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 OVERVIE

1.1 system introduction

XC609M multi-function and multi-purpose CNC system is a new generation CNC system developed by our company. Support milling, boring, tapping, drilling and automatic feeding. 001mm, high-speed, high-speed and high-performance interpolation technology. It is the best choice for CNC drilling machine, CNC milling machine, special machine tool, automation equipment, automatic welding robot, feeding robot, coordinate robot, etc.

Software and hardware characteristics of XC609M CNC system:

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

Three programs (one main program and two auxiliary programs) can be run at the same time, which is convenient for processing, loading and unloading.

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 and tapping cycle.

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 and data backup 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 acceleration and deceleration control before interpolation.

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

Support G code file of third party software.

1.2 technical specifications

basic function	
Number of control axes	Axis 1 ~ 6 (x, y, Z, a, B, c)
Number of linkage axes	Full linkage
Simulation spindle	2
Spindle monitoring	yes
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	0～150%
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
Trip switch	yes
Single section operation	yes
Program protection switch	yes
Self diagnosis function	yes
Emergency stop	yes
Power Supply	DC24V
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;Control the start and stop of lubrication
Spindle function	
Spindle function	Double spindle
tapping	support
Spindle analog output	Yes, double spindle
Tool function	
Tool function	It supports the tool setting in the middle and three-point centering
Tool compensation memory	-9999.999 ~ 9999.999, 99
Tool compensation	Length compensation of each axis
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

PART 2 OPERATION INSTRUCTIONS

Chapter I description of operation authority

1.1 permission level

XC609M uses a classified and hierarchical authority structure, and all kinds of permissions are aimed at 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	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"

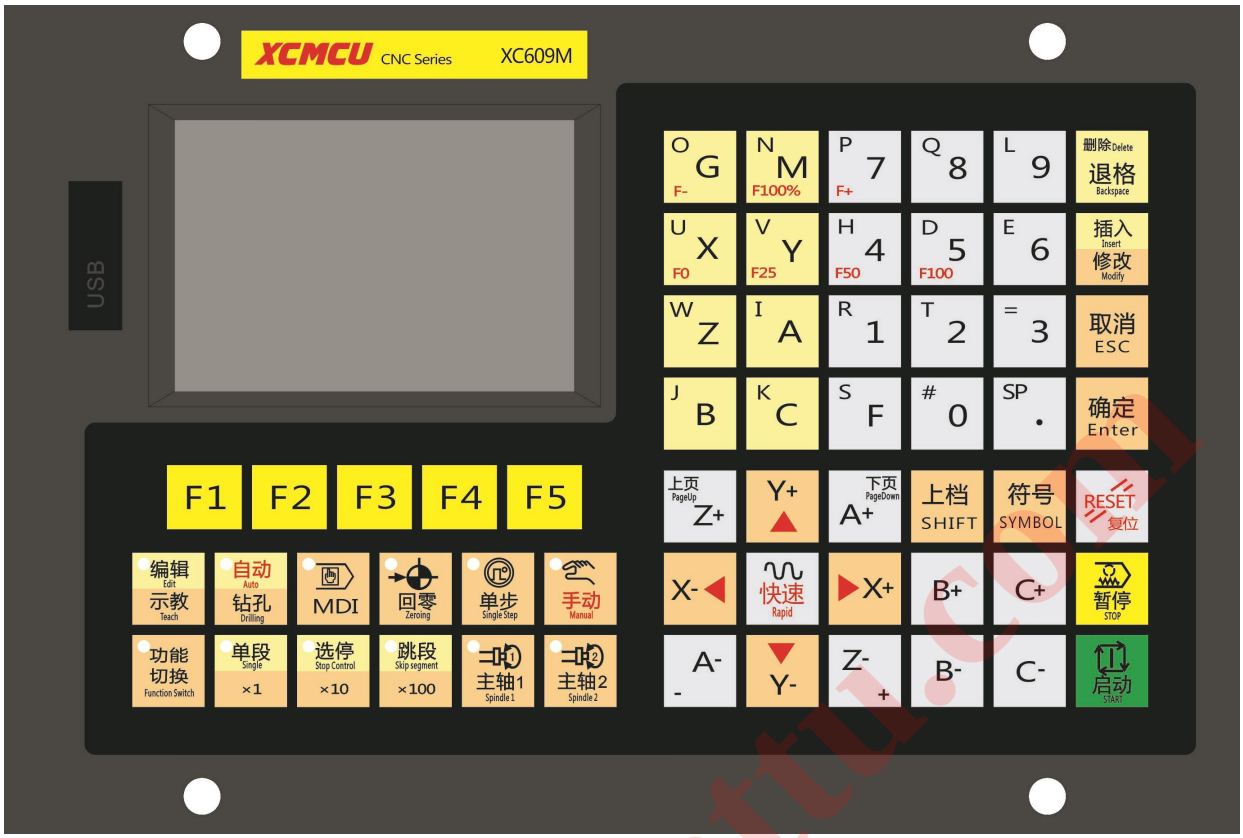
1.2 operation authority

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
	Knife compensation modification	Grade F	
	Backup and recovery of cutter	C	Edit mode or input mode, stop status
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

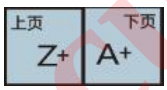
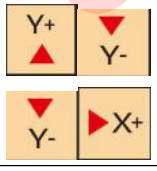
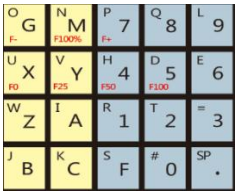
2.1.1 description

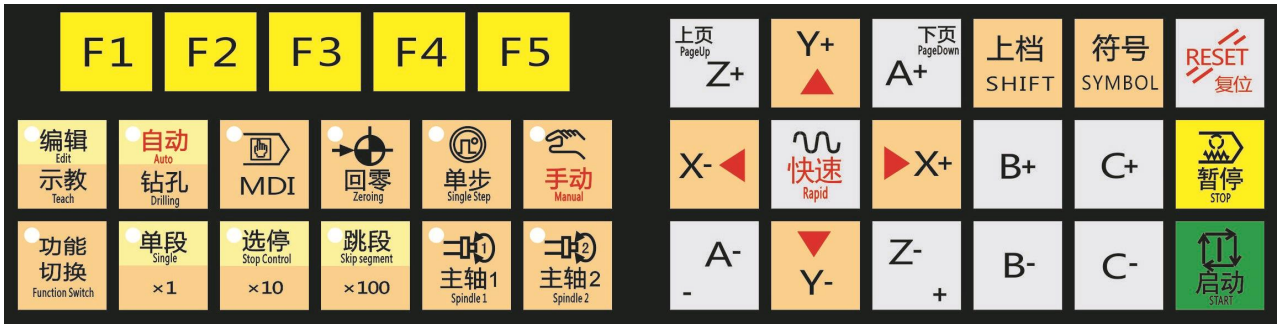
The upper part is the character number editing key.

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

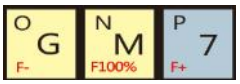
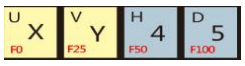
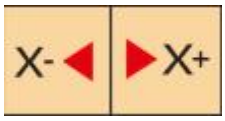
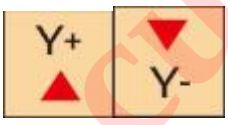
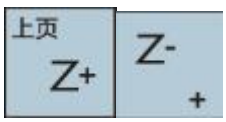
Key Icon	Key name	Functional use
	Reset key	CNC reset, program end processing, release alarm, terminate input and output.
	Backspace / delete key	Delete the character after the cursor position, delete key to delete the character in front of the cursor. (backspace and delete share a common entity key, which can be selected with the up file key)
	Insert / modify key	Modify edit mode, switch between insert and modify mode, or modify data by parameter
	escape key	Cancel the input and close the pop-up dialog box to return to the program content screen.
	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.
	shift	Select or cancel the upshift function.
	Symbol key	In editing mode, input special characters, such as +, -, */
	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)
	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)
	Character key	Character input. Double function key, you need to press the up shift key to input the second function character. (can be defined as port direct control key, which works in manual mode)

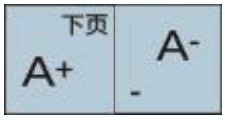
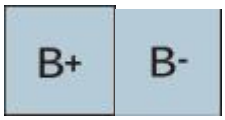
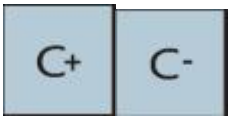


2.1.3 function operation key of machine tool

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 钻孔	Auto Key	Enter the automatic operation mode. Press again and switch the drilling function, and the indicator light flashes for drilling function
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 ×1	Single segment bond	Switch between single section operation and continuous operation during automatic operation, when single section operation is effective, indicator light is on, and when single handwheel is single, the magnification is x1.
选停 ×10	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. When single handwheel is used, the ratio is X10.
跳段 ×100	Skip key	In the automatic and input mode, whether the program section with the "/" number will skip the state switching, and the skip section indicator light will be on when the program section trip switch is on. When single handwheel is used, the

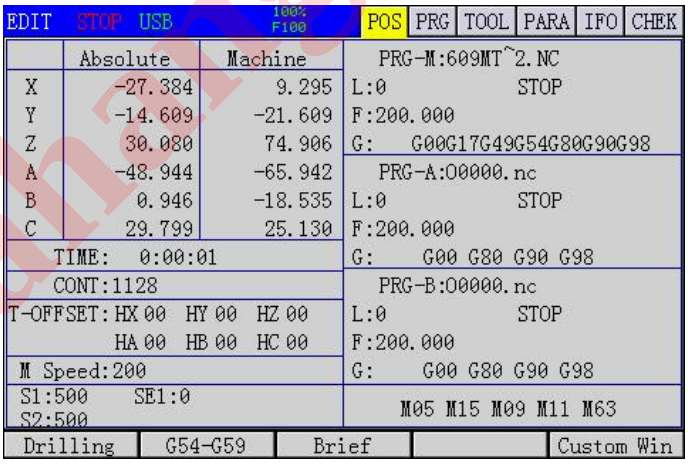
		magnification is X100.
	Feed rate key	Manual rate or feed rate key. In the automatic mode, press the key once to increase or decrease the feed rate by 10% or 100%. 10%--100%。 Works in automatic mode.
	Incremental selection or fast rate selection	Fast rate f0-100%, working in automatic mode
	Spindle 1 key	Manual, handwheel, return to zero effective, switch between spindle 1 forward transmission and stop
	Spindle 2 key	Manual, handwheel, return to zero effective, switch between forward transmission and stop of spindle 2
	Quick start key	Manual speed, speed switch. When the quick switch has effect, the indicator light will be on.
	Start key	Cycle start key. Automatic or MDI program running start.
	Pause key	Feed hold key. Automatic or MDI program running is suspended.
	X-axis movement or X-axis handwheel axis selection key	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.
	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.
	B-axis movement or b-axis handwheel axis selection key	In the zero return, manual and single step mode, axis B moves in positive or negative direction. In the single handwheel mode, select the handwheel shaft of b axis.
	Axis C or axis selection handwheel	In the zero return, manual and single step mode, the c-axis moves in positive or negative direction. In single handwheel mode, select the handwheel shaft of C axis.

Tool magazine returning to zero, tool changing hand, clockwise tool selection, counter clockwise tool selection, tool clamping / loosening, tool magazine backward, tool magazine advancing, inching, C / s, quasi stop key can be redefined as port output control key, and the original function will disappear after redefinition.

2.2 page display

2.2.1 page layout



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 selection display	Display magnification, display axis selection and magnification in handwheel mode
(3) How it works	Editor: program editor Teaching: edit program with teaching function

	Automatic: automatic operation (program running) Drilling: running a special drilling program 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 six display pages, which are [position], [program], [bias], [system], [information] and [diagnosis], which can be switched by pressing the function switch key. If the up shift key is enabled, the reverse switch will be performed.

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 supplement number of each axis cutter 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	Tool position selection in each coordinate system 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
bias	● tool offset	● set the length in each axis direction
parameter	System parameters Logic parameters ● advanced operation	Parameter setting Logic parameter setting
information	● CNC alarm in progress System information	Check and clear the alarm ● authority setting System lock setting Parameter switch and program switch
diagnosis	● CNC related diagnostic information	Search by serial number

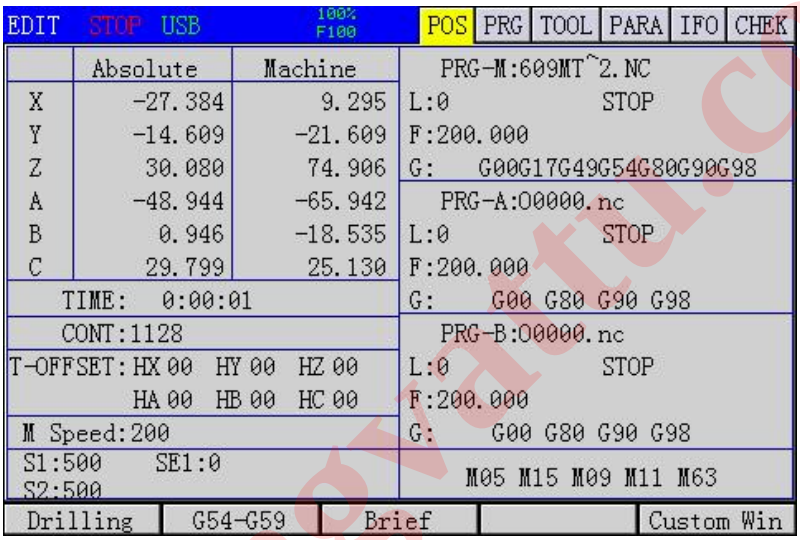
2.2.3 soft function key menu

Each main page is switched to each sub screen through the soft function key. The function of soft function key is triggered by the user's press lift action, which is classified as follows according to the operation form:

A	Highlight, do not operate the page
B	Enter the next submenu
C	Page display options or display content switching
D	Pop up window

3.3 screen position

2.3.1 picture composition

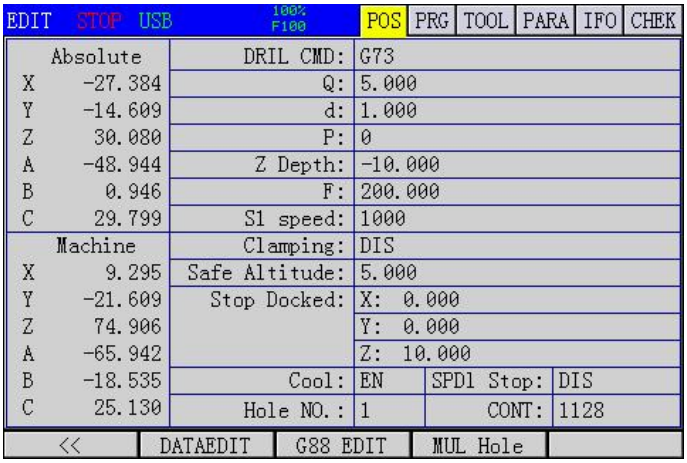


The position main interface displays coordinates, processing time, number of pieces processed, tool offset number of each axis, manual speed, spindle speed, operation information of each program, and some mode m codes.

S1: setting speed of spindle 1; SE1: measured speed of encoder of spindle 1; S2: setting speed of spindle 2

2.3.2 drilling function setting screen

Press the soft function key [drilling function] on the position main page to switch to the drilling page. This page displays the data of drilling function, as shown in the figure:



Explanations:

Drilling instruction: select the fixed cycle instruction required for drilling.

Q: The Q data in the fixed cycle is used to ignore the Q data in the parameters and use the drilling parameters data.

d: D data in the fixed cycle, the drilling function is to ignore the D data in the parameters and use the data of drilling parameters.

d: The delay P (MS) in the fixed loop.

Hole depth: drilling depth, absolute coordinates are used in special drilling, R (drilling start) plane is fixed at 0, and drilling axis is fixed on Z axis.

Drilling speed F: the processing speed when drilling.

Spindle speed s: the spindle speed during drilling. Note refers to the speed of spindle 1.

Automatic clamping: whether to clamp before drilling.

Safe height: the height of z-axis when drilling multiple holes.

End in place: the stop position of each axis after drilling all holes. If it is multi axis, return to Z axis first and then return to other axes.

Cooling on: whether to turn on cooling.

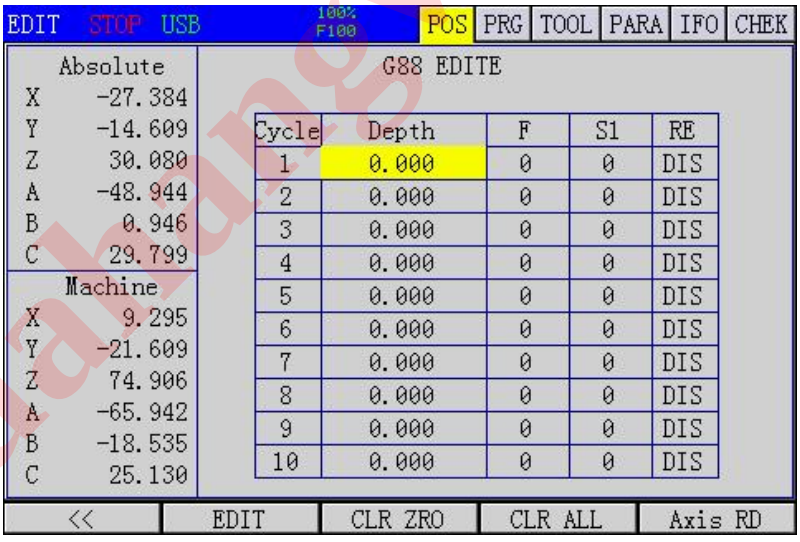
Stop spindle: stop spindle 1 after drilling.

Current hole number: the current number of holes in multi axis, which can be modified when stopping and starting from the current hole.

Machining count: Statistics of workpiece quantity.

2.3.3 G88 editing

Press the [G88 edit] soft function key to switch to this screen. The total drilling speed can be specified by G88



Depth: the depth of each segment. Note that it is an absolute value. If 0 is encountered, it will end after the depth specified by the program. If the hole depth is greater than the depth specified in the program or drilling function, the end of the hole is the depth specified in the program or drilling function.

Speed: the speed of each segment. If it is 0, the speed specified by the program will be followed.

Speed: refers to the speed of spindle 1, which cannot be controlled by spindle 2. If it is 0, the speed specified by the program in this section.

Chip removal: according to the way of chip removal.

[depth reading] the current absolute value coordinates of Z axis can be read in.

2.3.4 multi hole editing

If it is a multi axis controller (larger than one axis), press [multi hole edit] to enter this page. As shown in the figure:

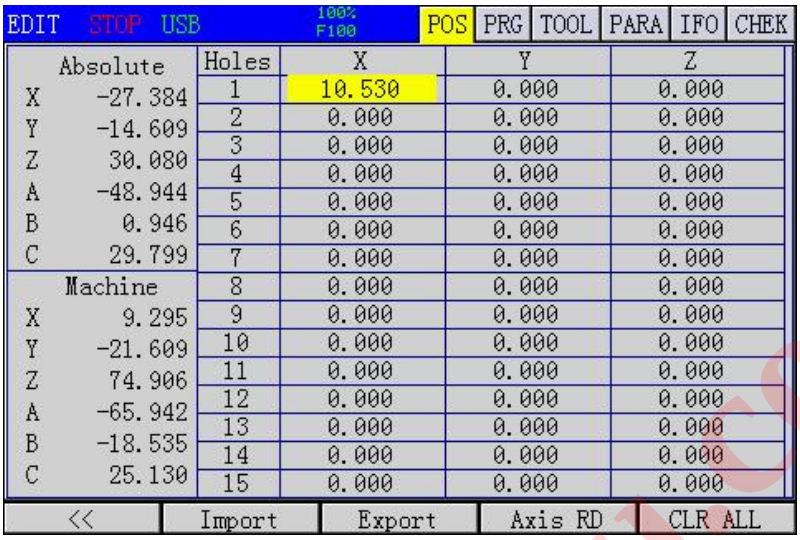


Figure 2.3.4 coordinate system screen

This function can be used to set multi hole processing. The number of holes can reach 9999.

The drilling axis is z-axis, the others are non machining axis.

All coordinates are absolute coordinates.

If it is not the first hole, if the non machining axis data is all 0, then the processing ignores the later data and the processing is finished.

That is to say, the non machining coordinates of the first hole can be all 0, and other holes can not be all 0.

[import data] the processing data in U disk can be imported.

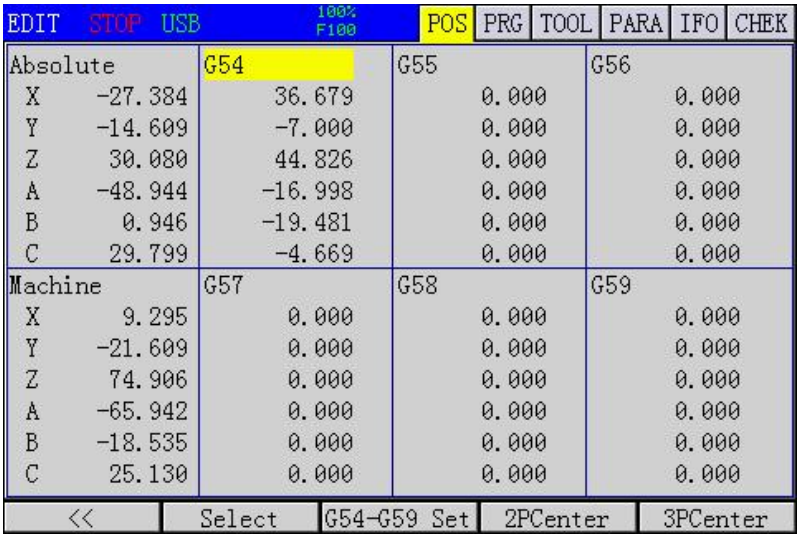
Data format requirements: it must be in CSV format and saved in Excel as CSV format. The first column is X-axis data, the second column is Y-axis data (if there is no y-axis, please give 0), and the third column is hole depth data. If the hole depth data is 0, it will be processed according to the hole depth in the drilling function.

for example

1	2	0
1	4	0
4	4	0
0	0	0

2.3.5 coordinate setting

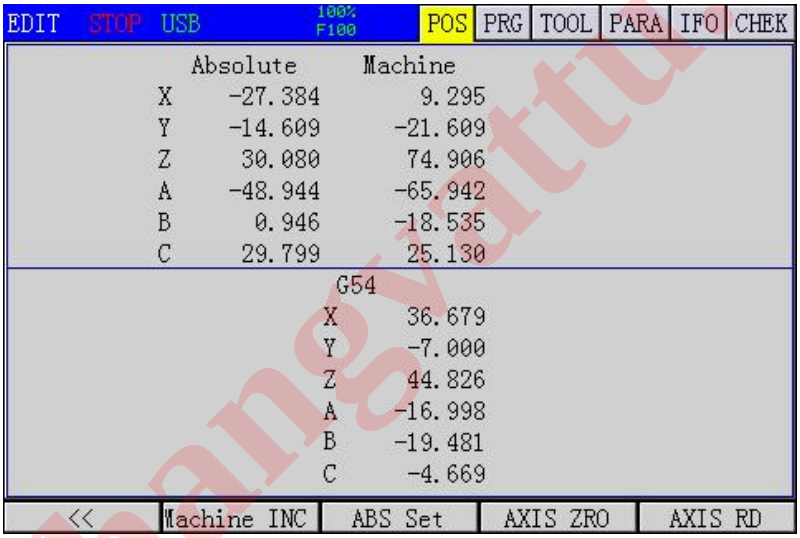
Position main interface, press [coordinate setting] to enter this page. As shown in the figure:



This page sets G54-G59 working coordinate system.

