form	Pin	Signal name	Function description
	1	Y13	Output 13 (optional function, please refer to the table below for setting)
	2	Y12	Output 12 (optional function, please refer to the table below for setting)
	3	Y11	Output 11 (optional function, please refer to the table below for setting)
	4	Y10	Output 10 (optional function, please refer to the table below for

| DB type 25-pin socket (output)			setting)
	5	Y09	Output 09 (optional function, please refer to the table below for setting)
	6	Y08	Output 08 (optional function, please refer to the table below for setting)
	7	Y07	Output 07 (optional function, please refer to the table below for setting)
	8	Y06	Output 06 (optional function, please refer to the table below for setting)
	9	Y05	Output 05 (optional function, please refer to the table below for setting)
	10	Y04	Output 04 (optional function, please refer to the table below for setting)
	11	Y03	Output 03 (optional function, please refer to the table below for setting)
	12	Y02	Output 02 (optional function, please refer to the table below for setting)
	13	Y01	Output 01 (optional function, please refer to the table below for setting)
	14	+24V	Power 24V
	15	Y24	Output 24 (optional function, please refer to the table below for setting)
	16	Y23	Output 23 (optional function, please refer to the table below for setting)
	17	Y22	Output 22 (optional function, please refer to the table below for setting)
	18	Y21	Output 21 (optional function, please refer to the table below for setting)
	19	Y20	Output 20 (optional function, please refer to the table below for setting)
	20	Y19	Output 19 (optional function,

		please refer to the table below for setting)
21	Y18	Output 18 (optional function, please refer to the table below for setting)
22	Y17	Output 17 (optional function, please refer to the table below for setting)
23	Y16	Output 16 (optional function, please refer to the table below for setting)
24	Y15	Output 15 (optional function, please refer to the table below for setting)
25	Y14	Output 14 (optional function, please refer to the table below for setting)

2.4.2 optional function of output port

Serial number	function	level	Serial number	function	level
1	Universal	nothing	7	blow on the knife	nothing
2	Spindle 1 rotates forward	nothing	8	green light	nothing
3	Spindle 1 reverse	nothing	9	yellow light	nothing
4	cool down	nothing	10	red light	nothing
5	clamp	nothing	11		
6	release	nothing	12		

2.4.3 The principle of output port circuit

The output signal is used to drive the relay and indicator light on the machine side. When the output signal output is 0V, the output function is effective;Otherwise, the output function will be invalid.There are 48 digital output channels in the I / O interface, all of which have the same structure, as shown in figure 2.5.6
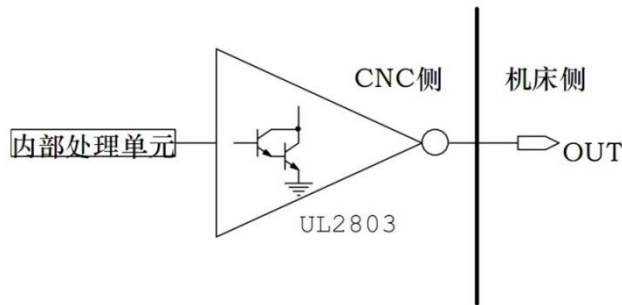

Digital output module circuit structure diagram

2.4.4 output circuit principle

The logic signal y1-y24 output by CNC main control module is sent to the input end of the output interface chip (ULN2803) through the connector. Each output of ULN2803 has Darlington structure, and the output port has two output states: 0V output or high resistance;Each output can withstand 200 Ma tube current.