2.3.6 setting coordinates

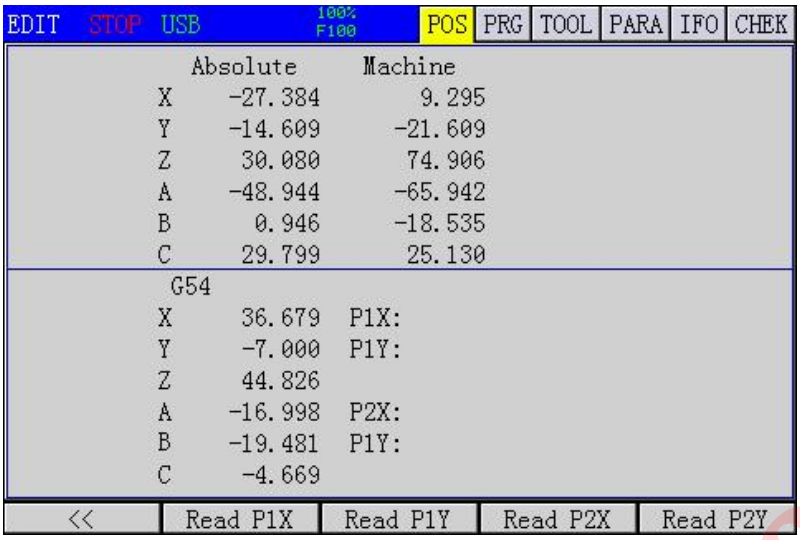
Press [set coordinates] to enter this page.As shown in the figure:



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

2.3.7 line segmentation

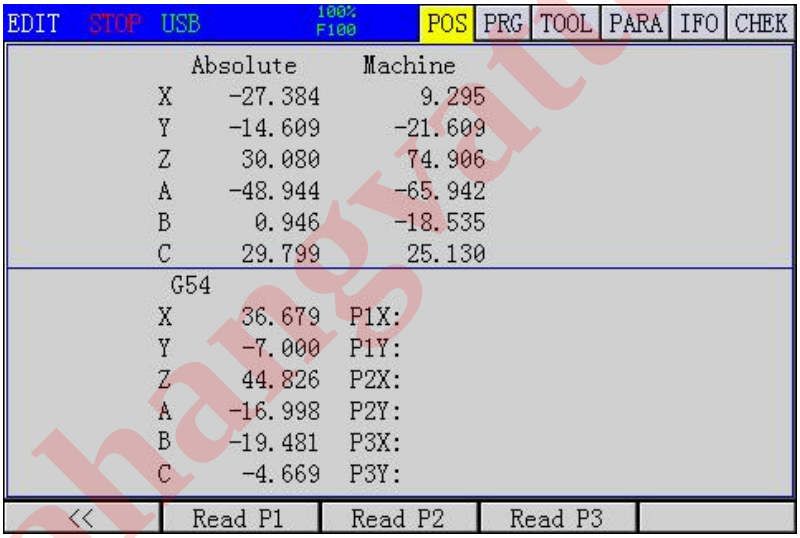
If it is a multi axis system (greater than one axis), press [segment division] to enter this page.As shown in the figure:



If it is three axis or above, press [plane switch] to switch plane.

2.3.8 center coordinates

If it is more than one axis, enter the page.As shown in the figure:

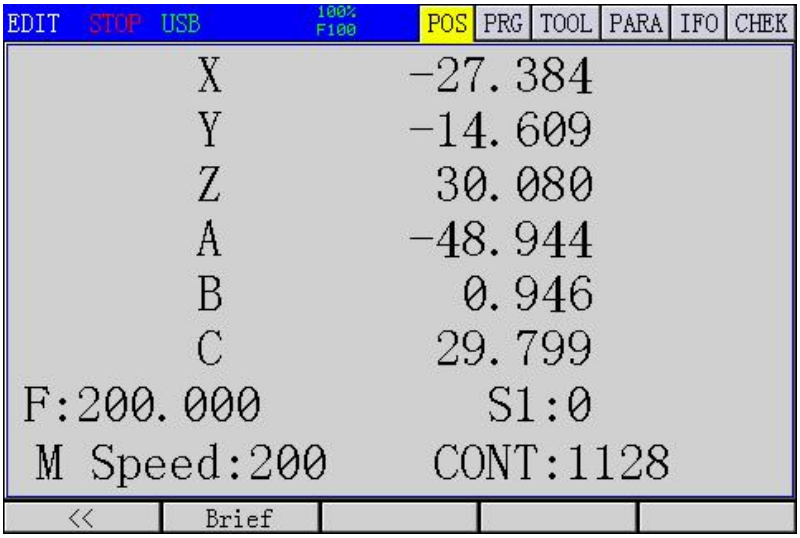


This page calculates the center of the circle according to the three non collinear points in the plane and sets it as the offset of the workpiece coordinate system.

If it is three axis or above, press [plane switch] to switch plane.

2.3.9 brief display

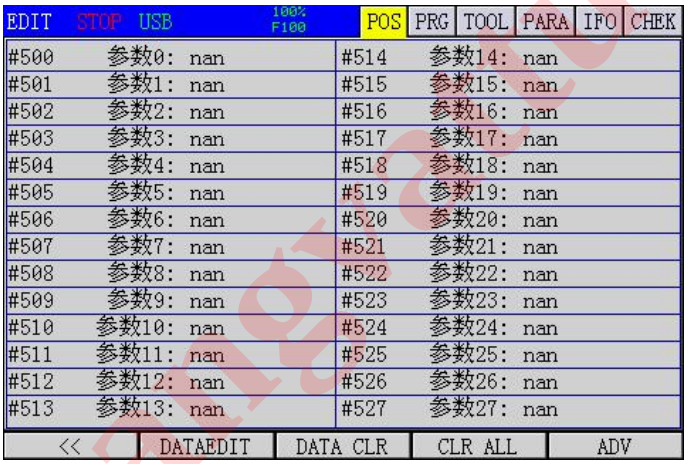
Press [brief display] to enter this page.As shown in the figure:



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

2.3.10 user interface

In the main interface, press user interface to enter this page.As shown in the figure:

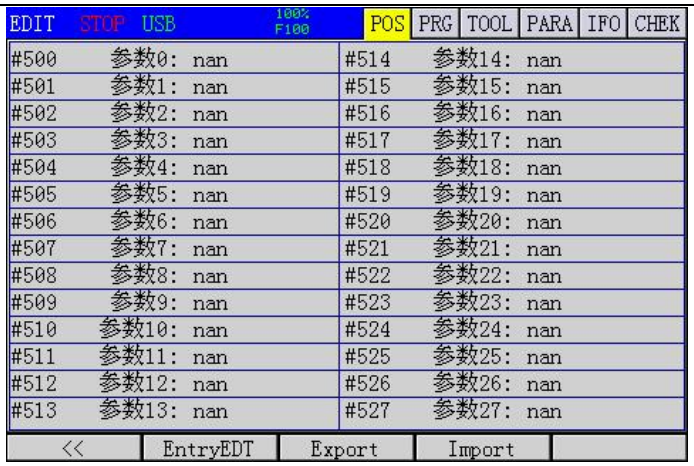


In this interface, 28 parameters can be defined, ranging from 500 to 599100 variables.

This function is mainly used to display the parameter variables that need to be changed frequently on the desktop, so that the user can quickly modify the parameters that need to be changed frequently, and it is not necessary to repeatedly enter the program to change the processing program. It simplifies the operation difficulty of CNC system, and is convenient, fast and convenient to use. Workers with low culture and no understanding of CNC system can operate the machine tool quickly and conveniently and greatly reduce the production cost.

2.3.11 user interface management

Press [interface management] to enter this page.As shown in the figure:



Here's how to set up a user interface

Operation steps

The control system panel is directly established

1. Press the [entry edit] soft function key, and the yellow area appears.

2 press the [modify] button to input the variable number (500 ~ 599).

3 press the →] button, and the yellow area appears.

4 press the [modify] button to input the user-defined variable name, such as speed1.(Note: the system panel directly establishes the user interface, only English variable name can be input)

5 press the [Enter] button to complete the establishment of a user parameter.

6 repeat steps 1-5 to continue to create user parameters.At the same time, the display can create up to 28 user parameters.

Introduction and establishment of PC computer

1 open excel.

2. Input the variable number (500 ~ 599) according to the figure below.(see the figure below)

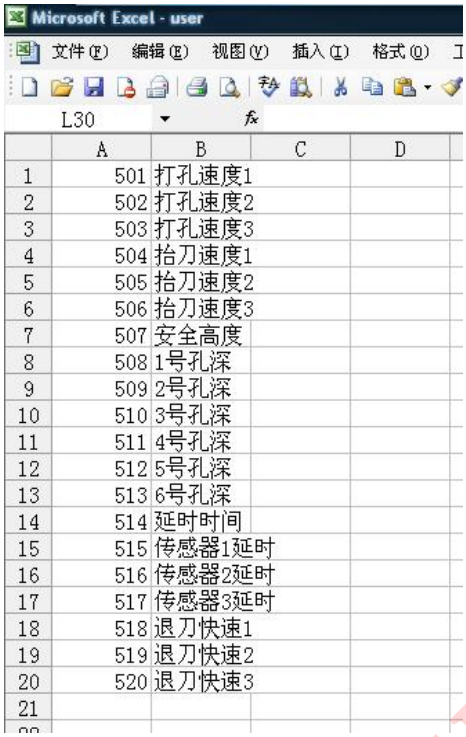
3. Input user-defined variable name, such as safe height, hole depth, etc.(Note: variable name can be Chinese or English)

4. Save as CSV file format with the file name of usercsv。

5 copy into USB flash disk.Plug in the control system USB port.

6 in the user interface screen, press the [interface import] soft function key to select the path and file name.Press the [Enter] button.

7. The user interface is successfully established.



	A	B	C	D
1	501	打孔速度1		
2	502	打孔速度2		
3	503	打孔速度3		
4	504	抬刀速度1		
5	505	抬刀速度2		
6	506	抬刀速度3		
7	507	安全高度		
8	508	1号孔深		
9	509	2号孔深		
10	510	3号孔深		
11	511	4号孔深		
12	512	5号孔深		
13	513	6号孔深		
14	514	延时时间		
15	515	传感器1延时		
16	516	传感器2延时		
17	517	传感器3延时		
18	518	退刀快速1		
19	519	退刀快速2		
20	520	退刀快速3		
21				

How to modify user parameters

Press the [parameter modification] soft function key, select the first parameter, press the [modify] button, input the required parameter value, and press [Enter] to confirm the modification.

Then press the →, and ← keys to select the parameters to be modified, and repeat the above operation to modify.

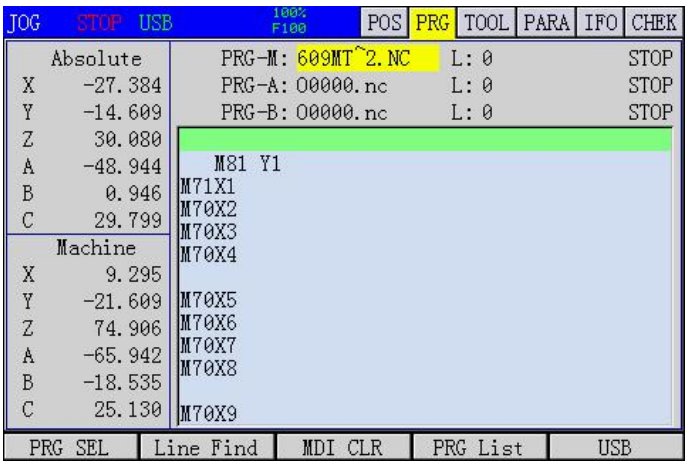
How to call parameters

For example, 諒 represents the common variable number. The value in 諒 is called or modified in the program, such as G1 x 諒 508. After execution, the x-axis moves to 50.000.

2.4 program screen

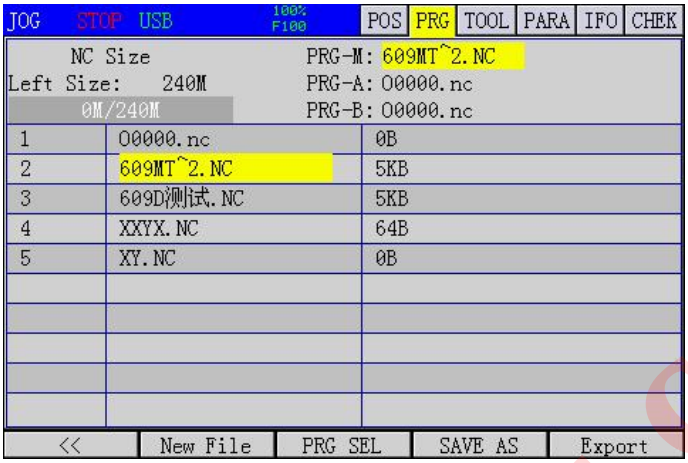
The program screen includes three sub screens: program content, local directory, and U disk directory. These sub screens can be switched to by soft keys [program content], [local directory], [U disk directory].

2.4.1 program content screen



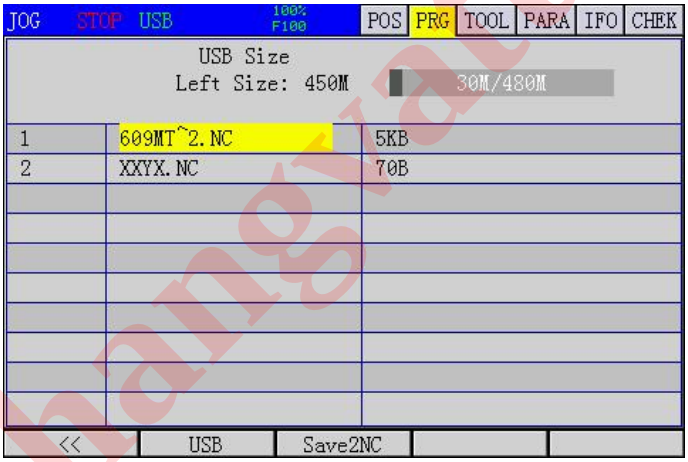
This interface displays the current part program. Move the cursor up and down to preview the program content line by line, and press [pageup] and [pagedown] to preview the program content in ten lines. Use [program switch] to switch between main program and sub program a and sub program B.

2.4.2 local directory screen



The local catalog lists all part programs stored in CNC and shows the status of storage space. Note o0000NC is a space occupying file. This file cannot be edited or deleted.

2.4.3 USB flash disk directory screen

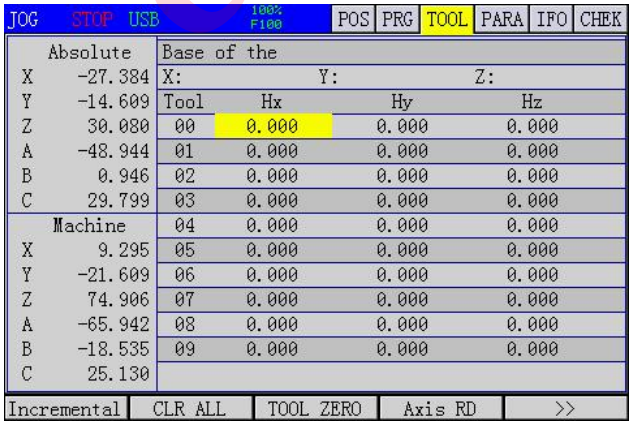


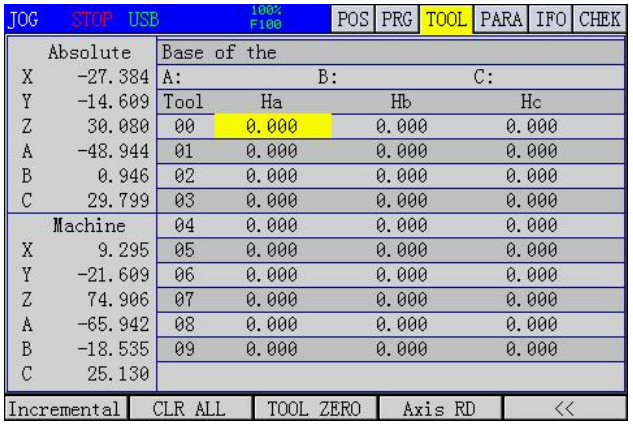
Press [program import] to import the selected file into the local directory.

2.5 offset screen

The offset screen contains two sub pictures: X, y, Z cutter compensation and a, B, C cutter compensation

2.5.1 tool compensation screen





Setting cutter compensation: use the [insert / modify] button for absolute input and [incremental input] for incremental input. No. 0 cutter compensation cannot be modified.

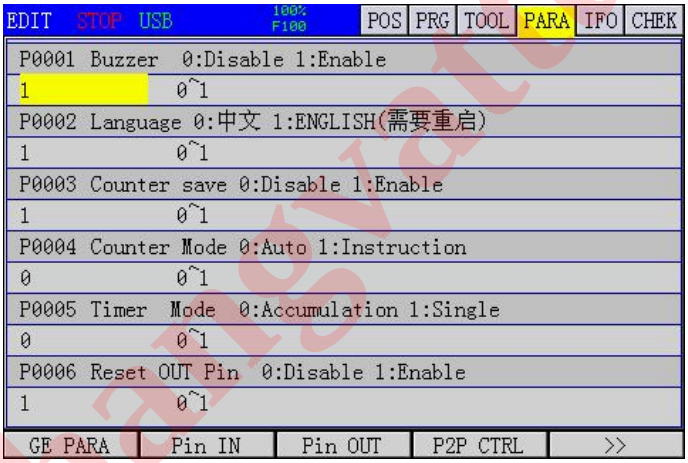
Tool setting method: take a tool as the reference, move the tool tip to a reference point, press [tool setting reference] for axis selection, input the selected axis machine tool coordinate to the reference coordinate, then change the target tool, move the tool tip to the reference point manually or by hand wheel, press [tool setting reading in], select positive and negative (because positive and negative tool compensation cannot be specified except for Z axis). If the tool is parallel to the tool setting axis, use positive direction, such as Z axis, if it is perpendicular to tool setting axis, use negative direction. The system automatically calculates the difference between the length of the tool and the reference tool, and stores it into the corresponding tool complement number.

2.6 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.

2.6.1 comprehensive parameter screen

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

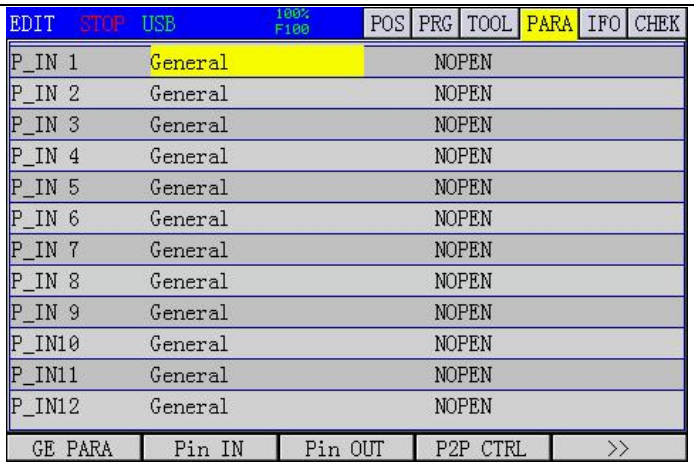


Note: the position units in the parameters are all 0.001, for example, G73 tool withdrawal 1000 represents 1mm.

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

2.6.2 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.

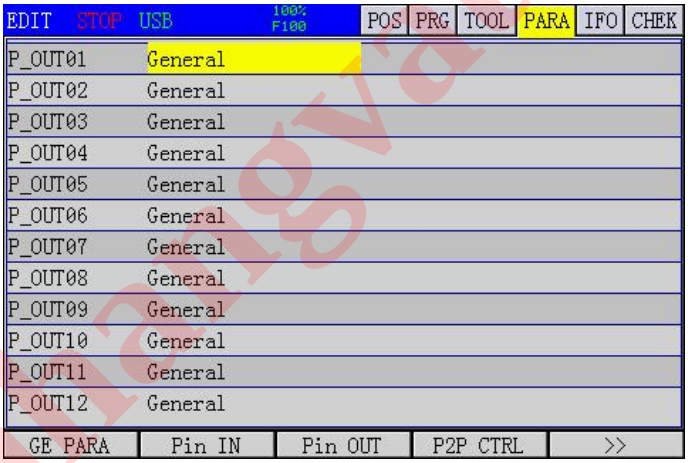


1 ~ 48 input ports can be configured as buttons, limit, alarm, handwheel shaft selection and other functions, while 25 ~ 48 are expansion input ports. The speed of extended input ports is slow. Please do not configure limit, alarm and other functions that need to be detected in time. In the editing mode, 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.

2.6.3 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.



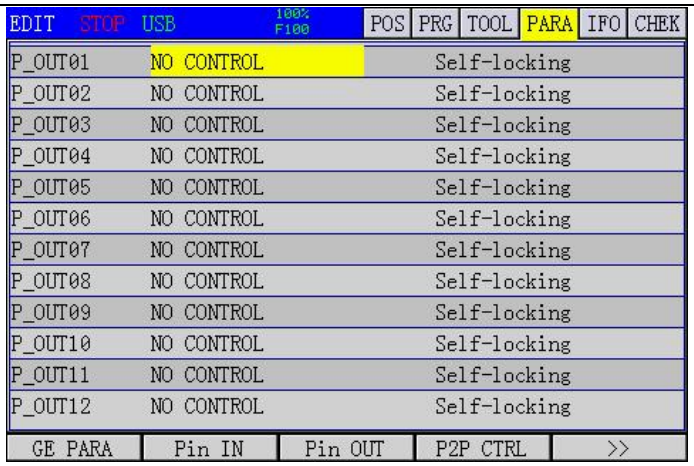
The output port can be configured as spindle cooling, clamping, spindle forward rotation and reverse rotation signal, state tricolor lamp, which is convenient for programming.

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.

2.6.3 direct control parameter screen

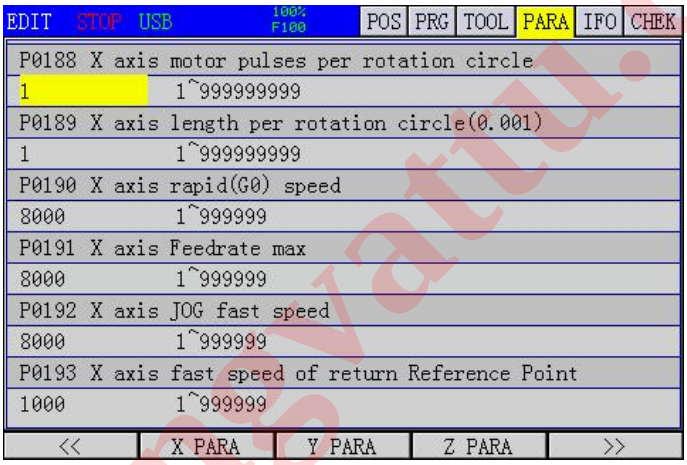
Direct control configuration can be configured to directly control the output port in manual mode.



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.

2.6.4 axis parameter screen

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



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 200 kHz

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

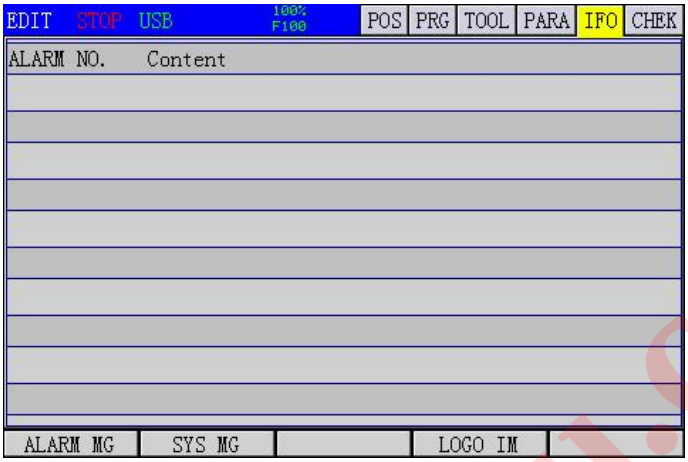
2.7 information screen

The parameter screen includes three sub screens: alarm information and system information. These sub screens can be switched to by soft keys [alarm information] and [system information].

2.7.1 alarm information screen

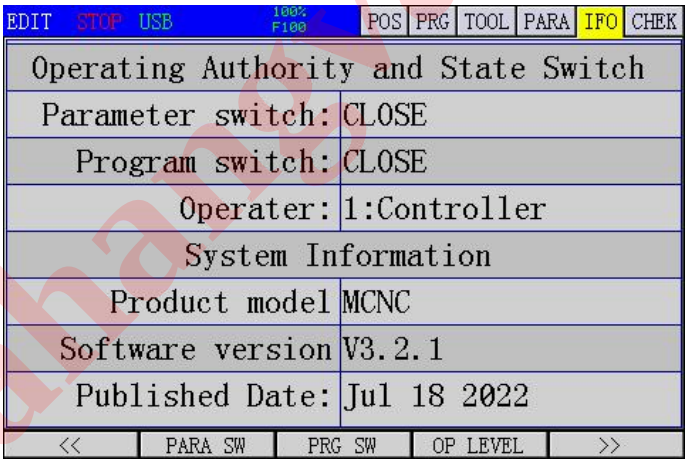
The alarm information screen displays the list of current alarms, and each alarm message contains the alarm number and alarm content. The alarm content is a brief description of the alarm and its release method.

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.

2.7.2 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

2.7.3 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.

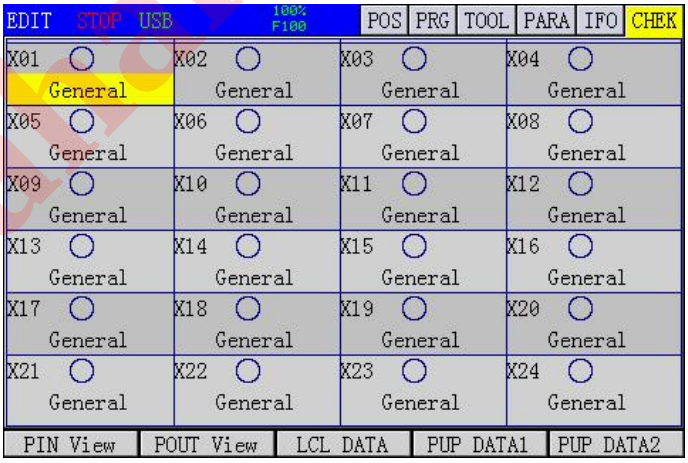
Factory default: 888888

2.8 diagnosis screen

The diagnosis screen includes six sub screens: input diagnosis, output diagnosis, local variable, common variable 1 and common variable 2. These sub screens can be switched to by soft keys [input diagnosis], [output diagnosis], [local variable], [common variable 1], [common variable 2].

2.8.1 Diagnosis input screen

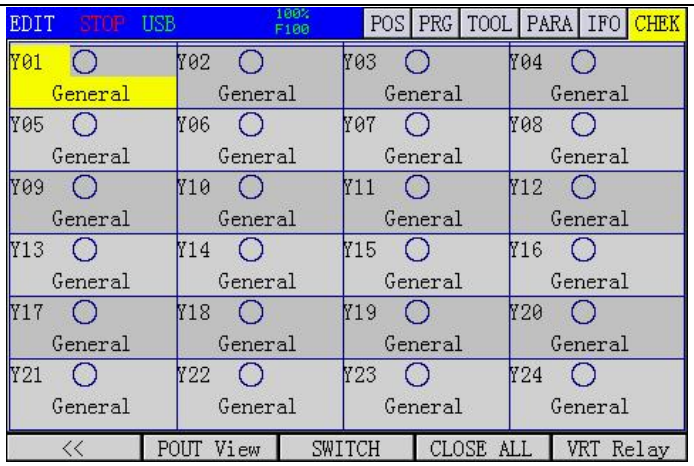
As shown in the figure below, when the external input signal is valid, the circle of corresponding input point (in01-in96) will be filled and displayed, and the port function name is below the port number.



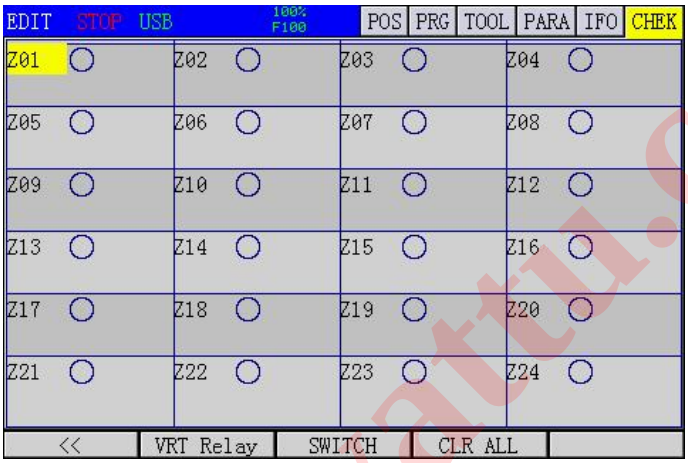
In editing mode, press modify to set the function of input port.

2.8.2 output diagnosis screen

[output switch] and [all off], the corresponding output points (out01-out96) can be tested. When opened, the circle will be filled with display, and the load of the corresponding port will be turned on. The port function name is below.



2.8.3 auxiliary relay diagnosis screen



Auxiliary relays (z01 ~ z96) have no physical input and output, but they can be used as marker bits for communication between main and auxiliary programs and for flag bits in programs.

2.9 macro variable screen

The macro variable screen displays the value of the macro variable and the coordinate value of each axis. Users can directly read and set the values of these macro variables in the program through the format of "# < variable number >" (such as "○ↄ", or set the values of macro variables in the macro variable screen. The macro variable picture contains three sub pictures, one local variable quantum picture and two common variable sub pictures.

Local variables

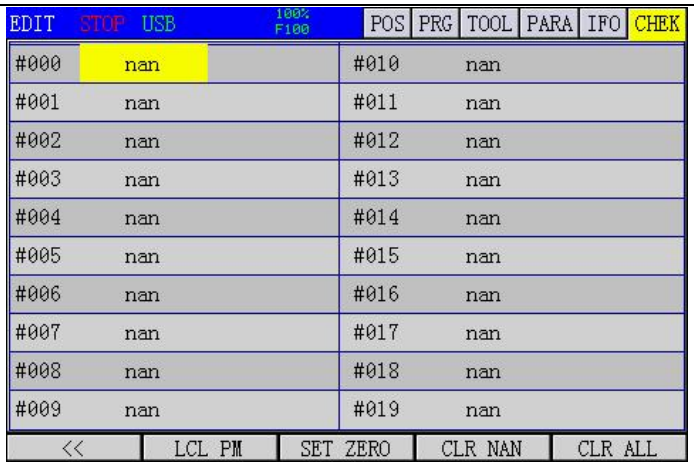
As shown in the figure below, the local variable quantum picture shows the values of ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ○ↄ ○ↄ○ↄ○ↄ○ↄ ○ↄ○ↄ○ↄ ○ↄ.These local variables can be used by the program to hold some internal processing data without affecting other programs. The local variable ○ↄ000 can only be empty and cannot be modified.

Operation steps

1 move the cursor to select the desired variable number.

2 press the [modify] soft function key to input the required value.

3 press enter to confirm the modification. If you do not want to modify it, press the [Cancel] key to return.

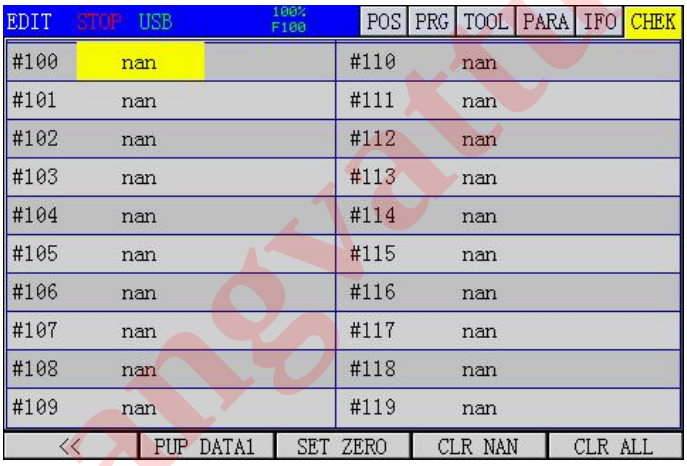


The local variables are cleared each time the program starts running. Press the [F2] key to switch and display the local variables in the main and auxiliary programs.

2.9.2 common variable 1 screen

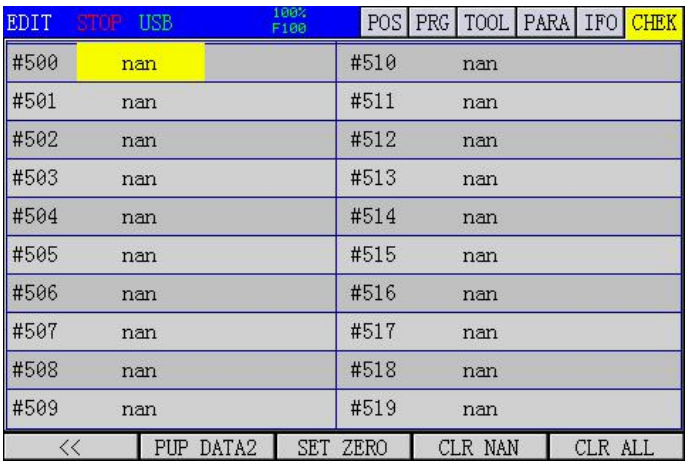
As shown in the figure below, the sub screen of common variable 1 displays the values of macro variables no.100-199. These variables are global and can be accessed by all programs. Users can directly set the value of macro variables through MDI keyboard.

Common variable 1 is cleared every time power is turned on.



2.9.3 common variable 2 screen

As shown in the figure below, the sub screen of common variables 2 displays the values of macro variables ϑ500 and 訟. These variables are also global, but they are stored in CMOS, so they can still maintain the set values after power failure. They are set in the same way as the variables ○ɔ100 ~ ○ɔ199.



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 + Z pulse

3: Limit switch

4: Z pulse

5: None (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 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, manual low-speed feeding with comprehensive parameters is adopted, which is shared by all axes.

Manual rapid feed

In manual mode, pressing 

key can control the manual movement to manual rapid feed. 

is a key with self-locking. When pressed repeatedly, it will switch in the switch state. The manual speed is displayed as fast, indicating that the manual fast switch is open.

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%.

3.3 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, and the input multiple of each step can be 10 times, 100 times and 1000 times.

Operation steps

1. Press the mode key [single step], the system enters the single step feeding mode, and the indicator light on the key is on.

2 press one of the multiple selection keys XTI30425 to select the multiple of the minimum unit. These three keys are compound keys. In single step mode, the text on the key is valid, indicating that the movement is the minimum programming unit×10、×100、×1000。

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.

3.4 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.

XTI30737 e ixt307 key to select a single pulse rate corresponding to the handwheel. These three keys are composite keys. In handwheel mode, the text on the key is valid, indicating that the corresponding movement of 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.

3.5 manual auxiliary operation

3.5.1 manual coolant switch

In manual / handwheel / single step / return to zero mode, press the [cooling] key (external button needs to be set) 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. In either case, M08 is displayed when the coolant is on and M09 is displayed when the coolant is off.

3.5.2 manual clamping switch

In the manual / handwheel / single step / zero return mode, press the [clamp release] key (external button needs to be set) 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 → release → clamp open". In any case, when clamped, it will display M11 and when clamped, it will display M11.

3.5.3 manual spindle control

In manual / handwheel / single step / zero return mode, press the [spindle 1] key, if spindle 1 runs, it will stop;If spindle 1 stops, it will forward.

Press the [spindle 2] key, if spindle 2 runs, it will stop;If spindle 2 stops, it will be forward.

Chapter 4 automatic operation

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

4.1 program operation

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

Automatic mode can run main program, sub program a and sub program B at the same time.

It can also only run the main program. The method is to set whether the auxiliary program in the comprehensive parameters starts with the main program (p0010, p0013).

The auxiliary program can be opened with M31 instruction in the main program, otherwise it will not work, and the auxiliary program cannot run M31 instruction.

The sub program is not completely equivalent to the main program, the main purpose is to do loading and unloading.

Refer to parameters p0010 ~ p0015

For example, program a illustrates:

P0010 = 0 M31 P1 can only be called in the main program to start program a.

P0010 = 1 press the main start key, the main program will run, and program a will also start.

P0010 = 2 is set to [Auxiliary a start] by external key.

Here are some similarities and differences between the main program and the sub program

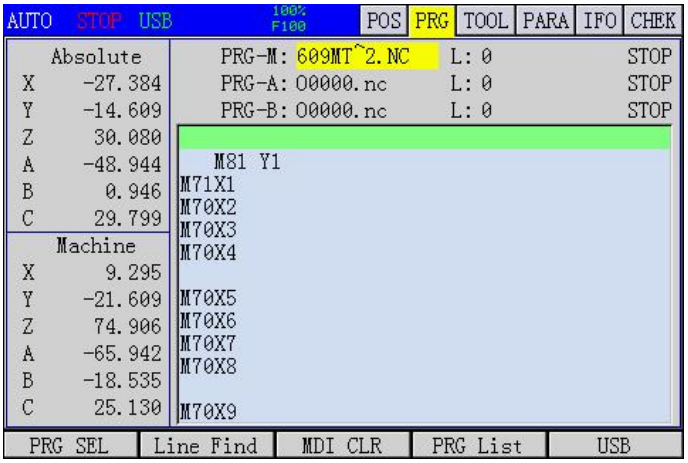
Program function	main program	Subroutine
G1	straight line	Approximate straight line, so the subprogram cannot process the contour.
G2, G3, G12	support	not support
Workpiece coordinate system, tool compensation, spindle	Shared, any program changes, other programs also change at the same time.	
M31, M32 open, close the program	support	not support

In principle, different programs cannot specify the movement of the same axis at the same time. If this is the case, they will be executed in sequence, which program will be executed first, and the other programs will wait. Therefore, the sequence cannot be predicted. One solution is to use auxiliary relays to communicate between programs.

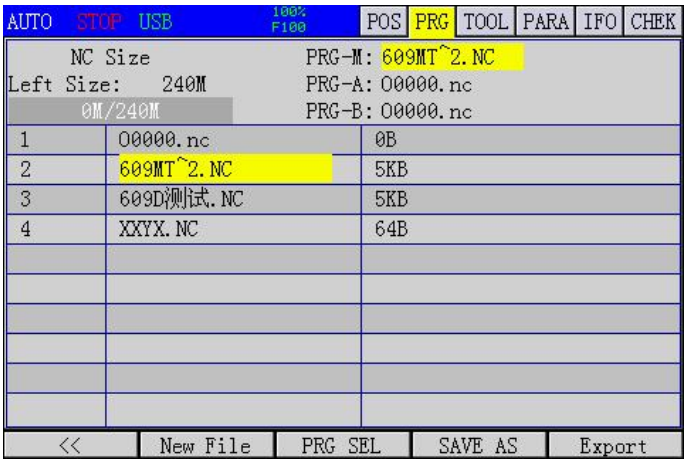
Operation steps

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

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 end the automatic operation immediately, and the tool deceleration stops (Note: the system does not reset and maintains the current state).If the subroutine is currently being called, the system will exit the subroutine and return to the main program, and the system will enter the reset state.

The M99 in the main program

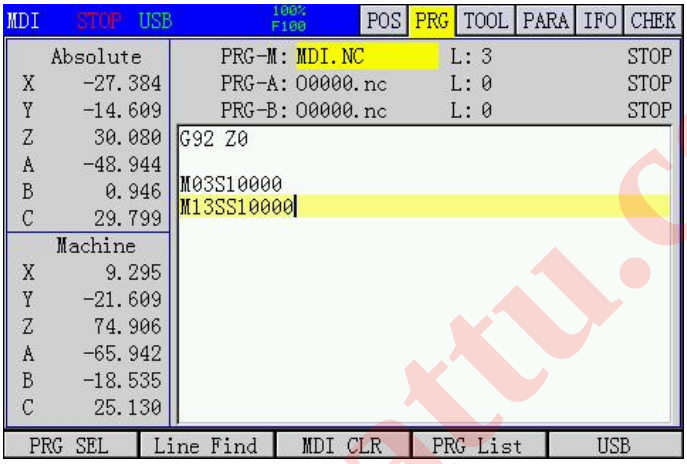
When the program is running, if there is M99 in the main program, after the execution of M99, it will jump to the beginning of the program to continue to execute, count plus one, and repeatedly execute the main program. If l is specified, press l times to stop

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, and press [MDI], as shown in the figure below.



2Press the cursor key【↑】or【Reset】to move the cursor to the beginning of the program.

3 press the [start] key to start 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.

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.

Side program

MDI mode cannot be started.

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.

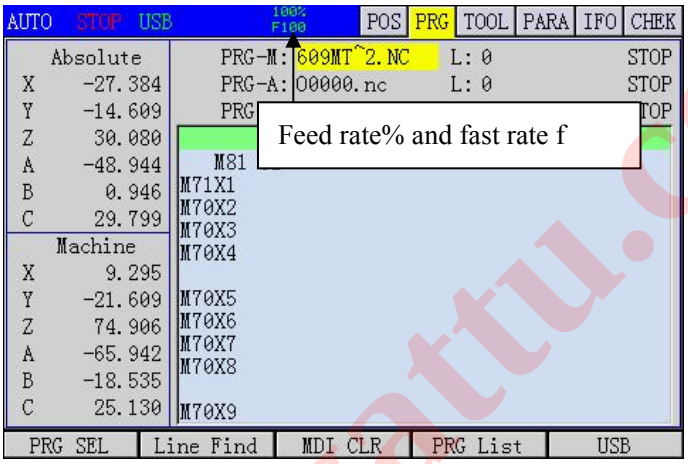
5.1 Feed rate override

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 from 0% to 100%, and the difference of each gear is 10%. The final value of actual feed rate = feed rate / feed rate specified by the program.

Operation steps

Before or during automatic operation, press the feed rate key on the machine operation panel to adjust the feed rate to the required percentage. As shown in the figure below.



5.2 Rapid feed override

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 G28;

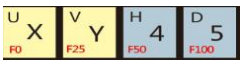
4. Manual rapid feed;

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.

Relevant explanation

The fast magnification is divided into four levels, which can be selected by pressing the four XTI33512 keys on the system operation panel. These four keys are composite keys. The fast magnification corresponds to the text in the second line of the key. The functions of each key are as follows:



Key Icon	Key usage
	Set the fast magnification to the lowest gear F0
	Set the fast magnification to 25%

	Set the fast magnification to 50%
	Set the fast magnification to 100%

Note: the fast rate is F0, which can be specified in the comprehensive parameter (p0024).

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 light 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.

5.4 skip optional segments

When the skip optional segment switch (referred to as the skip switch) is turned on, the system will ignore the block containing the '/'.

The skip section switch is controlled by the [skip section] key on the panel of the machine tool. The [skip section] key is like a button with self-locking. When the indicator light on the key is on, it means that the skip section switch is on. When the indicator light on the key is off, it means that the skip section switch is on. When the indicator light on the key is off, it means that the section skipping switch is on.

Operation steps

Before or during the automatic operation, press the [skip section] key to make the indicator light on the key on. At this time, the system will not execute the program section containing "/".

Operation explanation

When the instructions contained in the program segment are read from memory to the buffer register, the system decides whether to skip the program segment according to the status of the skip switch and whether the segment contains a '/', but the program segment that has been read into the buffer register is not affected by the skip switch.

notes

Users can not accurately predict when the program segment will be read into the buffer register. In order to ensure the effect of the skip switch, it is necessary to run the program automatically There is a certain risk to turn on the trip switch before the line and turn on the trip switch during automatic operation.

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:

1CNC's feed axis is stopped.

2 Accessibility is off.

3. First cut off the CNC power supply, and then cut off the machine tool power supply.

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 adjust the tool again.

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

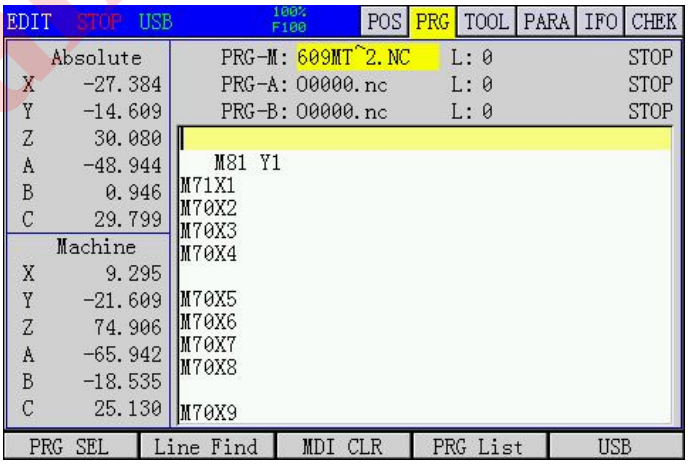
4 use the address keys, number keys, backspace and function keys of the keyboard to insert and delete programs

explain

Program area

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

程序区是指系统中程序显示和编辑的窗口。如下图所示。



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

3 press [program switch] to switch the display of main and auxiliary programs.

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. The user can turn it off in the system information screen in chapter 2.7.3.

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.

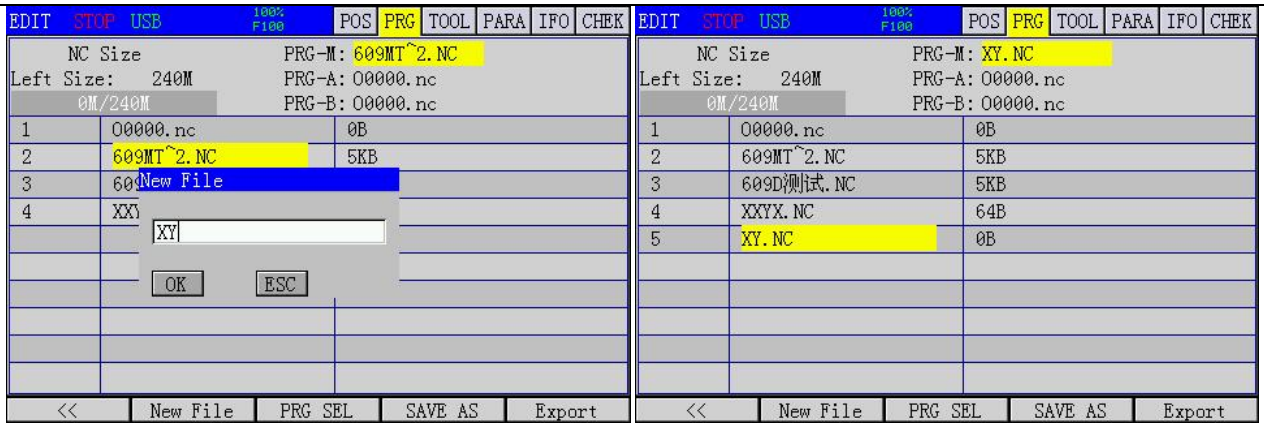
When you press the character keys x, y, Z, a, B, C, I, J, K, if the corresponding axis is valid, the current absolute coordinates will be automatically added.