Typical applications are as follows:

Using ULN2803 output to drive inductive load, it is necessary to connect freewheeling diode at both ends of relay coil to protect output circuit and reduce interference.
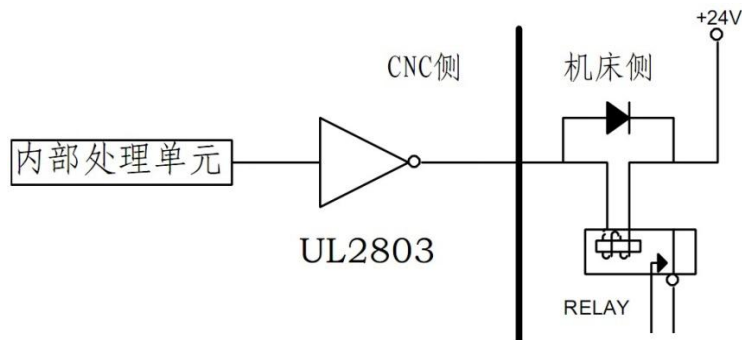
Chapter 3 IO port expansion

3.1 IO port expansion board

Xc609m multi-function and multi-purpose CNC system has 24 input and 24 output. The system can be expanded

to 96 input and output channels. The appearance of the expansion board is shown in the figure below
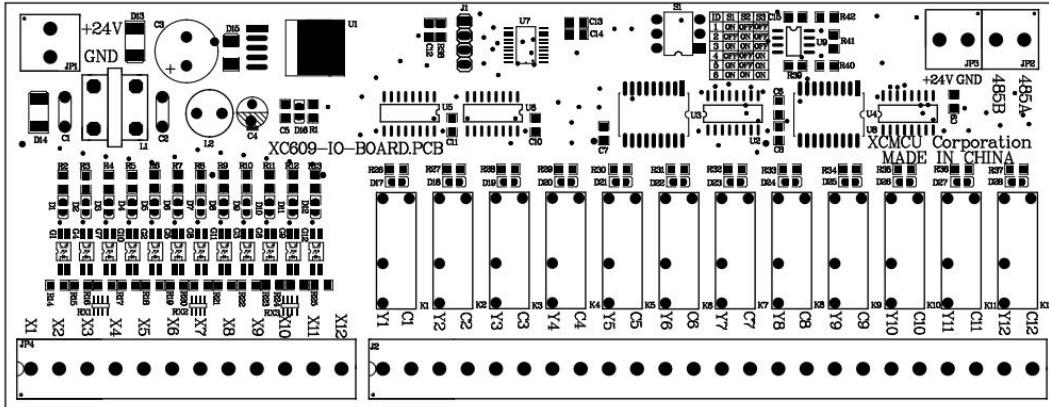
3.2 Use and connection of IO port expansion board

The IO port expansion board of this system adopts 485 bus structure. An IO expansion board has 12 inputs

and 12 outputs. The 485 bus of xc609m system can carry 6 IO expansion boards at most. Each IO expansion

board is defined by dial switch.

The code of dial switch is shown in the table below.

ID (address number)