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, so it can be input normally.

7.3 new program



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 XY of processing program.As shown in Figure a.

4 press enter to confirm.As shown in Figure B, XY appears on the leftNC file name.If it is a subroutine, please name it oxxxx, 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 INSTRUCTION

Chapter 1 Introduction to programming 46

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

绝对值指令使用轴移动的终点位置坐标值进行编程。也即是刀具移动到终点的坐标位置。如图 2-1

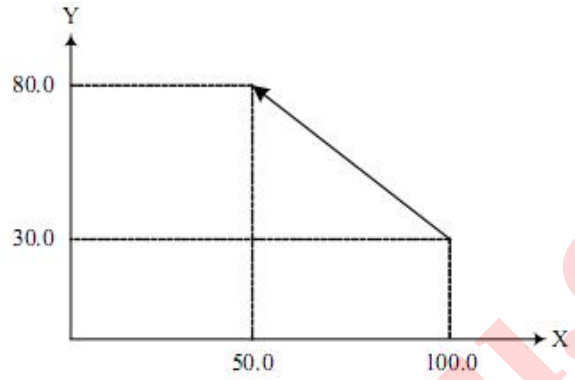


Figure 2-1 Schematic diagram of absolute value command and incremental value command

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 control shaft

1.3.1 number of control axles

Number of control axes

Number of control axes	1~6
Control axis name	X、Y、Z、A、B、C

1.3.2 Units

The units involved in the system include minimum input unit, minimum output unit and minimum moving unit.

Minimum input unit

Also known as the minimum setting unit or minimum programming unit, it refers to the minimum unit of the amount of movement during programming, or the minimum unit of absolute coordinates. In mm or deg.

Minimum output unit

Also known as the minimum unit of machine tool, refers to the minimum unit of machine tool coordinates. In mm or deg.

Minimum moving unit

The minimum unit of command sent to the driver (when the system electronic gear ratio is 1:1, it represents the length or angle represented by one pulse). In mm or deg.

Quick guide:

Minimum unit of linear axis

The minimum input unit for a linear axis is metric input.

Minimum unit of rotation axis

The units of the axis of rotation are expressed in deg.

The minimum unit is suspended

The minimum unit of pause is 0.001 seconds, regardless of the minimum unit of linear axis or rotation axis.

For detailed setting unit, please refer to the instruction manual of the machine tool manufacturer.

1.4 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 X1.0 is considered 1.000, which is different from other controllers.

Chapter 2 Program Composition

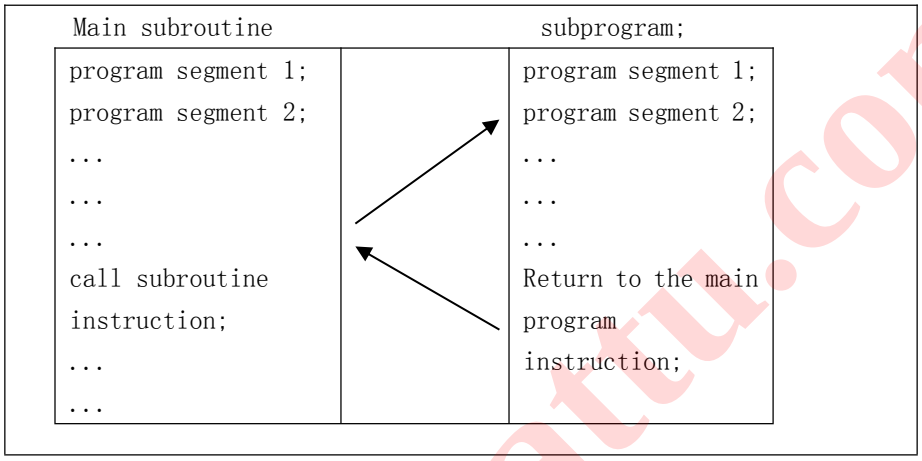
2.1 procedure

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 main program and subprogram

2.1.1.1 main program

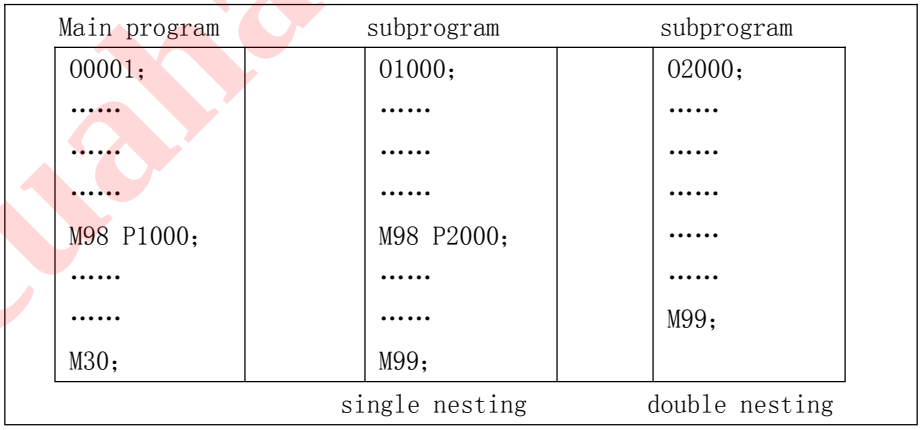
The program is divided into main program and subprogram. Generally, CNC machine tool moves according to the instruction of main program. If there is instruction calling subroutine during the execution of main program, it will be executed by subroutine. When the instruction returning to main program is encountered in subprogram, it will return to main program to continue execution.



2.1.1.2 subroutine

When there are some fixed sequence and repeated program segments in the machining program, they can be edited as subprograms. The main program can call the subroutine when necessary, so the programming can simplify the main program. The subroutine can be called out when the main program is running, and the called out subroutine can also call another subroutine.

Main program subprogram subprogram

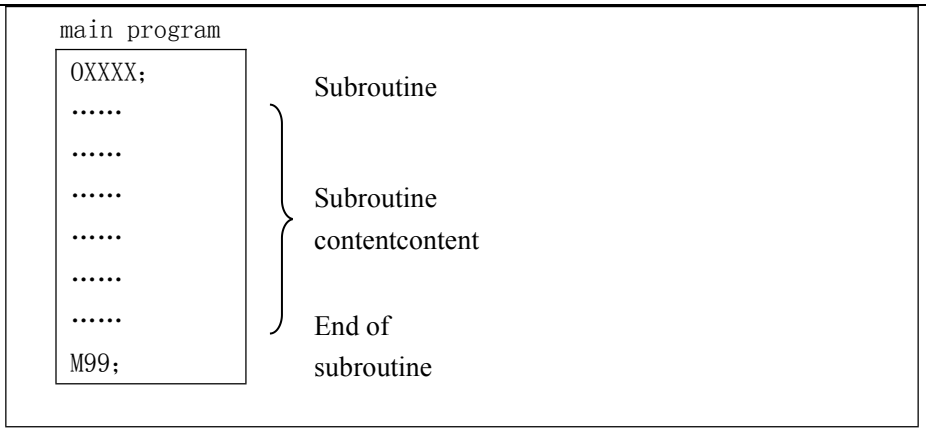


Subroutines can be embedded in 4 layers.

Quick guide:

Preparation of subroutines

Write a subroutine in the following format:



At the beginning of the subprogram, write the subprogram number after the address 0, and at the end of the subprogram is the M99 instruction.

For example:

example
X…M99 ;

In order to be used interchangeably with other devices, the subroutine in the beginning block can use Noooo instead of oooooo. The sequence number immediately following n is stored in the memory as a subroutine number.

The execution of subprogram

The subroutine is called out and executed by the main program or subroutine calling instruction. The instruction format of calling subroutine is as follows:

M98 PXXXX LNN;
XXX: the ordinal number of the subroutine being called.
If the number of repetitions is omitted, the number of repetitions is considered to be one.

If the number of repetitions is omitted, the number of repetitions is considered to be 1.

example
M98 P1002L5 ;

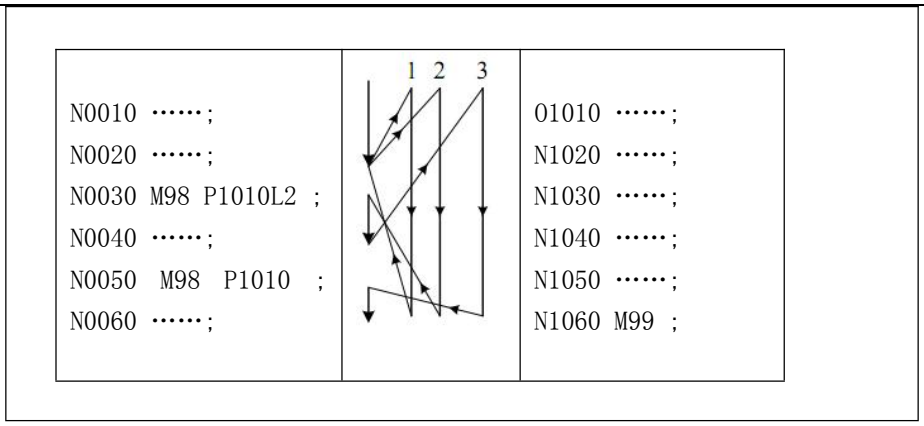
Indicates that the subroutine with program number 1002 is called five times in a row.

The M98 instruction can also be programmed in a block with the move instruction.

example
X1000 M98 P1200 ;

At this time, after the X-axis movement is completed, subroutine No. 1200 is called.

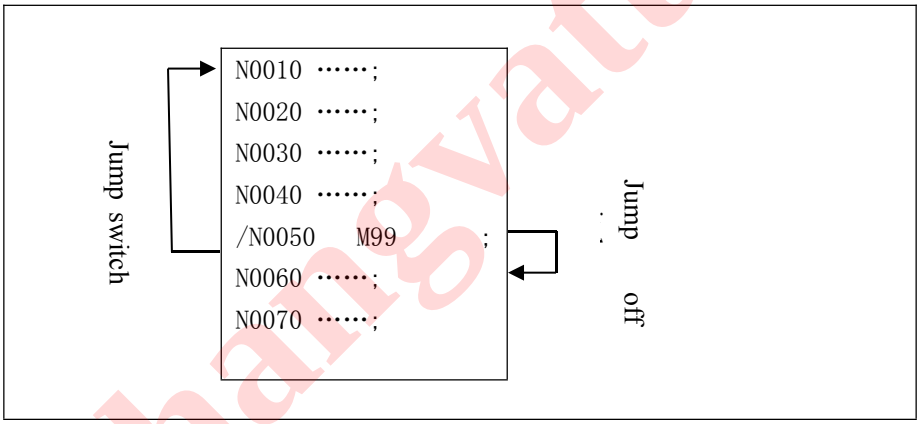
The order in which subroutines are called from the main program is shown in the following example.



Calling a subroutine in a subroutine is the same as calling a subroutine in the main program.

Note:
the subroutine number specified by address P cannot be retrieved, an alarm is generated

2 in the main program, if M99 is executed, it returns to the beginning of the main program and continues to execute repeatedly.For example, there is a segment / M99 in the main program. If the switch of skipping any program segment is off, M99 will be executed and returned to the beginning of the main program and repeated from the beginning.During the execution of the program, the program will be skipped repeatedly during the execution of the program.



2.1.2 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 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. For example, when the tool is changed, or when the table index is moved to a new machining surface, etc.

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 ;
```

In the above program, if the skip switch is on, the N101 block is skipped.

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	X Y Z A B C U V W H	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, SS	Spindle speed specification, S spindle 1, SS spindle 2
刀具功能	T	Designation of tool number
辅助功能	M	Machine tool auxiliary function assignment
偏置号	H, HX,	The designation of the tool compensation offset

	HY,HZ,HA,HB,HC	number of each axis, H and HZ are the same.
暂停	P/X	Designation of Pause Time
子程序号指定	P	Specify the subroutine number
重复次数	L	the number of repetitions of the subroutine
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
程序号	O	1～9999
顺序号	N	Preparation function
准备功能	G	0～99
尺寸字	X Y Z A B C U V W I J K Q R	±999999.999
每分进给	F	0.001～15000.0
主轴功能	S	0～9999
辅助功能	M	0～99
暂停	X P	0～999999.999S
Number of repetitions 重复次数	P	1～9999
重复次数	L	1～99999
偏置号	H，HX，HY,HZ,HA,HB,HC	0～99

These parameters are the commandable range of the CNC system and have nothing to do with the actual working range of the machine tool. For example, the system can instruct the axis movement to be about 100m, while the actual machine tool X-axis travel may only be 2m. When writing a program, refer to this manual and the machine manual at the same time.

2.2 end of procedure

The program ends with M30 or M99.

format

```
M30; End of procedure
M99; End of subroutine
```

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.At the end of the subroutine, the system returns to the program calling the subroutine to continue execution.

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:

G00 和 G01 为同组模态 G 代码。加工程序如下：

G00X__; (G00 有效)

Y__; (G00 有效)

G01Z__; (G01 有效)

X__;

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 comprehensive parameters
G01		Linear interpolation, run according to the given F
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	Return to the parameter point (return to mechanical zero), involving parameters including speed, zero return direction and method
G31		Jump mechanism, if the G31 input port is valid, stop the current movement to the next command
G50		Lateral movement, the input port is invalid and stopped, similar to G31, but the input port can be arbitrarily specified
G51		Lateral movement, the input port is effectively stopped, similar to G31, but the input port can be arbitrarily specified
G22	0	loop instruction
G23		loop instruction
G43	05	Tool length compensation in the positive direction, independent of the plane, always in the Z axis, other axes are invalid
G44		Tool length compensation in the negative direction, independent of the

		plane, always in the Z axis, other axes are invalid
G49 *		Tool length compensation is canceled, always in Z axis, other axes are invalid
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
G74		Anti-tapping cycle, tapping according to the encoder.
G80 *		Canned cycle cancellation
G81		Drilling cycle (spot drilling cycle)
G82		Drilling cycle (boring step empty cycle)
G83		Deep hole drilling cycle
G84		Tapping cycle, tapping by encoder
G85		boring cycle
G86		Drilling cycle
G88		Customized drilling commands, edit G88 in the re-drilling function of specific actions.
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 with * is the default G code of the system. When the power is connected, the mode G code will be in the default state.

The G code of group 200 is modeless G code, which is valid only in the current program segment.

3 if a G code not listed or enabled in the G code list is used, an alarm will appear.

Several different groups of G codes can be instructed in the same block. If multiple G codes of the same group are instructed 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

G00 IP__;

IP: X, Y, Z, A, B, C, U, V, W, etc., representing any combination of axes. For the absolute value command, it is the end point coordinate value of the tool movement, and for the incremental value command, it is the tool movement amount.

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

Instruction description

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

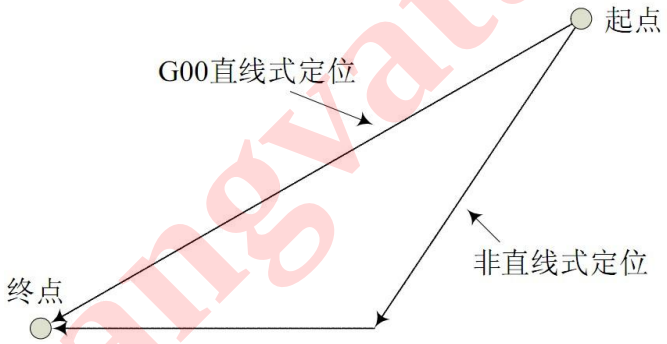
Linear and non-linear can be selected in the comprehensive parameters

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:

两种定位方式示意图如下：



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__ ;

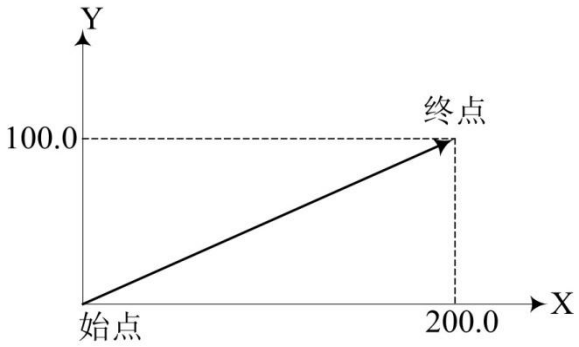
F: tool feed rate.

Instruction description

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 in the XY plane

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

Arc in the XZ plane

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

Arc in the YZ plane

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

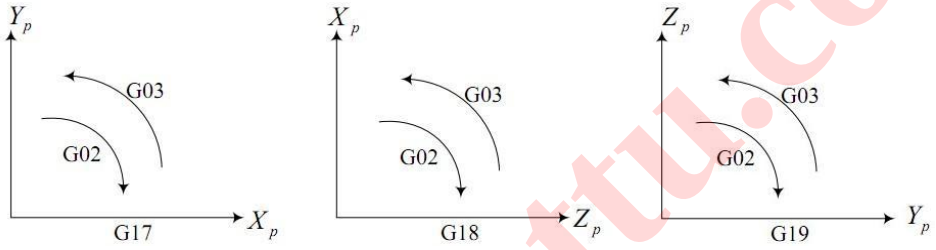
project	Specified content	command	describe
1	Plane designation	G17	XY plane arc designation
		G18	ZX plane arc designation
		G19	YZ plane arc designation
2	Interpolation direction	G02	Clockwise circular interpolation (CW)
		G03	Counterclockwise circular interpolation (CW)
3	End position or	Two axes in X Y Z	End position in absolute coordinate system

	distance	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

Instruction description

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

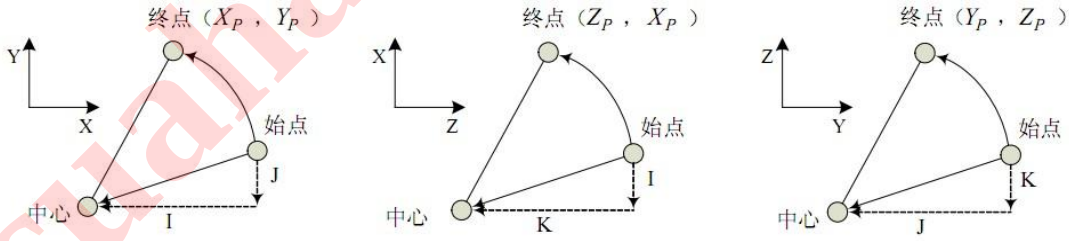


The amount of movement on the arc

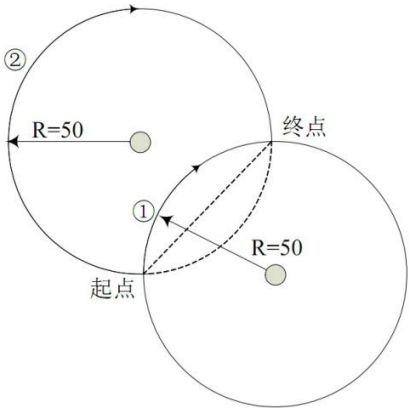
With the address x \Y_Or Z_Specify the end point of the arc. The absolute value is indicated under the G90 command, and the incremental value is indicated under the G91 command.The increment value is the distance from the start point to the end point of the arc.

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 180°, execute G code G91 G02 X60 Y50 R50 F300;

When the arc of ② is greater than 180°, 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

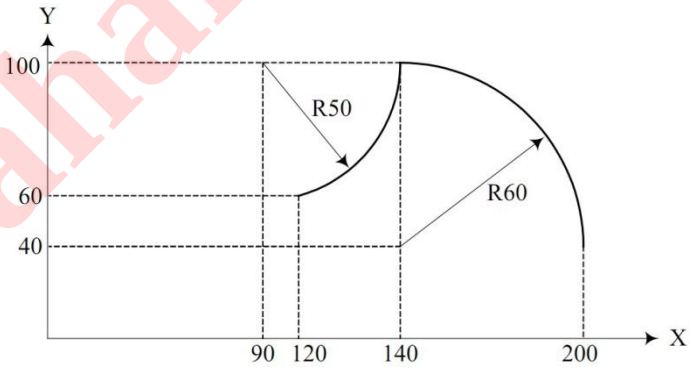
When I, J and K are 0, they can be omitted.

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 ;
```

3.4* 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

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

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

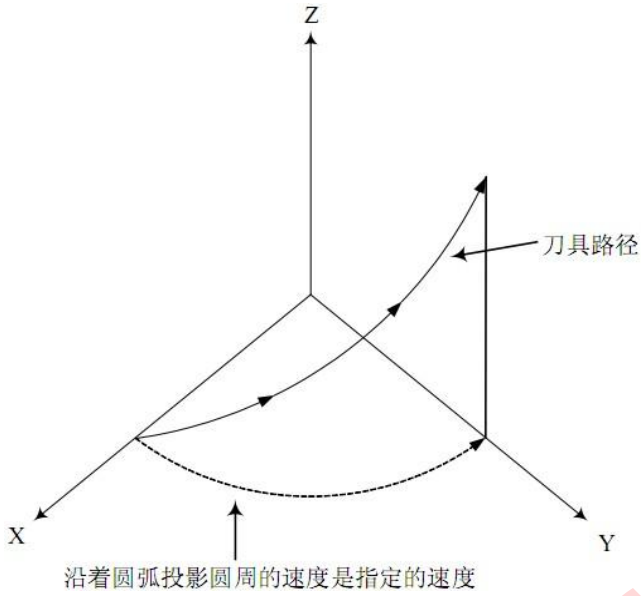
Arc of YZ plane

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

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:

$$F \times \frac{\text{直线轴的长度}}{\text{圆弧投影的弧长}}$$



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 of XY plane
G12 I J X Y
Arc of ZX plane
G12 I K X Z
Arc of 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__ ;
或
G04 P__ ;
P: Delay waiting time setting (decimal is not allowed).

Instruction description

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	0.001 seconds

notes

The 1 x / P command unit is independent of the minimum unit of the linear axis or the axis of rotation.

If P and X instructions are omitted, it can be regarded as accurate stop.

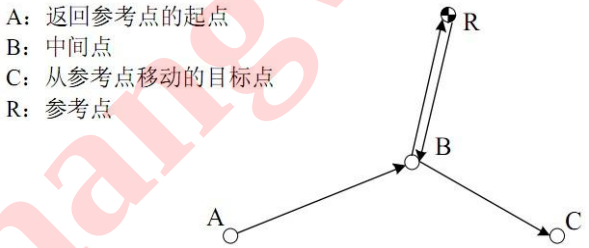
3 execution of G04 instruction will automatically disable read ahead and buffering.

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.

3.7.1 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. The return from reference point function (g29) enables the specified axis to move to the specified position through the middle point. As shown in the figure below.



G28自动返回参考点：A—>B—>R

G29从参考点移动：R—>B—>C

command format

G28 IP__;

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

Instruction description

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.

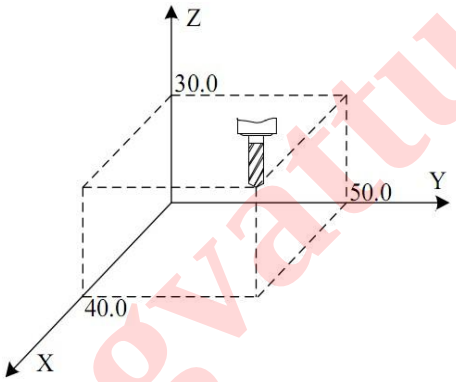
4 in G28, there are several axes in the order of Z > XY > ABC

example
N1 G28 X40.0 ; 中间点（40.0）
N2 G28 Y60.0 ;

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.

3.8 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.

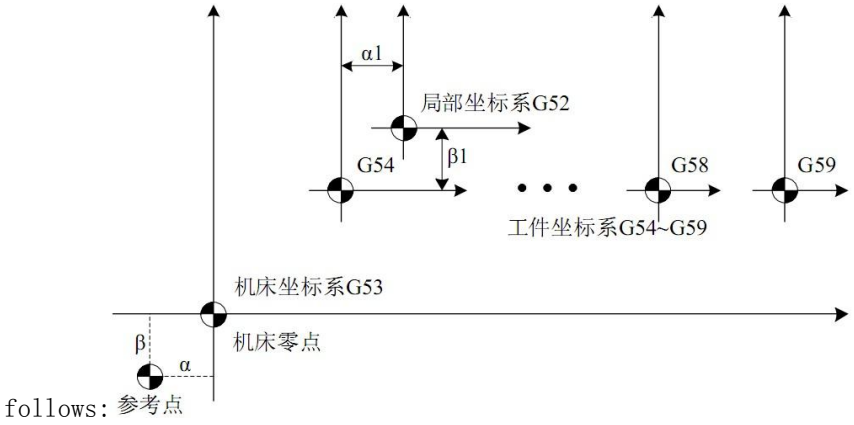


This system uses one of the machine tool coordinate system, workpiece coordinate system and local coordinate system to specify the coordinate position.

Machine zero is a fixed reference point set by the machine tool manufacturer for the machine. The coordinate system with the zero point of the machine tool as the coordinate origin is called the machine tool coordinate system 1. The coordinate system of machine tool is based on the fixed point on the machine tool, and it is the benchmark of other coordinate systems. Once it is established, the system will save it and it will be valid until it is reset.

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:



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

```
G53 IP__;
  Instructions
```

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 g53 command is specified, the compensation such as tool radius and length will be automatically cleared.
 The 2 g53 instruction suppresses the pre reading of G code.
```

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

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

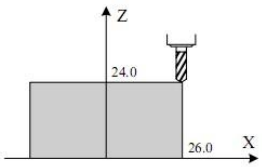
The way of establishing the workpiece coordinate system is to make the specified coordinate value (IP__) become the absolute coordinate value of the point (such as the tool nose) on the current tool in the set workpiece coordinate system.

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.

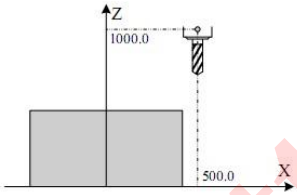
3 Normally, please set the workpiece coordinate system before specifying tool compensation.give an example

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、γWhen 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 ;
```

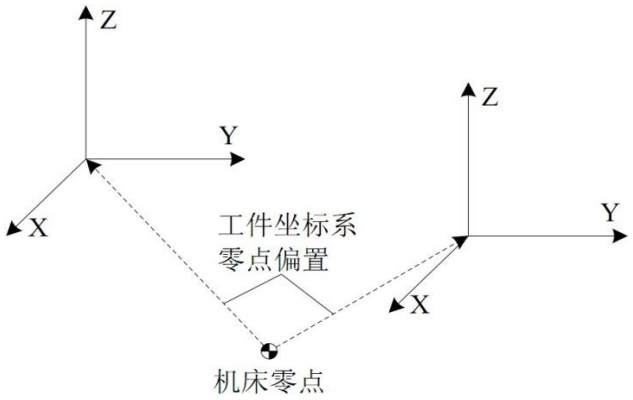
3.8.2.3 Select workpiece coordinate system (G54-G59)

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.

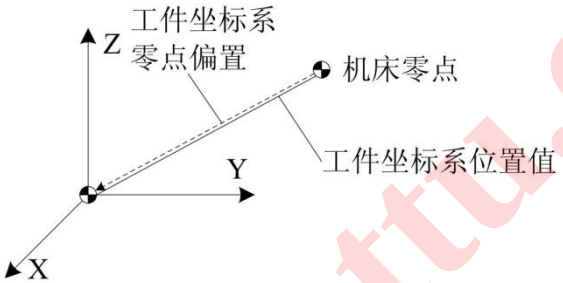
```
G54 工件坐标系 1
G55 工件坐标系 2
G56 工件坐标系 3
G57 工件坐标系 4
G58 工件坐标系 5
G59 Instructions
```

Instruction description

1. 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.

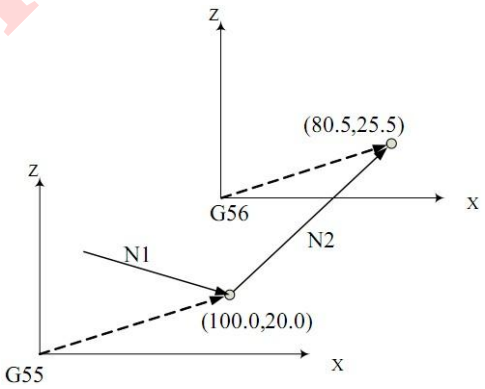


notes

When the workpiece coordinate system is selected, it is generally not necessary to set the coordinate system with G92. If set with G92, the workpiece coordinate system 1 ~ 6 will be moved. Therefore, do not mix G92 with g54 ~ G59, unless the workpiece coordinate system 1 ~ 6 is to be moved.

Whether the relative position changes with the setting of workpiece coordinate system depends on the corresponding setting in the parameter.

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 current machine tool coordinate. Please use it carefully when there is soft limit. The workpiece coordinates are all 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 local coordinate system

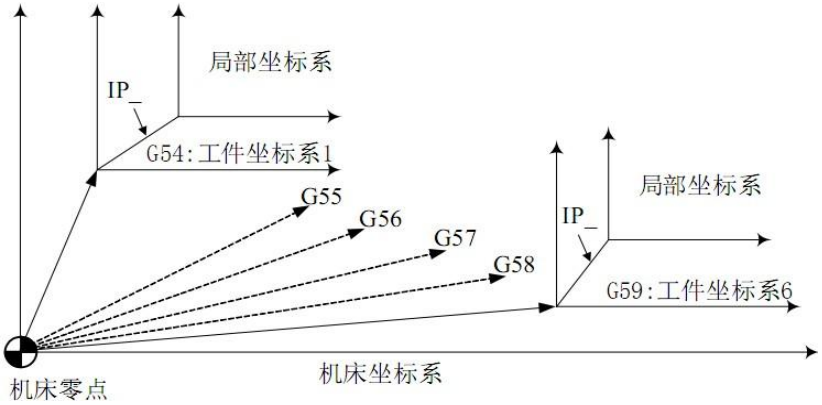
```
G52 IP_ ;Set the local coordinate system
...
G52 IP0; Cancel the local coordinate system
   IP__: Specifies the absolute coordinates of the origin of the local coordinate system in the
workpiece coordinate system.
```

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

G Code Opening action	Hole bottom action	Retraction action	Purpose	Use
G73	intermittent feed	——	Rapid feed	High speed deep hole machining cycle
G74	cutting feed	Spindle rotates forward	cutting feed	Counter Tapping Cycle
G80	——	——	——	Cancel a canned cycle
G81	cutting feed	——	Rapid feed	Drill, spot drill
G82	cutting feed	——	Rapid feed	Drilling and boring step holes
G83	intermittent feed	——	Rapid feed	Deep hole machining cycle
G84	cutting feed	Spindle reversal	cutting feed	Tapping cycle
G85	cutting feed	——	cutting feed	Boring
G86	cutting feed	Spindle stop	Spindle stop	Boring
G88	custom drilling	——	customize	drill
G89	cutting feed	——	cutting feed	Boring

Instruction format

```
G__ IP__ R__ Q__ P__ F__J__;
```

Address Description

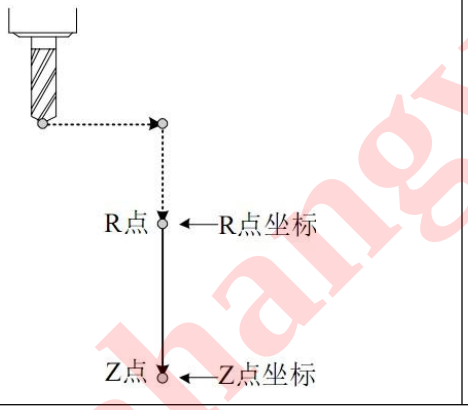
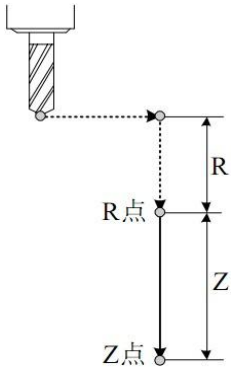
Specified content	address	explain
Hole processing method	G	Select canned cycle G73, G74, G80~G89
Hole machining direction		Fixed Z
Hole position data	Direction axis address of non hole machining in IP	The position of the hole is specified with absolute value or increment value. The control is the same as that of G00 positioning.
Hole machining	IP medium hole	As shown in the figure below, specify the distance from point R to the bottom of the hole with an incremental value or specify

address	processing direction address	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	as that specified by G04. Specifies the cutting feed rate. G74, G84 pitch designation.
	J	Machining axis specification, J0: X, J1: Y, J2: Z, J3: A, J4: B, J5: C, other values or do not specify the default Z axis

Instruction explanation

Absolute programming and relative programming

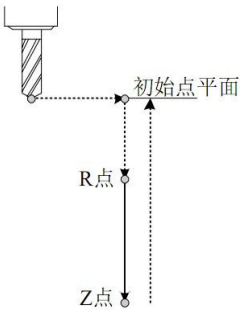
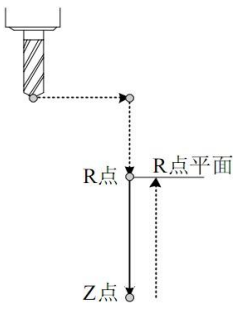
Use G90 and G91 to specify absolute programming and relative programming.

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

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)
	

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_ (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.

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.

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 G74 - tapping cycle

The G74 cycle is a left-hand tapping cycle, which is used to process the reverse thread.

Instruction format

G74 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_:Hole bottom pause time (unit: 0.001 s)
F_:Metric pitch.Value range: 0.001 ~ 500.00mm
J_:Machining shaft

Instruction interpretation

In this cycle, the left-hand tapping is performed, and the spindle enters in the reverse state. After reaching the hole bottom, the spindle pauses time p, and the spindle rotates forward to exit to complete the left-hand tapping action.

notes
In the 1 G74 reverse tapping cycle, the feed rate and feed hold are invalid.Even if the "feed" button is pressed, the action will not be stopped.
2 start spindle rotation before G74 is specified.If G74 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 performed 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.
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.

3.9.4 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.

notes

1 start spindle rotation before g81 is specified. If g81 and M code are specified in the same program segment, the M code will be sent out at the initial positioning, and the next loop action will be carried out after the M code execution is finished.

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.

example

```
M04 S800;                    Spindle rotation
G90 G99 G81 X100. Y100. Z-20. R50. F100.;   Positioning, 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 machining start point
M05 ;                        Spindle stop
```

3.9.5 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_;

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

F_:feed rate

J_:Machining shaft

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.

notes

1 start spindle rotation before g82 is specified. If g82 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

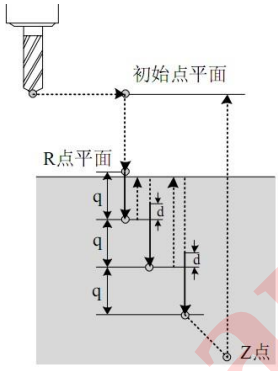
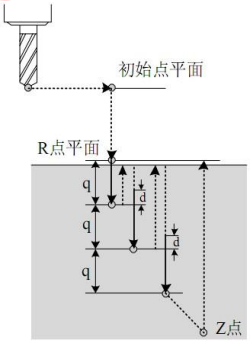
P of the instruction hole processing can not be stored as modal data.
4 in the fixed cycle mode, if the tool length offset has been instructed, the offset will be performed when the initial point plane is positioned.
5 tool offset command is invalid in fixed cycle.
6 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.6 G83 - deep hole machining cycle

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

G83(G98)	G83(G99)
	

Instruction explanation

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

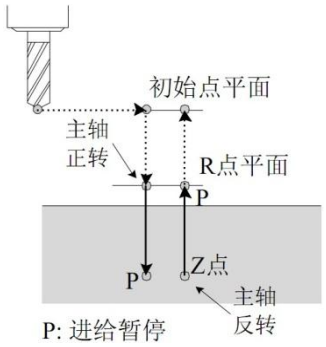
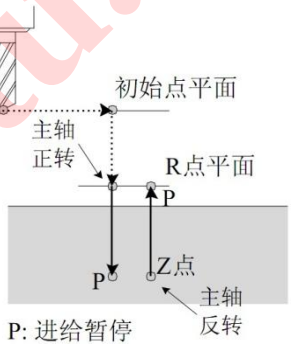
performed when the initial point plane is positioned.
5 tool offset command is invalid in fixed cycle.
6 the fixed cycle must be cancelled before changing the drilling axis or machining plane.

3.9.7 G84 - tapping cycle

G84 cycle is power wire cycle, which is used to process positive thread.

Instruction format

G84 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_:Hole bottom pause time (unit: 0.001 s)
F_:Metric pitch.Value range: 0.001 ~ 500.00mm
J_:Machining shaft

G84(G98)	G84(G99)
	

Instruction explanation

In this cycle, the tapping cycle is executed. When the spindle is in the forward rotation state, the spindle stops time p when it reaches the hole bottom, and the spindle reverses to exit to complete the tapping action.

notes
In 1 g84 tapping cycle, feed rate and feed hold are invalid.Even if the "feed hold" button is pressed, it will not stop before the end of the return action.
2 start spindle rotation before g84 is specified.If g84 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 performed 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.
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.

3.9.8 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

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.

3.9.9 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.

3.9.10 G88 - Custom drilling

G88 is used for custom drilling to achieve efficient drilling.

The specific realization in the drilling function G88 editing user-defined.

Instruction format

G88 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

3.9.11 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.

3.9.12 G80 - fixed cycle cancellation

G80 is used to cancel the fixed state.

Instruction format

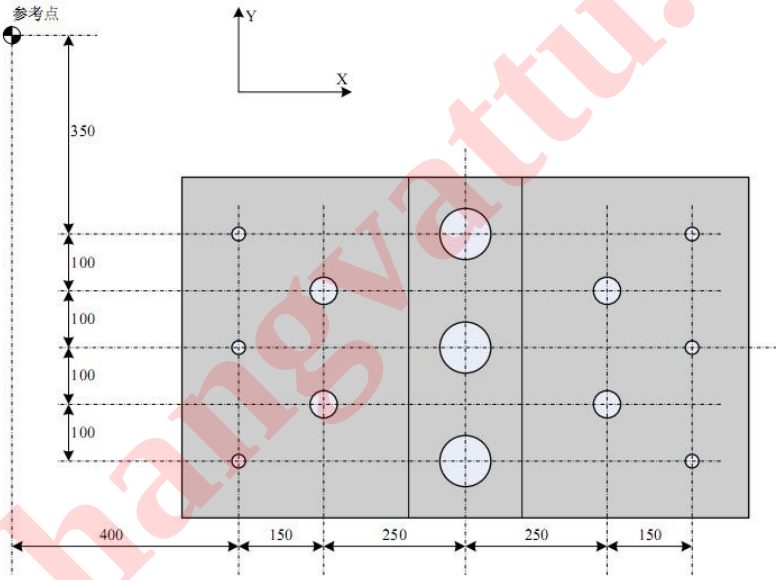
```
G80 ;
```

Instruction interpretation

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

3.10 drilling fixed cycle example (using tool length compensation)

Workpiece size data 1



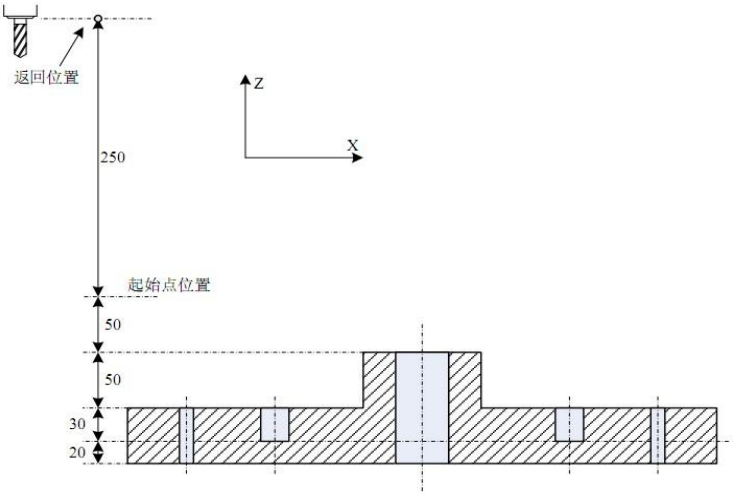
Workpiece size data 2

1 ~ 6Φ10 holes

#7 ~ 10 drillΦ20 holes

#11 ~ 13 drillΦ95 holes (50 mm deep)

#11～13　钻Φ95孔（深50mm）

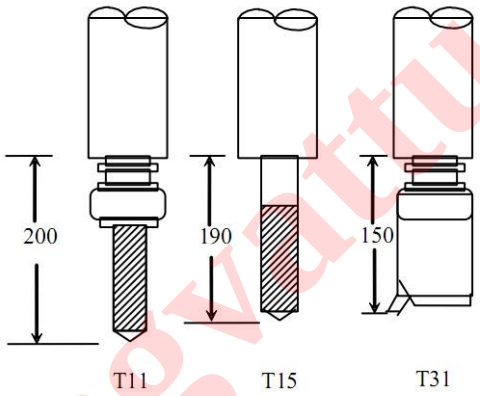


prop selection

The value of offset number 11 is 200.0

The value of offset number 15 is 190.0

The value of offset number 31 is 150.0



▲Machining program

```
N001 G92 X0 Y0 Z0 ;  The coordinate system is set at the reference point.
N002 G90 G00 Z250.0;
N003 G43Z0H11;       Plane tool length compensation is performed at the initial point.
N004 S1000 M3 ;      Spindle start
N005 G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120.0 ; After positioning, machine #1 hole.
N006 Y-550.0 ;       After positioning, machine #2 hole and return to the R point plane.
N007 G98 Y-750.0 ;   After positioning, process #3 hole and return to the initial point plane.
N008 G99 X1200.0 ;   After positioning, process the #4 hole and return to the R point plane.
N009 Y-550.0 ;       After positioning, process the #5 hole and return to the R point plane.
N010 G98 Y-350.0 ;   After positioning, process hole #6 and return to the initial point plane.
N011 G00 X0 Y0 M5 ;  Return to the reference point and the spindle stops.
N012 G49 Z250.0 ;
M00 ;                Pause, change tool
N013 G43 Z0 H15 ;    Initial point plane, tool length compensation.
N014 S20 M3 ;        Spindle start
N015 G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 P30 F70 ; After positioning, process the #7 hole and
                                                      return to the R point plane.
N016 G98 Y-650.0 ;                                    After positioning, process the #8 hole and
                                                      return to the initial point plane.
```

```
N017 G99 X1050.0 ;   After positioning, process the #9 hole and return to the R point plane.
N018 G98 Y-450.0 ;   After positioning, process the #10 hole and return to the initial point
                      plane.
N019 G00X0 Y0 M5 ;   Return to the reference point and the spindle stops.
N020 G49 Z250.0 ;
M00 ;                Pause, change tool
N021 G43 Z0 H31 ;    Initial point plane tool length compensation.
N022 S10 M3 ;
N023 G85 G99 X800.0 Y-350.0 Z-153.0 R47.0 F50 ;      After positioning, process hole #11 and return
                                                     to the R point plane.
N024 G91 Y-200.0 ; Y-200.0 ; After positioning, process holes #12 and #13, and return to the
                             R point plane.
N025 G00 G90 X0 Y0 M5 ;  Return to the reference point and the spindle stops.
N026 G49 Z0 ;        Cancel tool length compensation.
N027 M05 ;           Spindle stopped.
N028 M30 ;           Program stop
```

3.11 G22-G23 cycle execution

This instruction pair can realize program loop.

G22L3

..

..

G23

The intermediate program loop is executed 3 times

It can also be nested, but no more than 4 levels.

G22L3

G22L5

..

..

G23

G23

3.12 G31-Jump function

After G31, linear interpolation like G01 can be performed by command axis movement. During the execution of this instruction, if a jump signal is input, the program segment stops the rest and starts to execute the next segment. This function is mainly used to control the end of processing by external signal, or to measure the size of workpiece.

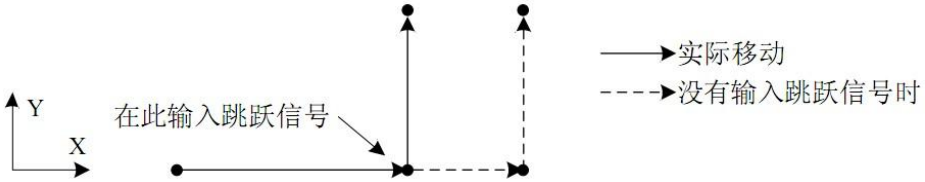
Instruction format

```
G31 IP__ F__ ;
  IP_:Specifies the coordinates of the end of the move.
  F_ :Specifies the feed rate.
```

Give an example

The program segment after G31 is incremental instruction

From the position where the jump signal is interrupted, it moves with increment value. At the same time, the x-axis absolute coordinates of the jump point are saved to the macro variable ○.




```
……
G91 G31 X100.0 F100. 0 ;
G04 ;
#1 = #5041 ;
Y50.0 ;
……
```

3.13 G50-G51 Positioning Movement

Position measuring motion is a supplement to G31, which can support more input ports. The motion mode of each axis is consistent with that of G1.

Instruction format

```
G50/G51 IP_ P_ ;
IP: Move axis
  P: Input port.
```

For example:

G50 X10 Z100 P2 ; During the movement, if input port 2 is invalid, the movement is interrupted and the next line is executed

G51 Y10 Z50 P20 ; During the movement, if input port 20 becomes valid, the movement is interrupted and the next line is executed

Chapter 4 auxiliary functions (M code)

If a 2-digit value is specified 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 reverse
M15	Spindle 2 stops
M30	Program ends, program returns to start
M62	Start spindle speed monitoring (encoder required). Example: M62 S1000, if the spindle is turned on and the speed is lower than 1000 rpm, the program will stop and the encoder needs to be supported. Here S is not used to set the speed
M63	Cancel speed monitoring
M64	Count plus one
M65	count clear
M70	Waiting for input port, output port or auxiliary relay 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	nput 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	Output port or auxiliary relay close example: M80 Y12
M81	Example of output port or auxiliary relay: 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 is 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 M00 - program pause

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.