ID(地址号)	S1	S2	S3
1	ON	OFF	OFF
2	OFF	ON	OFF
3	ON	ON	OFF
4	OFF	OFF	ON
5	ON	OFF	ON
6	ON	ON	ON

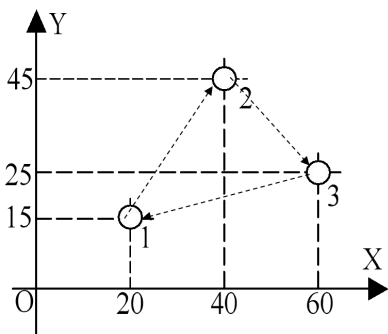The wiring diagram is as follows:
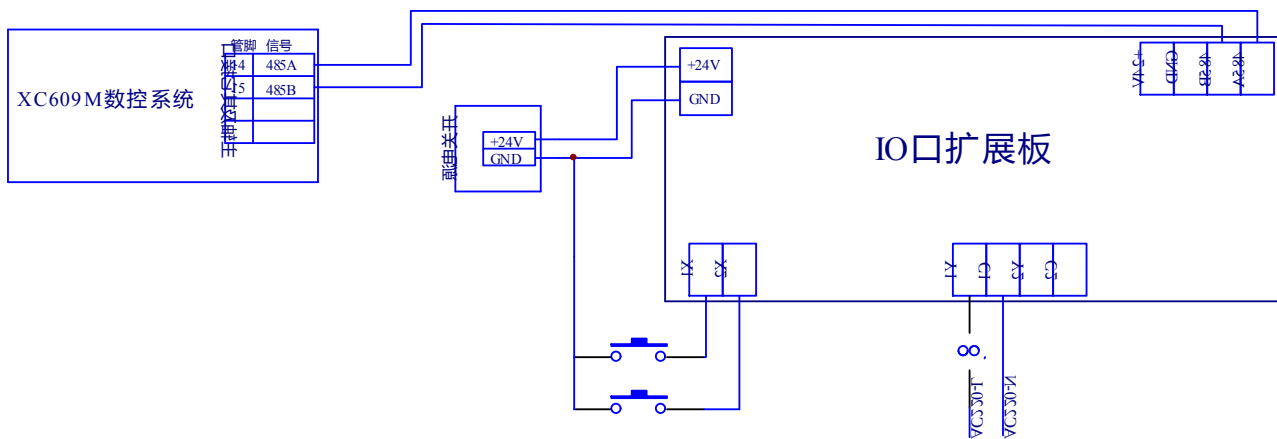
PART 5 REFERENCES

## 1.1 instructions on coordinates and coordinate systems

1.1.1 absolute programming G90 and incremental programming G91

Example: when the tool moves from the origin to 1, 2 and 3 points in sequence, use G90 and G91 instructions to program.


```
%0001
N1 G92 X0 Y0
N2 G90G01X20 Y15
N3 X40 Y45
N4 X60 Y25
N5 X0 Y0
N6 M30
```

```
%0002
N1G91G01X20 Y15
N2 X20 Y30
N3 X20 Y-20
N4 X-60 Y-25
N5 M30
```

1.1.2 Workpiece coordinate system setting G92
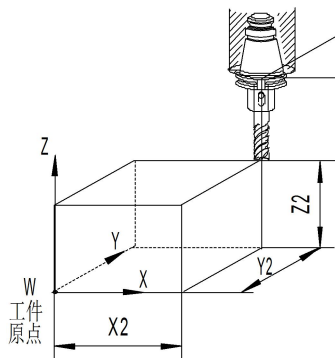
G92 设置加工坐标系

G92 X X2 Y Y2 Z Z2
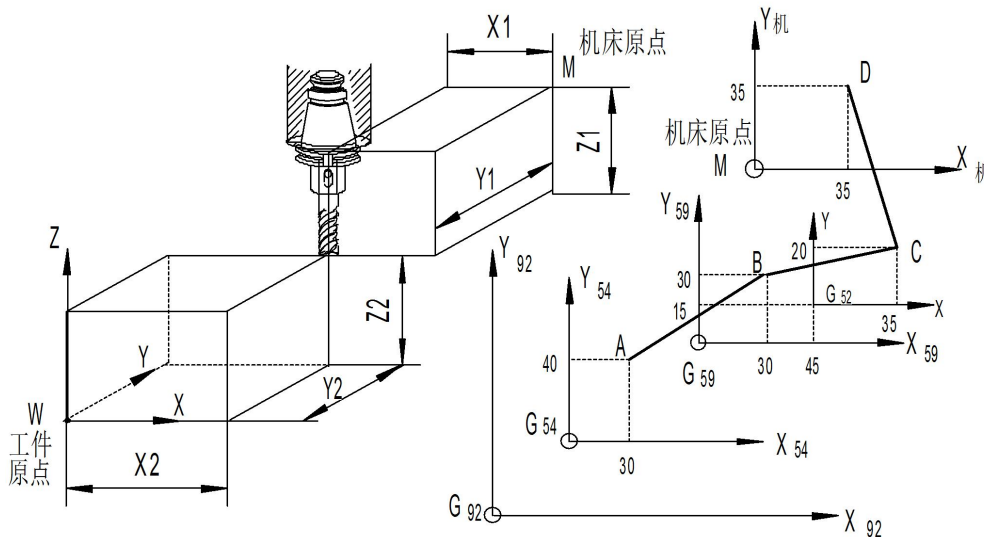
Then set the workpiece origin to the position where the distance from the tool start point is X=-X2, Y=-Y2, Z=-Z2.