4.2.12 M62 - speed monitoring

Instruction format

M62 S_

Command function

When the program is running, the encoder speed is monitored in real time. If it is lower than the monitoring value, the system will alarm and the program will stop. This function requires encoder support.

The program is closed by default when it starts.

4.2.13 M63 - cancel speed monitoring

Instruction format

M63

Command function

Cancel speed monitoring

4.2.14 M64 counter plus one

Instruction format

M64

Command function

Add one to the workpiece count.

4.2.15 M65 - counter clear

Instruction format

M65

Command function

The workpiece count value is cleared.

4.2.16 M70- Wait for the input port, output port, 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.

4.2.17 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.

4.2.18 M74 - waiting for input port, output port, falling edge of auxiliary relay

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.

4.2.19 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.

4.2.20 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.

4.2.21 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.

4.2.22 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.

4.2.23 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.

4.2.24 M84 - output port, auxiliary relay output is 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.

4.2.25 M98 / M99 - subroutine call and subroutine return

Instruction format

M98 P####Ln;

M99;

Command function

1. P: subroutine calls the characteristic character, which cannot be omitted.
2. Subroutine name must be four digits.
3. Ln: the number of subroutine calls, which is called once when omitted, up to 99999 times.

When there is a fixed program in the program and it appears repeatedly, it can be used as a subroutine. In this way, every place where the fixed program needs to be used can be executed by calling the subroutine, instead of having to write it repeatedly.

The last segment of the subroutine must be the subroutine return instruction, namely M99. After executing the M99 instruction, the program returns to the main program, and the next program calling the main program instruction continues to execute.

Give an example

Main program o001

```
Main program 0001
N0010 M03 S1000 ;
……
N0080 G0 X10 ;
N0090 M98 P0005 ;
N0100 G0X30 ;
……
N0150 M30 ;

Subroutine 00005
N0010 G01 X10 F100 ;
……
N0060 G0 Z30 ;
N0070 M99; subroutine return

Execute the 00001 main program, the execution flow is:
N0010 M03 S1000 ;
……
N0080 G0 X10 ;
  N0010 G01 X10 F100 ;
  ……
  N0060 G0 Z30 ;
N0100 G0X30 ;
……
```

```
N0150 M30 ;
```

Chapter 5 Prop compensation function (H code)

5.1 tool compensation

The difference between the assumed tool length value during programming and the tool length value used in actual machining is set into the offset memory. When machining a workpiece, it is not necessary to modify the program, only need to specify the corresponding length compensation value, then the same workpiece can be processed with different length cutters. This is the tool length compensation function.

Each axis can be specified with independent cutter compensation code HX, hy, Hz, ha, Hb, HC.

In order to get the correct compensation, it is necessary to pay attention to the compensation direction when setting the tool compensation again. For example, under the z-axis tool tip and the re axis, the whole direction compensation is needed. If the tool arrangement is carried out, the negative direction is generally used.

Z axis cutter compensation Hz will automatically open G43, other axes have no effect.

5.2 Z axis tool length compensation (G43, G44, G49)

(G43, G44, G49) only affected Z axis, other axes had no effect

Instruction format

```
G43  H_ ;
G44  H_ ;
  G43:
  G44: negative bias
  H: Offset number
```

Instruction description

Offset direction

When G43 is specified, the end coordinate value specified by z-axis movement command in the program plus the length compensation value specified by H code (in the offset memory) will be used as the end coordinate value;When g44 is specified, the end coordinate value specified by z-axis movement command in the program is subtracted from the length compensation value specified by H code, and the calculation result is taken as the end coordinate value.

When z-axis movement is omitted, only the value of tool length compensation is moved. When the offset is negative, the direction of movement is opposite.

G43 and g44 are modal g codes, which are valid until encountering other G codes in the same group.

Offset number

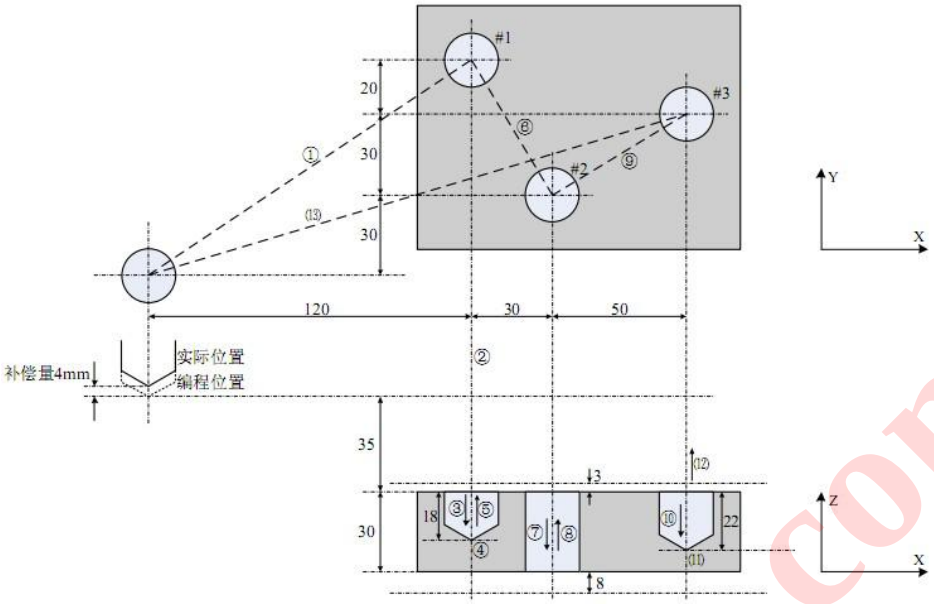
The offset number can be specified as hn00-hn99. In the LCD / MDI panel, the offset corresponding to the offset number hn01-hn99 can be set in the offset memory in advance. The tool length compensation value corresponding to the offset number hn00 is always 0 and cannot be set. (n:X,Y,Z,A,B,C)

999, the allowable range of tool length is [. 999-999].

```
fill
When changing the offset number and changing the offset amount, it only becomes a new offset
amount, instead of adding the new offset amount to the old compensation amount.
For example:
HZ01 offset 20.0
HX02 offset -30.0
HX02  偏置量-30.0
G90 HZ01 Z100.0    ;  Z 走到 120.0
G90 HX01 X100.0    ;
```

Cancel tool length compensation with Hn00.

give an example

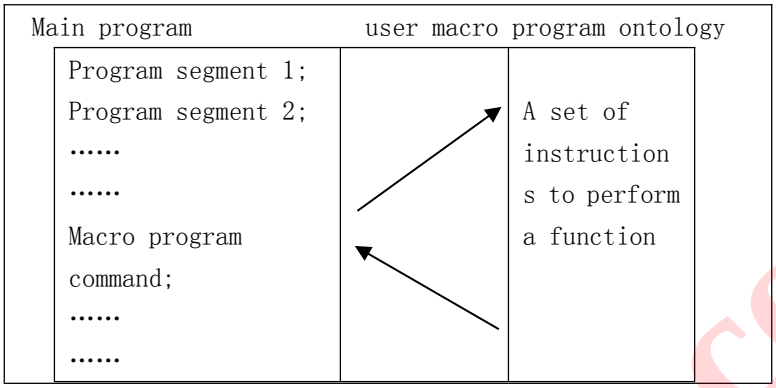



```
Processing program (HZ1=-4.0mm)
N1 G91 G00 X120.0 Y80.0 ;      (1)
N2 Z-32.0 HZ1 ;            (2)
N3 G01 Z-21.0 F1000 ;          (3)
N4 G04 P2000 ;                 (4)
N5 G00 Z21.0 ;                 (5)
N6 X30.0 Y-50.0 ;              (6)
N7 G01 Z-41.0 ;                (7)
N8 G00 Z41.0 ;                 (8)
N9 X50.0 Y30.0 ;               (9)
N10 G01 Z-25.0 ;              (10)
N11 G04 P2000 ;               (11)
N12 G00 Z57.0 H0 ;            (12)
N13 X-200.0 Y-60.0 ;          (13)
N14 M02 ;                     (14)
```

Chapter 6 User Macro Program

6.1 definition

Macro program allows users to use basic program language features such as variables, arithmetic operation, logical operation, bit operation, conditional transfer, loop control and program call, which makes programming more convenient, flexible, easy and quick. It can greatly improve the universality of the program, and the same subprogram can be called to process the same type of parts only by assigning values to different main programs.



6.2 variables

Instruction format

#i ;

Example: #1,#[#1+#2-12]

Instruction description

(1) Types of variables: variables can be divided into four types according to variable numbers.

Variable number	Variable type	function
#0	empty variable	The variable is always empty, and no value can be assigned to it.
#1～#99	local variable	Local variables can only be used in macro programs to store data, for example, operation results. When power is cut off, local variables are initialized to null. When macro programs are called, independent variables assign values to local variables.
#100～#199 #500～#599	public variable	Common variables have the same meaning in different macro programs. When the power is cut off, variables #100~#199 are initialized to null, and the values of variables #500~#599 are saved, even if the power is cut off.
#1000～	system variable	Used to read and write various data of CNC runtime.

(2) The reference of variables is to specify the address followed by the variable number in order to use the variable value in the program. When variables are specified by expressions, put the expressions in brackets. E.g. G01 X[#1+#2] F#3; G00 X-#1。

Note 1: Addresses O, G and N cannot refer to variables. For example, O#100, N#120 is illegally quoted;

2: If it exceeds the maximum code value specified by the address, it cannot be used; Example: When #130 = 120, M#230 exceeds the maximum code value.

(3) Empty variable. When the variable value is undefined, this variable is empty. Variable #0 is always empty. It can't be written, it can only be read.

▲ When referring to an undefined variable (null variable), the address itself is also ignored.

When #1= < empty >	When #1=0
G00 X100 Z#1 ↓ G00 X100	G00 X100 Z#1 ↓ G00 X100 Z0

▲ Operation. Except for assignment with <empty variable>, <empty variable> is the same as "0" in other cases.

When #1= < empty >	When #1=0
#2=#1 ↓ #2=<空>	#2=#1 ↓ #2=0
#2=#1 * 5 ↓ #2=0	#2=#1 * 5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

▲ Conditional expressions, <null> in EQ and NE are different from "0"

When #1= < empty >	When #1=0
#1 EQ #0 ↓ established	#1 EQ #0 ↓ invalid
#1 NE #0 ↓ invalid	#1 NE #0 ↓ invalid
#1 GE #0 ↓ established	#1 GE #0 ↓ invalid
#1 GT #0 ↓ invalid	#1 GT #0 ↓ invalid

(4) Display of variable values; When the variable is blank, the variable is empty; When the variable is displayed as "* * * *", it means that the variable value overflows.

6.3 System variables

System variables are used to read and write CNC internal data, such as input port, output port, tool offset value and current coordinates, but some system variables can only be read.

Description:

6.3.1 macro variables of interface signal system

CNC defines 96 macro variables of input signal system and 96 macro variables of output signal system. They are #1001~#1096 macro input ports and #1101~#1196 macro output ports respectively.

Assigning values to output macro variables #1101~#1196 can change the output signal states of Y01~Y96; When the value is "1", the output signal is turned on; When the value is "0", its output signal is turned off. But it is invalid when that output port is not universal.

Check the values of macro variables #1001~#1096, and check the input status of input interfaces X01~X96.

Correspondence table of macro variables of input signal system:

Macro variable number

Macro variable number	Macro variable function	Read write function
#1001~#1096	input port	read-only
#1101~#1196	output port	Reading and writing
#1201~#1296	Auxiliary relay	Reading and writing
#1301~#1312	Input port 8bit read, #1=#1301 read X01~X08 once, #1=#1302 read X09~X016 once...	read-only
#1401~#1412	Output port 8bit read and write, #1401=0, one-time Y01~Y08 clear	Reading and writing
#1501~#1512	Auxiliary relay 8bit read and write, #1501=0, one-time Z01~Z08 reset	Reading and writing

6.3.2 macro variables of tool compensation system

Macro variable number	Macro variable function	Read write function
#2001~#2006	Read the current tool compensation number of each axis (XYZABC)	read-only
#5081~#5086	Read the value of the current tool compensation number of each axis (XYZABC)	Read and write (No.0 cutter compensation read only)

6.3.3 other system variables

Macro variable number	Macro variable function	Read write function
#3091	Workpiece counter	Reading and writing
#4001~#4007	1~7 groups of G codes	read-only
#5041~#5044	Absolute coordinates of each axis	read-only
#5061~#5064	Machine coordinate of each axis	read-only

6.4 arithmetic and logic operations

The operations listed in the following table can be performed in variables. The expression to the right of the operator can contain a constant or a variable consisting of a function or operator. We can use the constants of ○ɔ and ○ɔ. The variable on the left can also be assigned an expression.

function

function	format	remarks
assign	#i=#j ;	assignment operator
addition	#i=#j + #k ;	If J = = I, the simplified symbol (+ =, -

subtraction multiplication division	#i=#j - #k ; #i=#j * #k ; #i=#j / #k ;	=, * =, / =) can be used. If ○ɔ I = ○ɔ I + ○ɔ K, it can be simplified as ○ɔ I + = ○ɔ K.
And Exclusive or Or Shift left Shift right	#i=#j & #k;Or ○ɔ #I= ○ɔ#Jand ○ɔ #K; #i=#j ^ #k ;Or #I = ○ɔ#J XOR #K; #i=#j | #k ;Or#I = #J OR#K; #i=#j << #k ; #i=#j >> #k ;	bit manipulation. This operation will cast a floating point number to an integer for operation. Bit operations operate on the binary form of integer numbers. If j==i, then shorthand notation (&=, ^=, |=, «=, »=) can be used. For example, #i = #i & #k can be simplified to #i &= #k.
N equal not equal to more than the greater or equal to less than less than or equal to	#i=#j == #k ;OR #i=#j EQ #k ; #i=#j != #k ;OR #i=#j NE #k ; #i=#j > #k ;OR #i=#j GT #k ; #i=#j >= #k ;OR #i=#j GE #k ; #i=#j < #k ;OR #i =#j LT #k ; #i=#j <= #k ;OR #i=#j LE #k ;	relational operations. The result is a 32-bit unsigned integer 0 (FALSE) or 1 (TRUE).
absolute value rounding Round up Rounding down Natural logarithm exponential function	#i=SQRT[#j]; #i=ABS[#j]; #i=FABS[#j]; #i=ROUND[#j]; #i=FUP[#j];或 #i =CEIL[#j]; #i=LN[#j];Or ○ɔ I = log [○ɔ J]; #i=LN[#j];或 #i =LOG[#j]; #i=EXP[#j];	
Arcsine cosine Cosine inverse tangent Arctangent	#i=SIN[#j]; #i=ASIN[#j]; #i=COS[#j]; #i=ACOS[#j]; #i=TAN[#j]; #i=ATAN[#j]	Trigonometric functions. When specified in degrees, such as 90° 30' table shown as 90.5 degrees. A constant or expression can be used in place of #j.

Description:

(1) angle unit: the angle unit of functions sin, cos, asin, ACOS, tan and atan is degree (°).For example, 90 ° 30 ′ should be expressed as 90.5 ° (degree).

⑵ARCSIN # i=ASIN[#j]

When #j is outside the range of -1 to 1, an alarm is issued.

Constants can be substituted for variable #j.

⑶ARCCOS # i =ACOS[#j]

The resulting output range is from 180° to 0° .

Raises an alarm when approximately outside the range of -1 to 1.

Constants can be substituted for variable #j.

(4) natural logarithm ○ɔ I = ln [#j]

When the opposition number (○ɔj) is 0 or less than 0, the alarm will be given.

The constant can replace the variable ○ɔ J.

(5) exponential function ○J I = exp [# J];The constant can replace the variable ○J J.

(6) round round round function

When the arithmetic or logic operation code if or while contains the round function, the round function is rounded to the first decimal place.

For example: when executing ○J 1 = round [○J 2], where ○J 2 = 1.2345, the value of variable 1 is 1.0.

(7) when the integer sum is greater than the original integer value, it is called absolute operation;If it is less than the absolute value of the original number, it is called rounding down.We should be careful when dealing with negative numbers.

(8) divisor: when the divisor of 0 is specified in division or tan [90], the system will alarm.

6.5 transfer and circulation

In the program, goto statement and if statement can be used to change the flow direction of control.There are three transfer and loop operations available.

oto statement (unconditional transfer).

If statement (conditional transfer: if. Then.).

while statement (loops when.).

6.5.1 unconditional transfer (goto statement)

Transfer to the block marked with sequence number n.When a sequence number other than 1 to 99999 is specified, an expression can be used to specify the sequence number.

Instruction format

```
GOTOn;
  N:顺序号(1~99999)
```

give an example

```
GOTO1;
GOTO# 10;
```

6.5.2 Conditional Control (IF Statement)

Command format 1

If [< conditional expression >] goton;

If the specified conditional expression is true, it is transferred to the program segment with sequence number n;If the specified conditional expression does not hold, the next segment is executed.

give an example

If the value of variable ○J 1 is greater than 10, it is transferred to the block of sequence number N2.

Instruction format 2

If [conditional expression] then < macro program statement >;

If the conditional expression is satisfied, only one macro program statement can be executed by executing the statement after then.

give an example

IF[#1 EQ #2] THEN #3=0;

If the value of #1 is equal to the value of #2, assign 0 to variable #3; if not, the sequence goes down without executing the assignment statement after THEN.

Instruction description

You must use conditional expression, conditional expression, or conditional expression.

Conditional operators are shown in the following table.

operator	meaning
EQ	equals (=)
NE	not equal to (≠)
GT	greater than (>)
GE	Greater than or equal to (≥)
LT	less than (<)
LE	Less than or equal to (≤)

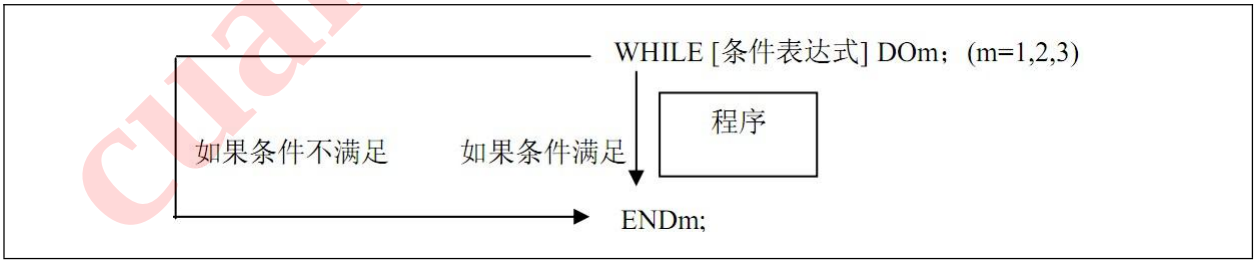
Example: The following program calculates the sum of the integers 1 to 10.

```
O9600
#1=0;                    Store and count the initial value of the variable
#2=1;                    initial value of the summand variable
N1 IF[#2 GT 10]GOTO2;    branch to N2 when the summand is greater than 10
#1= #1+#2;               calculation and number
#2= #2+1 ;               next summand
GOTO1;                   go to N1
N2 M30;                  End of procedure
```

6.5.3 Loops (WHILE Statement)

A conditional expression is specified after while. When the specified condition is true, the program segment from do to end is executed;Otherwise, jump to the segment after end.

Instruction format 1




```
While [conditional expression] do m;
WHILE[条件表达式]DO m;
……
END m ;
```

Instruction description

If the result of the expression is not 0, the result is considered to be true, and the statement between while and end is executed in a loop until the expression result is false. When the result of

the expression is zero, it jumps to the next segment of the end statement for execution. If the expression result is not an integer, it should be cast to an integer, otherwise the alarm will be given. M is a positive integer just to match the do and end statements.

give an example

```
……
N1 #1 = 1 ;
N2 #2 = 0 ;
N3 WHILE[#1 <= 100] DO 5 ;
N4 #2 += #1 ;                    (Calculate 1+2+3+…+100)
N5 #1 += 1 ;
N6 END 5;
……
When the program executes, #1 adds from 1 to 100, and #2 gets the sum of 1+2+3+...+100.
```

Command format 2

```
DO m ;
……
END m ;
    m: Specifies the label of the loop execution range (1 to 1023).
```

Instruction description

When the while judgment condition statement is not specified in the do statement, it is an infinite loop.

For example, if the while judgment condition is not specified in the routine in format 1, the values of ○3 1 and ○3 2 will be added infinitely until the data overflow alarm.

give an example

```
……
N1 #1 = 1 ;
N2 #2 = 0 ;
N3 DO 5;
N4 #2 += #1 ;          (Calculate 1+2+3+…+100)
N5 #1 += 1 ;
N6 END 5 ;
……
```

Label m and loop nesting

The label M can be reused, and loops can also be nested (up to 8 nesting levels), with the following limitations

① Do m and end m must be used in pairs (the m value is the same), and do must appear before end.

② The two loops cannot cross.

③ If the loop is nested, the child loop cannot be labeled the same as the parent loop.

④ Goto statement can be transferred from in vitro circulation to extracorporeal circulation, but cannot be transferred from extracorporeal circulation to extracorporeal circulation; otherwise, an alarm will be given when the end statement is executed.

Chapter 7 integrated routines

7.1 grinder routine

This routine can compensate the specific distance of grinding wheel after every n machining

Define panel parameters first

501 compensation processing times

502 compensation length

Record the processing times with ◌ɔ 500

Assuming that the grinding wheel is in the z-axis direction, the cutter compensation should be added before tool setting

Set Hz1 as the grinding wheel radius during tool setting

```
Procedure o0001NC
G90HZ1 ;            Offset 1 with z-axis tool
M03S1000 ;          Open spindle 1
IF[#500<#501] GOTO 10 ;       If the processing times are decimal, the property compensation is ignored
#500=0 ;            Clear processing technology
#5083-=#502 ;       Z axis cutter compensation minus each compensation
N10 Z0 ;            Z axis to machining position
G1X100F100 ;        Start machining of X-axis
X10
G0Z10;              Z axis lift
#500+=1 ;           Processing count plus one
M05 ;               Stop spindle
M30
```

For each specified number of machining operations, tool compensation decreases by the specified distance.