1.1.3 workpiece coordinate system selection G54-G59
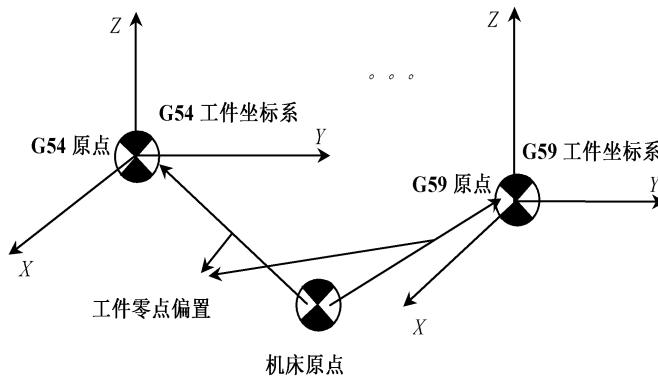
工件坐标系选择(G54~G59)

Example: Walk the route from A-B-C-D as

shown.

N01 g54 G00 G90 x30.0 y40.0   moves quickly to point a in g54

N02 G59 sets G59   as the current workpiece coordinate system

N03 G00 x30.0 y30.0   moves to point B in G59

N04 G52 x45.0 y15.0   establishes local coordinate system G52 in current workpiece coordinate system G59

N05 G00 G90 x35.0 y20.0   moves to point C in G52

N06 g53 x35.0 y35.0   is moved to point D in g53 (mechanical coordinate system)

N06 G53 X35.0 Y35.0   Move to point D in G53 (machine coordinate system)

......

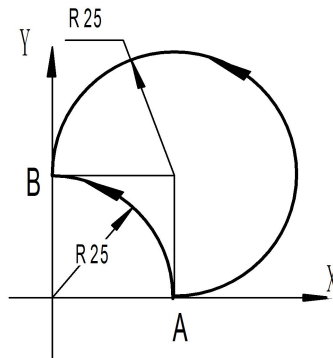## 1.2 Understanding of the coordinate plane

The coordinate plane is G17, G18, G19.

Example 1:
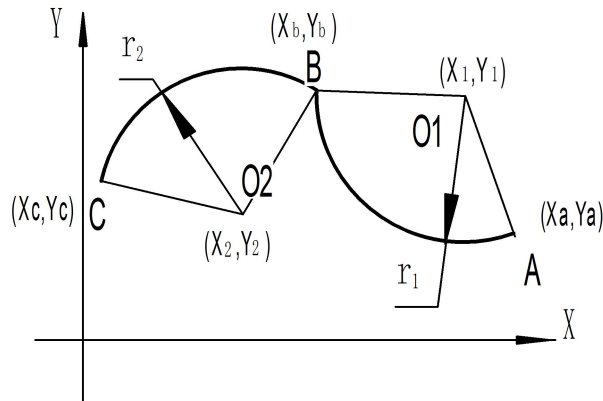
Arc AB: Absolute: G17G90 G02 X xb Y yb R r1 F f;
 or G17G90 G02 X xb Y yb I(x1-xa) J (y1-ya) F f ;
  Increment: G91G02 X (xb-xa)Y (yb-ya) R r1 F f ;
G91G02 X(xb-xa)Y(yb-ya)I(x1-xa)J(y1-ya)F f ;

Example 2: programming arc program segment
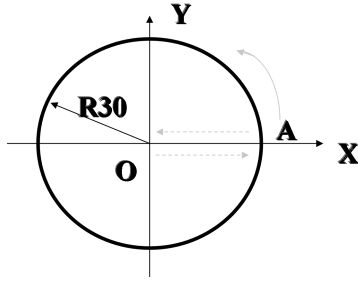