7.2 using macro operation to realize tooth division without accumulated error

Define panel parameters first

501 number of gears.Divided gear axis X axis, processing axis Z axis

```
G90G54
M03S100  ;    open spindle
G93X0 ;       X mechanical coordinate setting 0
#1=0 ;        Clear the teeth counting
N10 G0Z0
G1Z-10F200
Z0
G0 Z10;              One tooth is machined on the Z axis above
#1+=1 ;              Counter plus one
#2=#1*360.0/#501 ;   Calculate the current tooth position
G0X#2 ;              X goes to the current tooth position
IF[#1>#501] GOTO 10 ;If the number does not reach the last tooth, return to N10 to continue
M05 ;                Turn off the spindle
M30 ;                End of procedure
```

7.3 punch and feed routine

In this routine, the single axis controller is used to realize loading, processing and blanking at the same time to improve the processing efficiency

Set parameter p0010 1
Set parameter p0011 0
Set parameter p00120
Set parameter p0013 1

Set parameter p0014 0
Set parameter p0015 0

Main program 00001.NC machining program
Top dead center of X1 punch
Y1 signal to punch
Y2 clamping

```
G90
G0Z0
M80 Z2;        Clear Z2 auxiliary relay
M71 Z1 ;       Wait for the feed to complete the auxiliary relay
M81 Y2 ;       intensify
G4100 ;        Delay 0.1 second
M81 Z2 ;       Set a Z2 auxiliary relay
G91
G22L10;        Execute the loop 10 times
Z10 ;          Feeding 10 mm
M82Y1P500 ;    Signal the punch for 0.5 seconds
M71X1 ;        Waiting for top dead center signal
G23 ;          Loop execution
M80Y2 ;        release
G4P500 ;       delayed
M81Z3 ;        Notice of completion of cutting procedure for relay
G90Z0 ;        Back to zero
M99
```

Sub-A program 00002.NC feeding program
X2 suction test
Y3 suction
Y4 upper and lower cylinders
Y5 left and right cylinder

```
M81Y4 ;  Under cylinder
M81Y3 ;  Open suction
M71X2 ;  Chat detected
M80Y4 ;  On the cylinder
G4P500 ; Delay 0.5 seconds
M81Y5 ;  Cylinder forward
```

```
M70Z2 ;  Process feed notification
M81Y4 ;  Under cylinder
G4P500 ;  0.5 seconds
M80Y3 ;  Suction loose
G4P100 ;  0.1 second
M80Y4 ;  On the cylinder
M81Z1 ;  Feeding finished
M71Z2 ;  The waiting is tightening up
M80Z1 ;  Cleaning and feeding auxiliary relay
M80Y5 ;  Cylinder retraction
G4P500 ;  Delay 0.5 seconds
M99 ;    Loop execution

Sub B program o0003NC cutting program
X3 suction test
Y6 suction
Y7 upper and lower cylinders
Y8 left and right cylinder

M71Z3 ;  Waiting for cutting notice
M80Z3 ;  Blanking notice
M81Y7 ;  Under cylinder
M81Y6 ;  Suction
M71X3 ;  The signal is detected
M80Y7 ;  On the cylinder
G4P500 ;  0.5 seconds
M81Y8 ;  Cylinder forward
G4P500 ;  Delay 0.5 seconds
M81Y7 ;  Under cylinder
G4P500 ;  0.5 seconds
M80Y6 ;  Suction loose
G4P100 ;  0.1 second
M80Y7 ;  On the cylinder
G4P500 ;  0.5 seconds
M80Y8 ;  Cylinder retraction
G4P500 ;  0.5 seconds
M99 ;    Loop execution
```

7.4 three axis circle equal division drilling

3-axis circle bisection drilling

Define panel parameters first

501

502 circle radius

The zero point of the workpiece is in the center of the circle,

```
Main program o0001NC
G90G54G98HZ1
M03S1000
G0Z10
#10=0;     Current number of holes
G22L#501 ; Number of cycles
#2=COS[#1]*#502 ; Current X coordinate
#3=SIN[#1]*#502 ; Current y coordinate
G73 X[#2]Y[#3]Z-20R0Q5J2F500 ; Deep hole machining instruction G73
#10+=1;
#1=#10*360.0/#501 ;Calculate the next hole angle
G23 ;              End of cycle
M05
M30
```

7.5 Three-axis rectangular array punching

Define panel parameters first

501 x number of holes

502 number of Y holes

503 x spacing

504 y spacing

505 hole depth

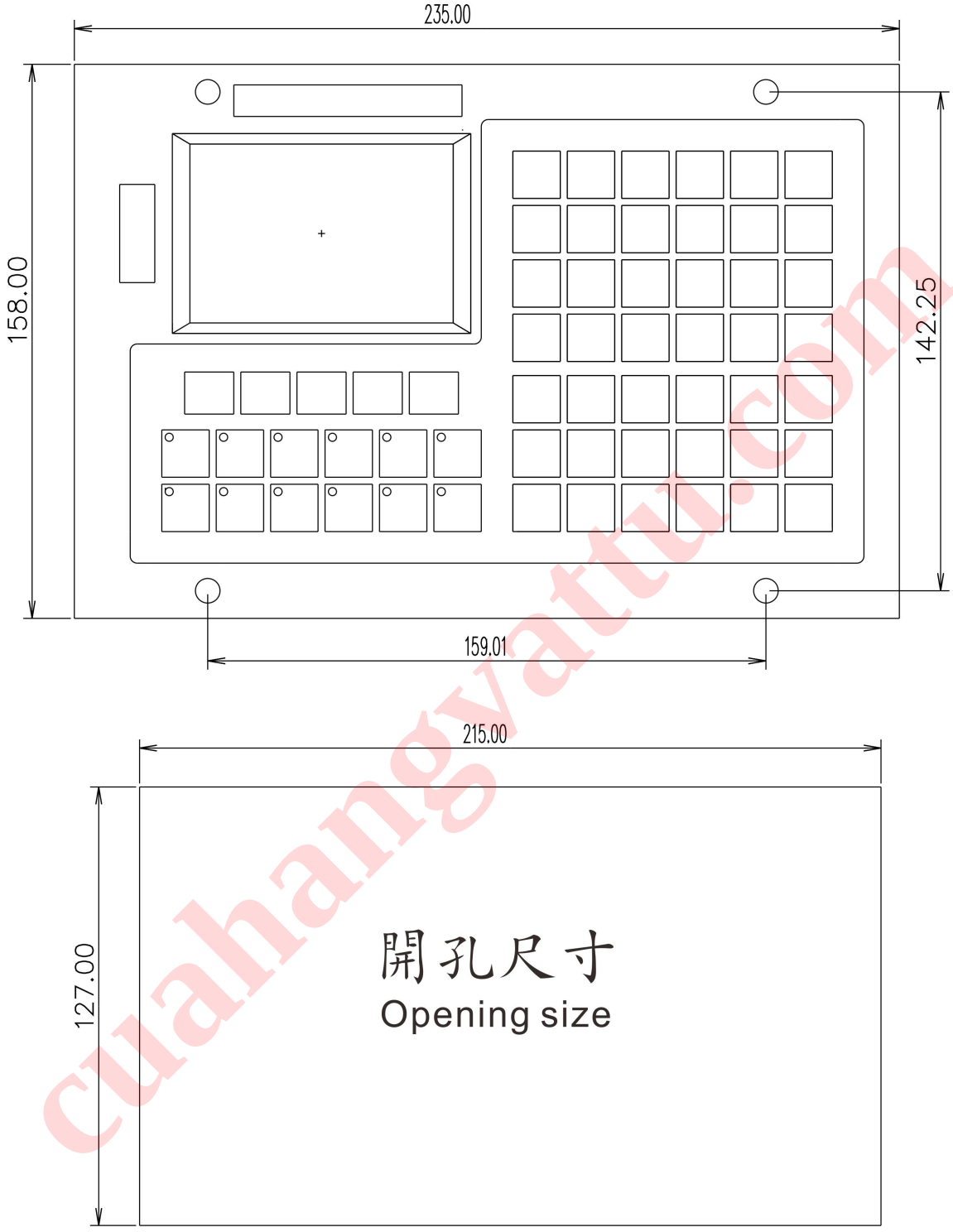
Drilling sequence, in line zigzag

```
Main program o0001NC
G90G54G98HZ1
M03S1000
G0Z10
#2=#501-1          ;Line cycle times, the first hole does not need to be cycled
#3=#502-1          ;Number of column cycles. The first column does not need to be cycled
#4=#505            ;X-axis spacing
G0X0Y0             ;To the first point
G73Z[#505]R0Q5J2F500 ;Fixed drilling instruction
G91                ;Click increment below
G22L#3             ;Column loop
G22L#2             ;Line loop
X[#4]              ;X into a space
G23                ;End of row loop
#4*=-1             ;The X spacing is reversed, and the next line is in the opposite direction
Y[#504]            ;Y into a space
G23                ;End of column loop
M05                ;Turn off the spindle
M30
```

PART 4 Installation and Commissioning

Chapter 1 installation layout

1.1 Dimensions

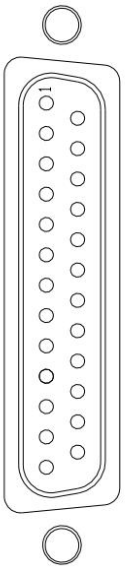


Note: The above dimensions are in mm.

Chapter 2 interface signal definition and connection

2.1 driver interface

2.1.1 driver interface definition

Interface form	Pin	Signal name	Function description
 (drive) (驱动器)	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+
	16	ADIR-	A-axis command direction signal-
	17	BCP+	B-axis command pulse signal+
	18	BCP-	B-axis command pulse signal-
	19	BDIR+	B-axis command direction signal+
	20	BDIR-	B-axis command direction signal-
	21	CCP+	C-axis command pulse signal+
	22	CCP-	C-axis command pulse signal-
	23	CDIR+	C-axis command direction signal+
	24	CDIR-	C-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:

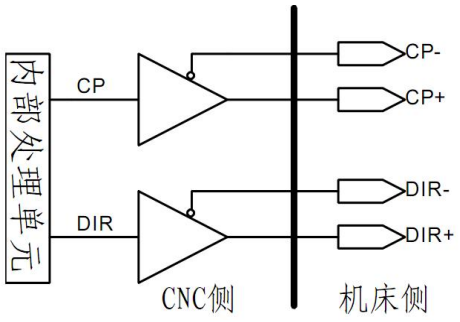
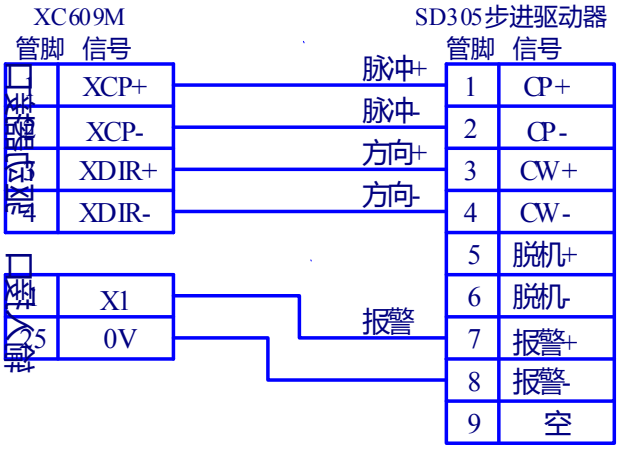


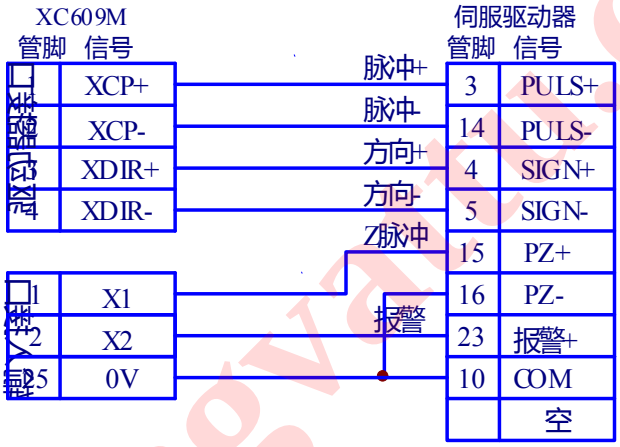
Figure 2.1.5 Internal circuit of command pulse signal and command direction signal

2.1.3 connection diagram with driving unit

2.1.3.1 Connection with stepper driver

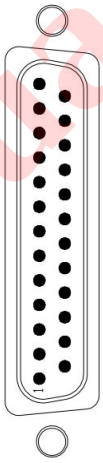


2.1.3.2 Connection with Servo Drive



2.2 Spindle and other interfaces

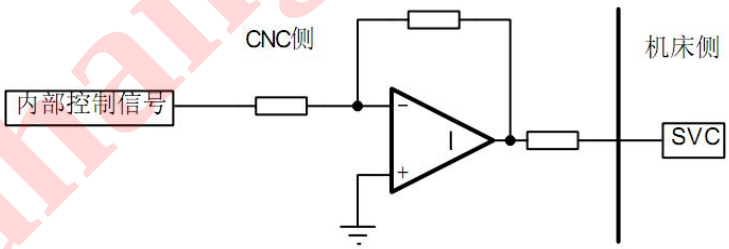
2.2.1 definition of spindle and other interfaces

Interface form	Pin	Signal name	Function description
 (spindle and others) （主轴及其它）	1	X13	Input 13 (optional function, please refer to the table below for setting)
	2	BMA-	Encoder A-phase pulse input-
	3	BMA+	Encoder A-phase pulse input+
	4	BMB-	Encoder B-phase pulse input-
	5	BMB+	Encoder B-phase pulse input+
	6	BMZ-	Encoder Z-phase pulse input-
	7	BMZ+	Encoder Z-phase pulse input+
	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

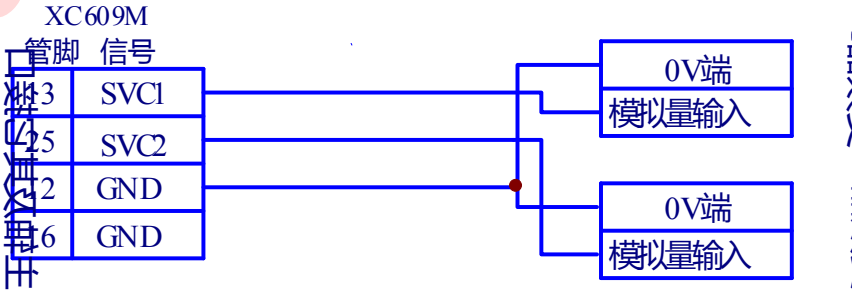
	14	485A	RS485 communication interface
	15	485B	RS485 communication interface
	16	GND	power ground
	17	X22	Input 22 (optional function, please refer to the following table for setting)
	18	X21	Input 21 (optional function, please refer to the following table for setting)
	19	X20	Input 20 (optional function, please refer to the following table for setting)
	20	X19	I Input 19 (optional function, please refer to the following table for setting)
	21	X18	Input 18 (optional function, please refer to the following table for setting)
	22	X17	Input 17 (optional function, please refer to the following table for setting)
	23	X16	Input 16 (optional function, please refer to the following table for setting)
	24	X15	Input 15 (optional function, please refer to the following table for setting)
	25	SVC2	Analog voltage 2. 0~10V analog signal output, connect to the analog voltage input terminal of the inverter

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:



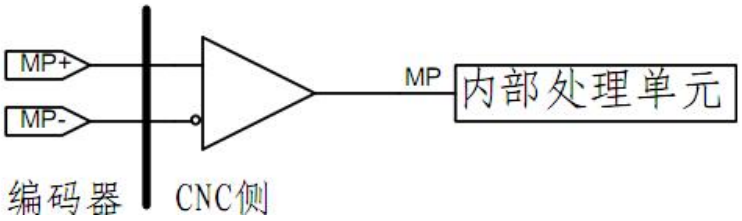
2.2.3 Description of the connection between the analog spindle and the inverter



2.2.4 Principle of encoder interface

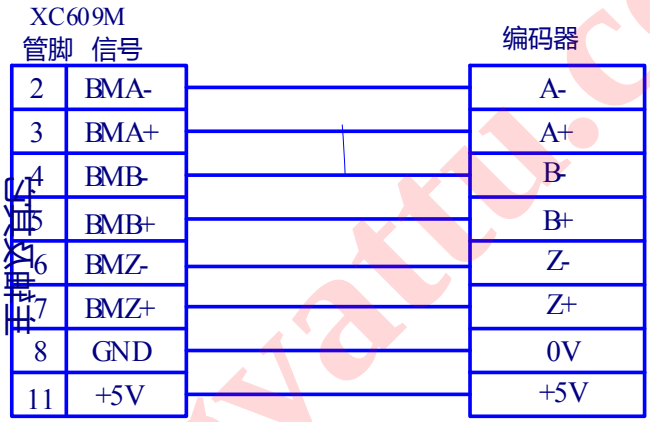
MPA + / MPA -, MPB + / MPB -, MPZ + / MPZ - are the differential input signals of a phase, B phase and Z phase of encoder respectively, which are received by 26ls32;B. Phase a signal is quadrature signal, and the highest signal frequency is less than 1MHz;The number of encoder lines used by the system is set in the parameter.

The internal connection circuit is as follows:

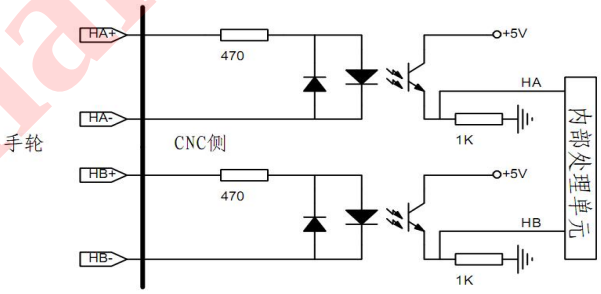


2.2.5 Encoder connection instructions

The connection between the system and the spindle encoder is shown in the figure below. Twisted pair is used for connection.

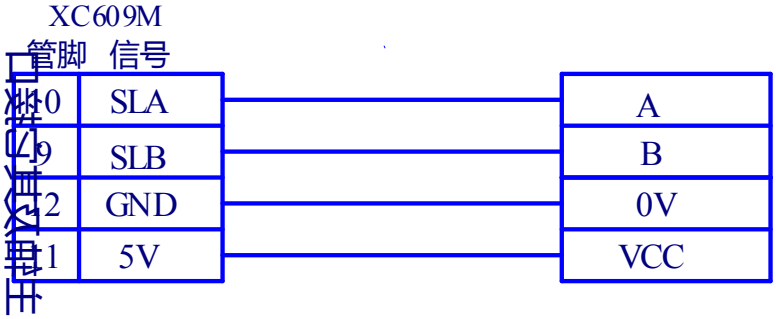


2.2.5 Principle of handwheel interface



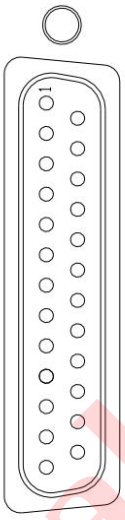
2.2.6 Handwheel connection instructions

XC609M 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
 (input) （输入）	1	X01	Input 1 (optional function, please refer to the table below for setting)
	2	X02	Input 2 (optional function, please refer to the table below for setting)
	3	X03	Input 3 (optional function, please refer to the table below for setting)
	4	X04	Input 4 (optional function, please refer to the table below for setting)
	5	X05	Input 5 (optional function, please refer to the table below for setting)
	6	X06	Input 6 (optional function, please refer to the table below for setting)
	7	X07	Input 7 (optional function, please refer to the table below for setting)
	8	X08	Input 8 (optional function, please refer to the table below for setting)
	9	X09	Input 9 (optional function, please refer to the table below for setting)
	10	X10	Input 10 (optional function, please refer to the table below for setting)
	11	X11	Input 11 (optional function, please refer to the table below for setting)
	12	X12	Input 12 (optional function, please refer to the table below for setting)
	13	X13	Input 13 (optional function, please refer to the table below for setting)
	14	X14	Input 14 (optional function, please refer to the table below for setting)
	15	X15	Input 15 (optional function, please refer to the table below for setting)
	16	X16	Input 16 (optional function, please refer to the table below for setting)
	17	X17	Input 17 (optional function, please refer to the table below for setting)
	18	X18	Input 18 (optional function, please refer to the table below for setting)
	19	X19	Input 19 (optional function, please refer to the table below for setting)
	20	X20	Input 20 (optional function, please refer to the table below for setting)
	21	X21	Input 21 (optional function, please refer to the table below for setting)
	22	X22	Input 22 (optional function, please

			refer to the table below for setting)
	23	X23	Input 23 (optional function, please refer to the table below for setting)
	24	X24	Input 24 (optional function, please refer to the table below for setting)
	25	GND	power ground

2.3.2 optional functions of input port

Serial number

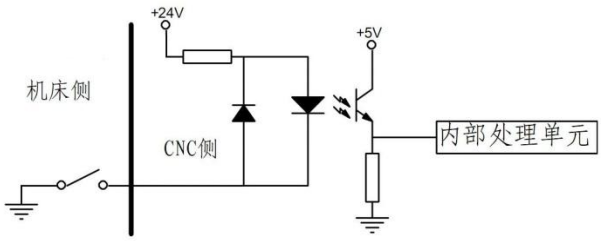
Serial number	function	level	Serial number	function	level	Serial number	function	level
1	Universal	Normally open, normally closed	19	Handwheel X100	Normally open, normally closed	37	A alarm	Normally open, normally closed
2	emergency stop	Normally open, normally closed	20	Spindle 1 alarm	Normally open, normally closed	38	B positive limit	Normally open, normally closed
3	reset	Normally open, normally closed	21	Spindle 2 alarm	Normally open, normally closed	39	B negative limit	Normally open, normally closed
4	start up	Normally open, normally closed	22	X positive limit	Normally open, normally closed	40	B zero	Normally open, normally closed
5	pause	Normally open, normally closed	23	X negative limit	Normally open, normally closed	41	B alarm	Normally open, normally closed
6	cool down	Normally open, normally closed	24	X zero	Normally open, normally closed	42	C positive limit	Normally open, normally closed
7	Spindle 1 rotates forward	Normally open, normally closed	25	X alarm	Normally open, normally closed	43	C negative limit	Normally open, normally closed
8	Spindle 1 reverse	Normally open, normally closed	26	Y positive limit	Normally open, normally closed	44	C zero	Normally open, normally closed

9	Spindle 1 stops	Normally open, normally closed	27	Y negative limit	Normally open, normally closed	45	C alarm	Normally open, normally closed
10	Clamp 1 Release	Normally open, normally closed	28	Y zero	Normally open, normally closed	46	G31 input	Normally open, normally closed
11	HandwheelX	Normally open, normally closed	29	Y alarm	Normally open, normally closed	47	X-axis Z-CP	Normally open, normally closed
12	Handwheel Y	Normally open, normally closed	30	Z positive limit	Normally open, normally closed	48	Y-axis Z-CP	Normally open, normally closed
13	Handwheel Z	Normally open, normally closed	31	Z negative limit	Normally open, normally closed	49	Z-axis Z-CP	Normally open, normally closed
14	Handwheel A	Normally open, normally closed	32	Z zero	Normally open, normally closed	50	A-axis Z-CP	Normally open, normally closed
15	Handwheel B	Normally open, normally closed	33	Z alarm	Normally open, normally closed	51	B axis Z-CP	Normally open, normally closed
16	Handwheel C	Normally open, normally closed	34	A positive limit	Normally open, normally closed	52	C-axis Z-CP	Normally open, normally closed
17	Handwheel X1	Normally open, normally closed	35	A negative limit	Normally open, normally closed	53		
18	Handwheel X10	Normally open, normally closed	36	A zero	Normally open, normally closed	54		

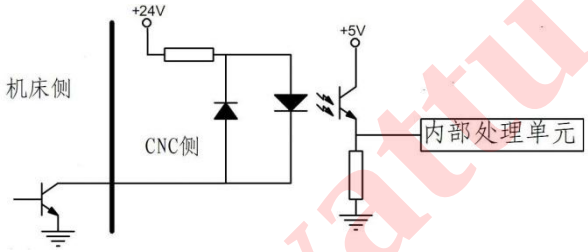
2.3.3 The principle of the input port circuit

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 figure 2.5.31 with contact type input

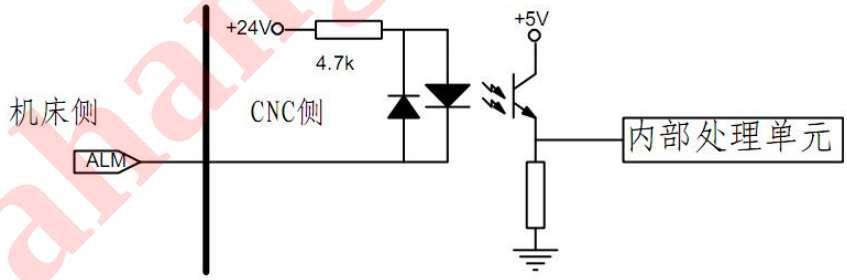


The other uses the non-contact switch (transistor) input, and the connection method is as shown in Figure 2.5.32 Non-contact switch (NPN transistor) input:



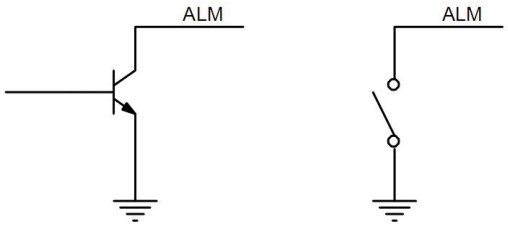
2.3.4 ALM principle of drive unit alarm signal

The internal circuit is shown in the figure below:



Drive unit alarm signal internal circuit

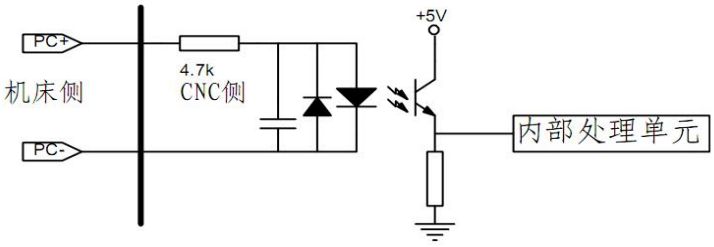
This type of input circuit requires the driver to provide signals in the following way:



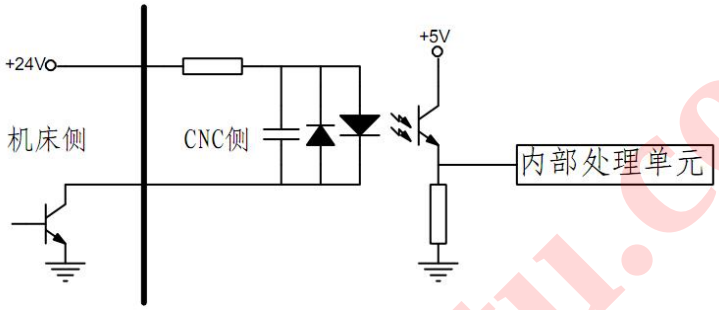
Two ways for the drive unit to provide the signal

2.3.5 CP principle of zero signal

When the machine tool returns to zero, the one turn signal of the motor encoder or the machine tool proximity switch signal are used as the zero point signal.The internal connection circuit is shown in the figure below:

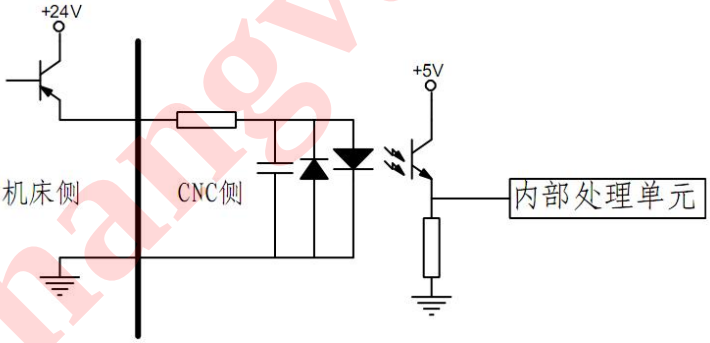


a) When a NPN sensor is used for both deceleration signal and zero signal, the connection method is shown in the following figure:



Connection with NPN sensor

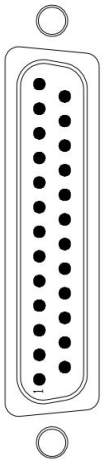
b) When a PNP sensor is used for both deceleration signal and zero signal, the connection method is shown in the following figure:



Connection with PNP type sensor

2.4 output interface

2.4.1 definition of output interface

Interface form	Pin	Signal name	Function description
 DB type 25-pin socket (output)	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 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 073 (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

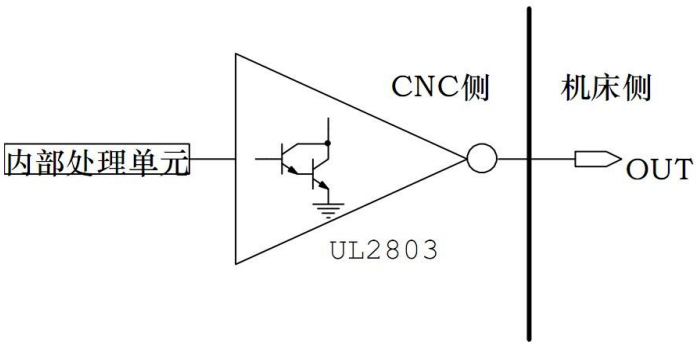
			setting)
	17	Y22	Output 22 (optional function, please refer to the table below for setting)
	18	Y21	Output 21 (optional function, please refer to the table below for setting)
	19	Y20	Output 20 (optional function, please refer to the table below for setting)
	20	Y19	Output 19 (optional function, please 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	clamp	nothing
2	Spindle 1 rotates forward	nothing	8	release	nothing
3	Spindle 1 reverse	nothing	9	green light	nothing
4	Spindle 2 rotates forward	nothing	10	yellow light	nothing
5	Spindle 2 reverse	nothing	11	red light	nothing
6	cool down	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

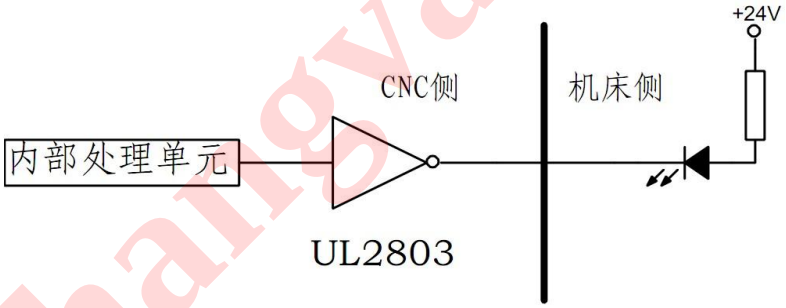
2.4.4 output circuit principle

The logic signal outx 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. Noutx has two output states: 0V output or high resistance;Each output can withstand 200 Ma tube current.

Typical applications are as follows:

Driving LED

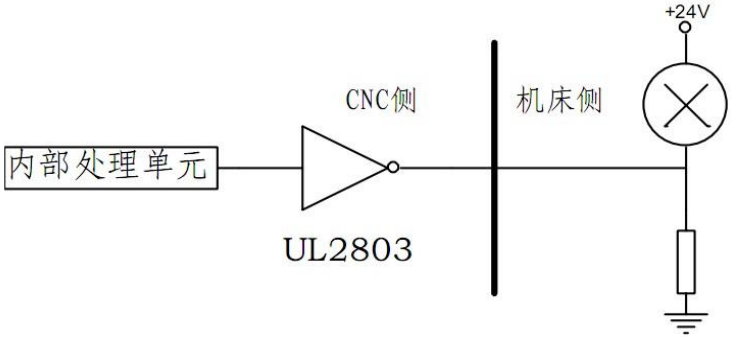
To drive LED with ULN2803 output, a resistor should be connected in series to limit the current flowing through the LED (generally about 10mA).As shown in figure 2.5.71:



Driving LED

Drive Filament Type Indicator

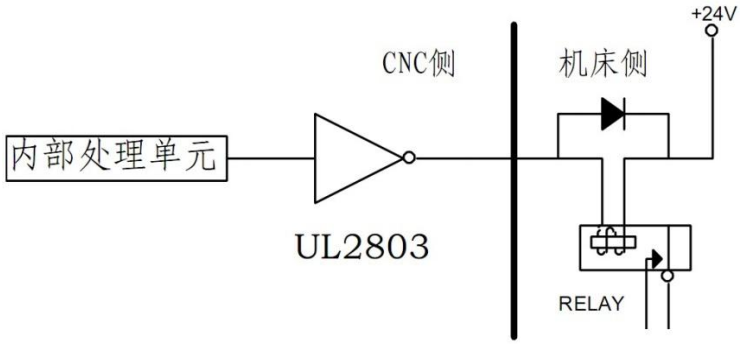
When ULN2803 output is used to drive the filament type indicator lamp, a preheating resistor shall be connected to reduce the current impact when conducting. The preheating resistance value shall be based on the principle that the indicator light is not on, as shown in Fig. 2.5.72.



Drive Filament Type Indicator

Driving inductive load (e.g. relay)

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.

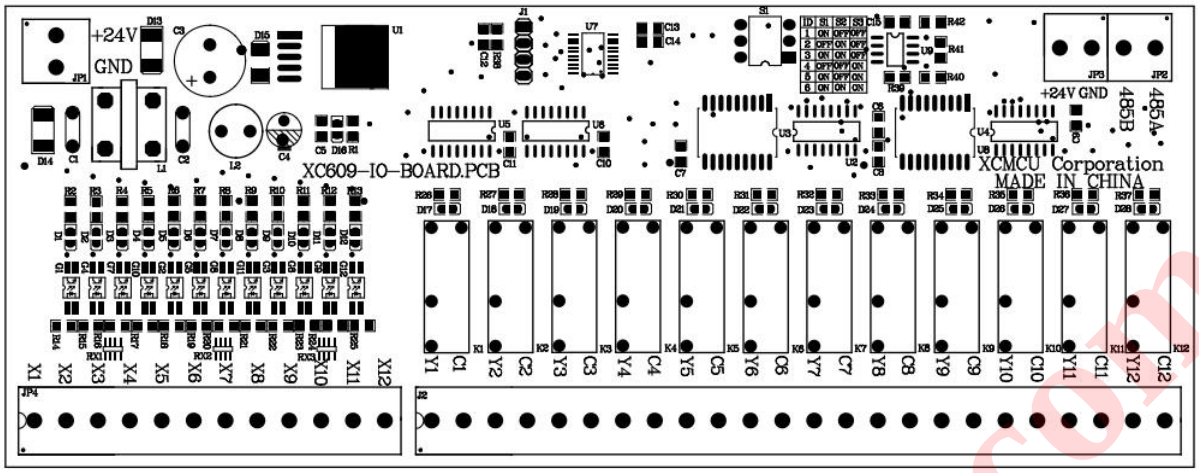


drive relay

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



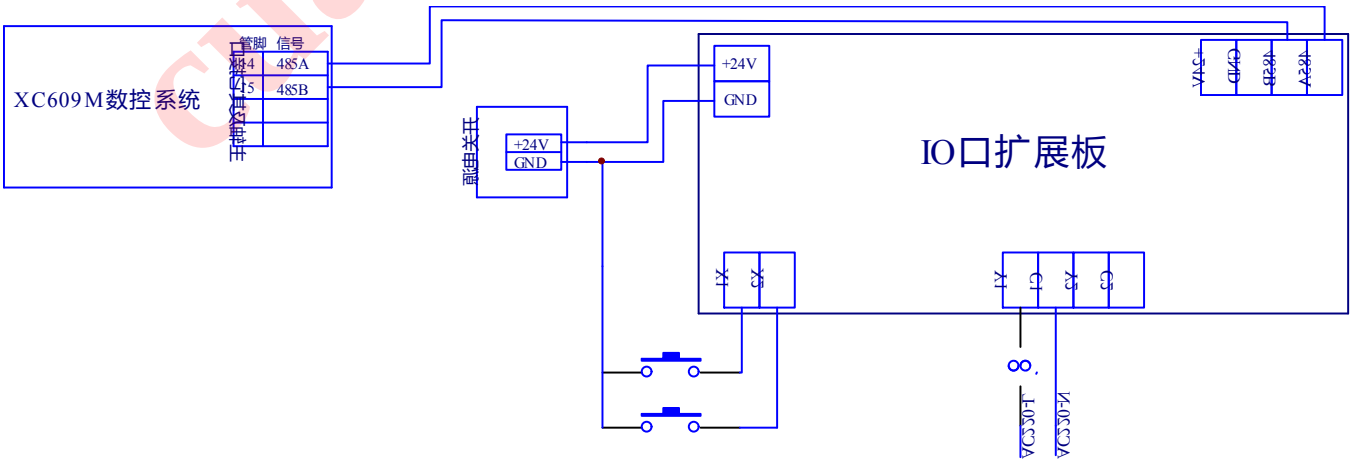
3.2 Use and connection of IO port expansion board

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)	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:

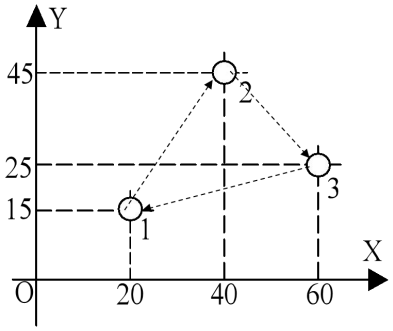


PART 5 is the basic method of NC milling machine programming

1.1 instructions on coordinates and coordinate systems

1.1.1 absolute programming G90 and incremental programming G91

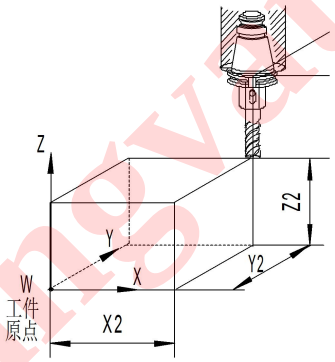
Example: when the tool moves from the origin to 1, 2 and 3 points in sequence, use G90 and G91 instructions to program.




```
%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
```

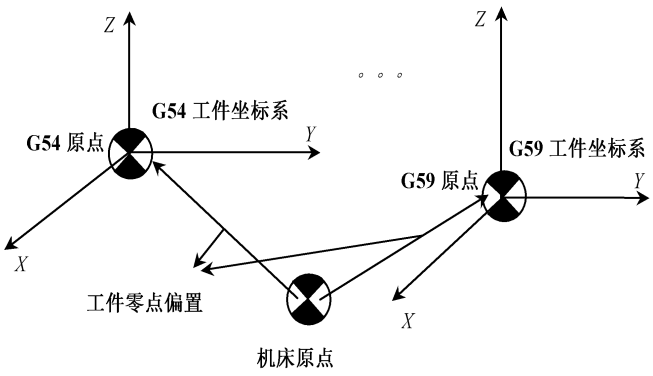
1.1.2 Workpiece coordinate system setting G92



G92 设置加工坐标系

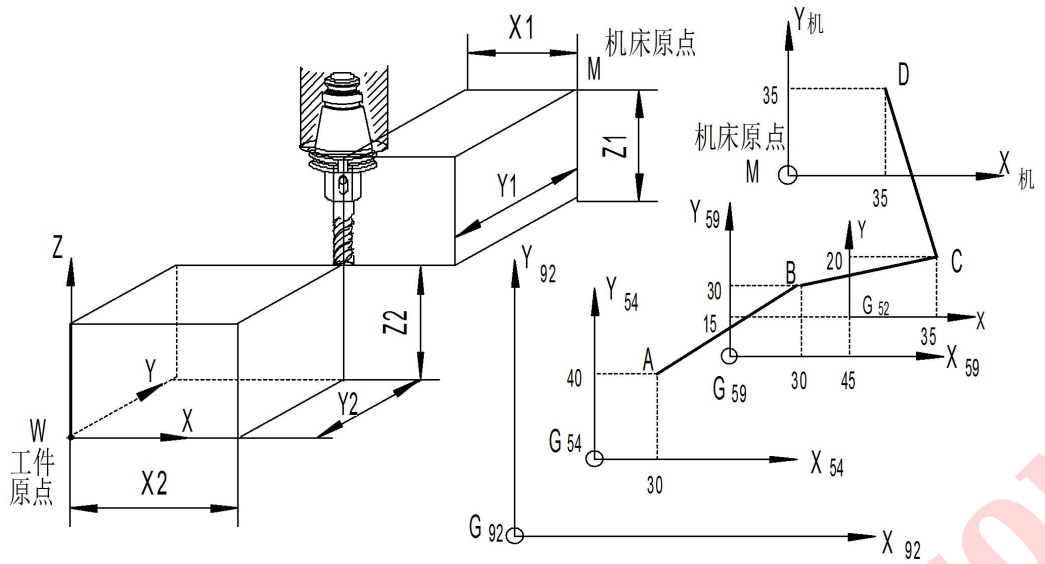
G92 X X2 Y Y2 Z Z2

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.

programming as follows

N01 G54 G00 G90 X30.0 Y40.0 Quickly move to point A in G54

N02 G59 Set G59 as the current workpiece coordinate system

N03 G00 X30.0 Y30.0 Move to point B in G59

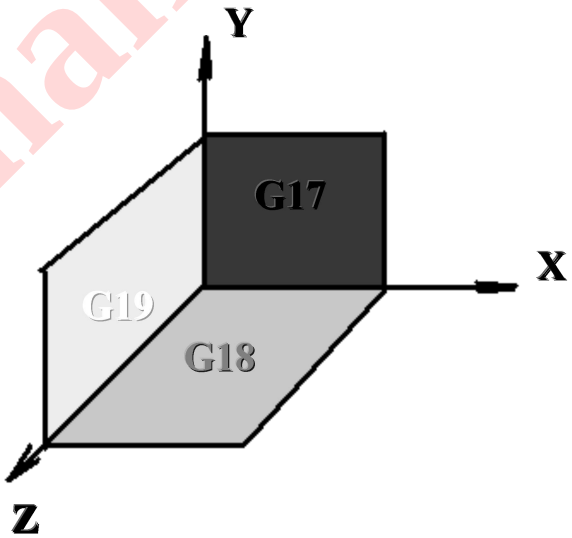
N04 G52 X45.0 Y15.0 Create the local coordinate system G52 in the current workpiece coordinate system G59

N05 G00 G90 X35.0 Y20.0 Move to point C in G52

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

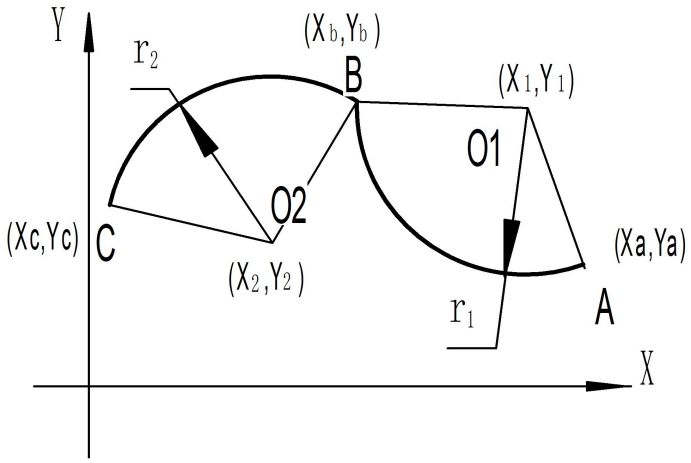
1.2 understanding of coordinate plane

The coordinate plane is G17, G18, G19.



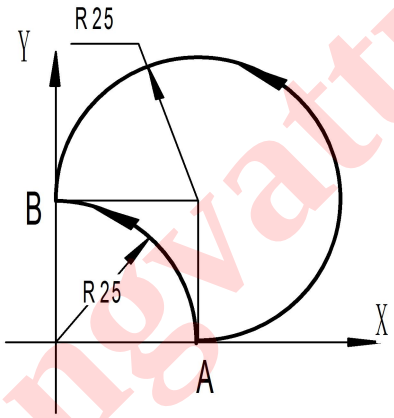
1.3 Supplement of circular interpolation command

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 ; Incremental: G91G02 X (xb-xa)Y (yb-ya) R r1 F f ; or G91G02 X(xb-xa)Y(yb-ya)I(x1-xa)J(y1-ya)F f ;

Example 2: programming arc program segment



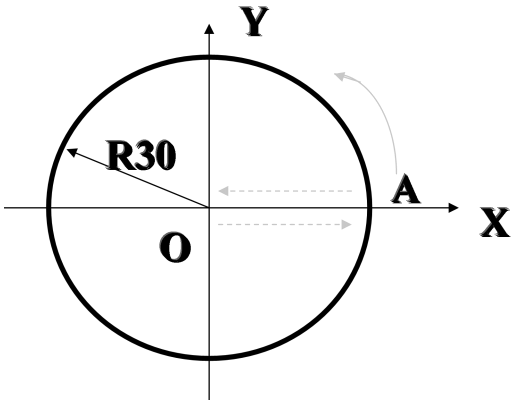
Great arc AB Each arc can be represented by four blocks

G17 G90 G03 X0 Y25 R-25 F80G17 G90 G03 X0 Y25 I0 J25 F80G91 G03 X-25 Y25 R-25 F80G91 G03 X-25 Y25 I0 J25 F80

Small arc AB

G17 G90 G03 X0 Y25 R25 F80G17 G90 G03 X0 Y25 I-25 J0 F80G91 G03 X-25 Y25 R25 F80G91 G03 X-25 Y25 I-25 J0 F80

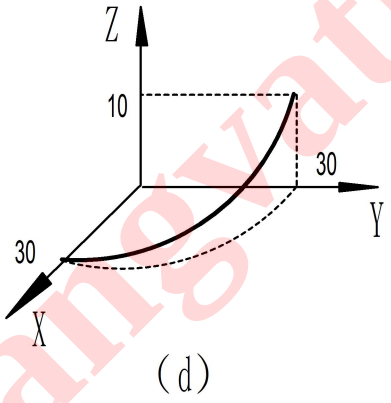
Example 3: Full circle programming



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: space helix feed realized by G02 / G03

That is to add a third axis movement command which is perpendicular to the machining plane at the back of the original G02 and G03 command format program segments. In this way, the feed in the direction of the third axis is carried out while the arc feed is carried out, and the composite track is a space spiral.10. The third and Z axes are the vertical coordinates of the end point of the arc



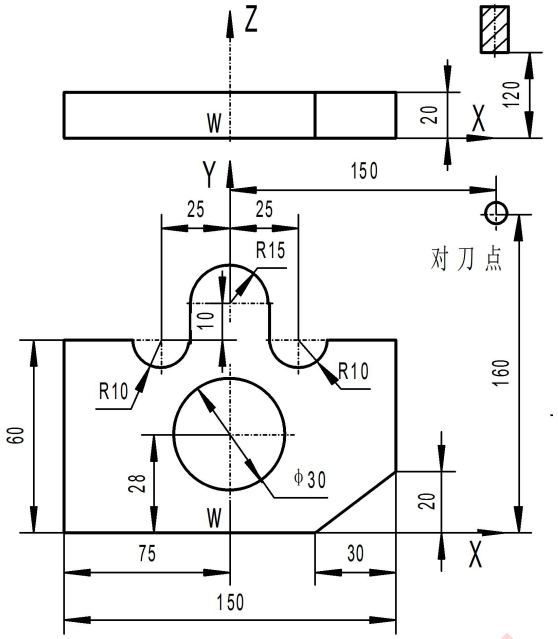
(d)

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 examples of basic instruction programming

Parts as shown in the figure to Φ The tool compensation is not considered for the moment.



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	Quickly lift the tool to the tool setting point plane at Z=120
X150.0y160.0	fast tool withdrawal to tool setting point
M05 M30	program ends, reset.