Great arc AB
Each arc can be represented by four blocks
G17 G90 G03 X0 Y25 R-25 F80
G17 G90 G03 X0 Y25 I0 J25 F80
G91 G03 X-25 Y25 R-25 F80 G91
G03 X-25 Y25 I0 J25 F80
Small arc AB
G17 G90 G03 X0 Y25 R25 F80
G17 G90 G03 X0 Y25 I-25 J0 F80
G91 G03 X-25 Y25 R25 F80 G91
G03 X-25 Y25 I-25 J0 F80

Example 3: Full circle programming

It is required to start from point A, realize counterclockwise circular interpolation and return to point A. G90 G03 X30 Y0 I-40 J0 F80 G91 G03 X0 Y0 I-40 J0 F80

Example 4: G02/G03 realizes space helical feed

That is, a third axis movement command perpendicular to the machining plane is added to the back of the original G02 and G03 command format blocks, so that the circular arc feed is also performed in the third axis direction, and the composite trajectory is a Space spiral. X, Y, Z are the end points of the projected arc, and the third coordinate is the end point of the axis perpendicular to the selected plane..
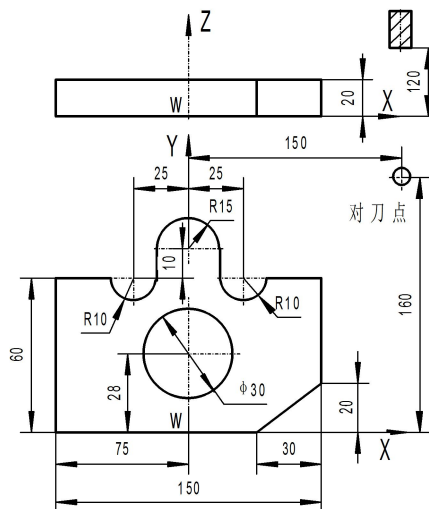

（d）
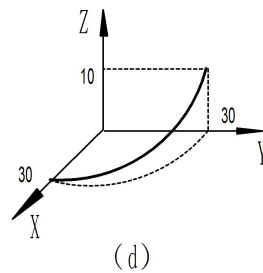
Track as shown above:

G91 G17 G03 X-30.0 Y30.0 R30.0 Z10.0 F100 or: G90 G17 G03 X0 Y30.0 R30.0 Z10.0 F100

1.4 Basic instruction programming example

For the parts shown in the figure, the outer contour is fine-milled with a hole of φ30, regardless of tool compensation.

```
G92 X150.0 Y160.0 Z120.0   Establish workpiece coordinate system and program zero point W
G90 G00 x100.0 y60.0       fast forward to x = 100, y = 60
Z-2.0 S100 M03             z-axis moves to Z = - 2, spindle
G01 x75.0 F100             linear interpolation to x = 75, y = 60
X35.0                     linear interpolation to x = 35, y = 60
G02 x15.0 r10.0           is interpolated to x = 15, y = 60
G01 y70.0                 linear interpolation to x = 15, y = 70
G03 x-15.0 r15.0          inverse circle interpolation to x = - 15, y = 70
G01 y60.0                 linear interpolation to x = - 15, y = 60
G02 x-35.0 r10.0          linear interpolation to x = - 35, y = 60
G01 x-75.0                linear interpolation to x = - 75, y = 60
G09 Y0                    linear interpolation to x = - 75, y = 0
X45.0                     linear interpolation to x = 45, y = 45
X75.0 y20.0               linear interpolation to x = 75, y = 20
Y65.0                     linear interpolation to x = 75, y = 65, contour complete
G00 x100.0 y60.0          quickly retreats to the position of x = 100, y = 60
Z120.0                    The tool setting point is 120 to 120
X150.0y160.0              fast tool withdrawal to tool setting point
M05 M30                   program ends, reset.
```