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:

When the temperature of the system is - 50 ℃ or even beyond the normal working environment, the system will not work normally.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 from entering the control system.

5. Protect the LCD screen of the control system (fragile): keep it 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.

Operation of the system:

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 man-made failure or accidental failure that violates the use requirements;

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

3. Damage caused by natural disasters and other reasons;

4. Damage caused by dismantling, refitting and repairing without permission.

Other matters:

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

The control system is subject to change.

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

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

CATALOG

Part 1 System Overview

1.1 System Introduction

XC709D, XC809D engraving, milling, drilling and tapping multi-functional CNC system is a new generation of CNC system developed by our company.Support carving, milling, drilling, tapping processing.It uses 32-bit high-performance microprocessor, real-time multi task control technology and hardware interpolation technology, full linkage, high-speed small line look ahead algorithm.The interpolation accuracy is 0.001mm and the maximum speed is 30 m / min.It is the best choice for engraving machine, small machining center, CNC milling machine and CNC drilling machine.

The software and hardware characteristics of xc709d and xc809d CNC system are as follows

Based on 32-bit microprocessor, full linkage, 0.001mm interpolation accuracy, maximum speed of 30 m / min, support direction + pulse and orthogonal pulse.

It adopts 7-inch color wide screen LCD with resolution of 800×480, windows interface style.Equipped with 8 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 and English operation interface, complete help information, more convenient operation.

The system adopts acceleration and deceleration control before interpolation.

It supports long tool compensation and radius compensation.

It supports the trial processing of handwheel, which is convenient for program debugging.

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

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

It supports self programming tool library, and can flexibly use a variety of tool libraries.

1.2 Technical Specifications

basic function	
Number of control axes	Axis 3 ~ 6 (x, y, Z, a, B, c)
Number of linkage axes	Full linkage
Tool magazine	Code library, need to edit
principal axis	There are 2 analog spindles, of which spindle 1 can use digital spindle (occupying one digital axis)
Spindle monitoring	yes
Minimum instruction unit	0.001 mm
Maximum instruction value	±99999999×Minimum instruction unit
Rapid feed rate	30000 mm / min
Rapid feed rate	F0, 25%, 50%, 75%,100%
Pulse mode	1: Direction + pulse 2: quadrature pulse (recommended if driver supports, stronger anti-interference)
Maximum frequency	100kHz, 200kHz, 500KHz can be set.(500KHz can only be realized in all axes orthogonal output mode)
Rate feed rate	0～150%
Electronic gear ratio	1～65535: 1～65535
Automatic acceleration and deceleration	yes
Forward looking algorithm for high speed	yes

small line segments	
Trial machining of handwheel	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	7-inch TFT LCD with resolution of 800×four hundred and eighty
MDI software key	8
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	Single phase AC220 V + 10% - 15%, 50 Hz ± 1 Hz
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
Digital spindle	Spindle 1 supports, occupies a digital axis (Z axis does not support), and the speed is more stable.
tapping	support
Spindle analog output	Yes, double spindle
Tool function	
Tool function	It supports automatic tool change, tool setting in center and three-point centering
Tool compensation memory	-9999.999 ~ 9999.999, 99
Tool compensation	Knife length compensation and radius compensation
Edit operation	
Editing function	Parameters, diagnosis bit input, program editing, MDI multi program segment execution
storage capacity	512M
Number of stored programs	500
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
Xc709d host size (mm)	
Overall dimensions (height×wide×Thick)	328×236×80
Opening size (height×Wide)	305×207
Xc809d host size (mm)	
Overall dimensions (height×wide×Thick)	390×253×80
Opening size (height×Wide)	370×228

PART II CONNECTION AND COMMISSIONING

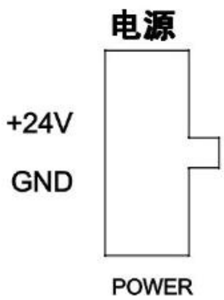
Chapter 1 Interface

1.1 Interface Introduction

The back of the system has power supply, input, driver, spindle and other, output interface.Each interface is marked with specific functions beside the back of the system.Therefore, the interface function is viewed on the back of the system.

Note: the interface line sequence in the manual is used for internal circuit diagram, not actual value.The actual line sequence shall be subject to the mark beside the interface on the back of the system.

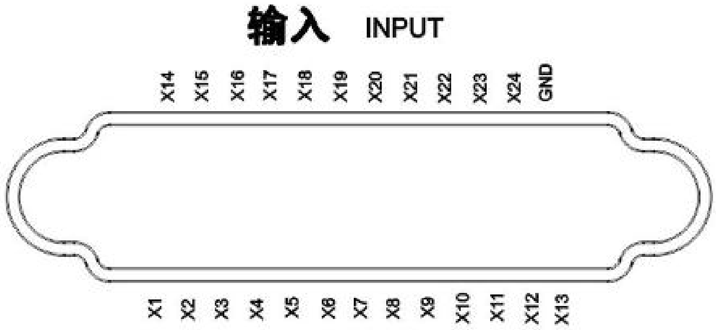
1.2 Power Interface



If you have your own power supply, it is forbidden to use transformer plus rectifier bridge, and try not to use LED power supply with 10A or above.

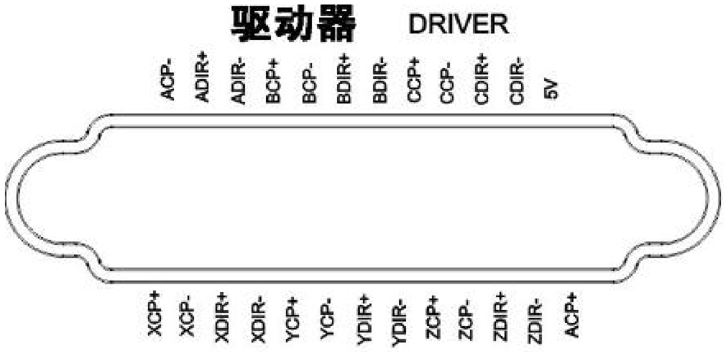
Note: except for the power interface, it is strictly forbidden to connect the power supply to other interfaces of the system. The voltage marked on other interfaces is the voltage provided by the system from outside.

1.3 Input Interface



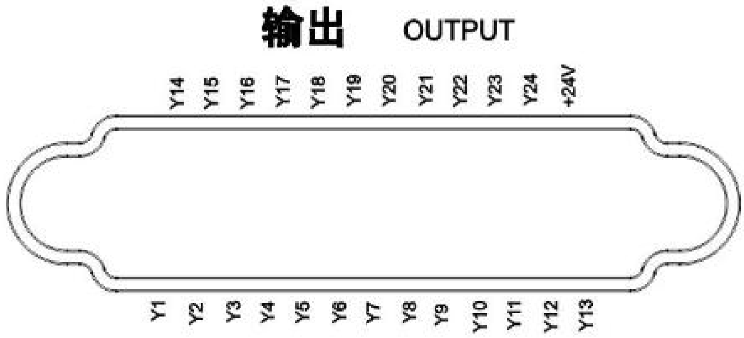
X1 ~ x24: input port 1 ~ x24

1.4 Driver Interface



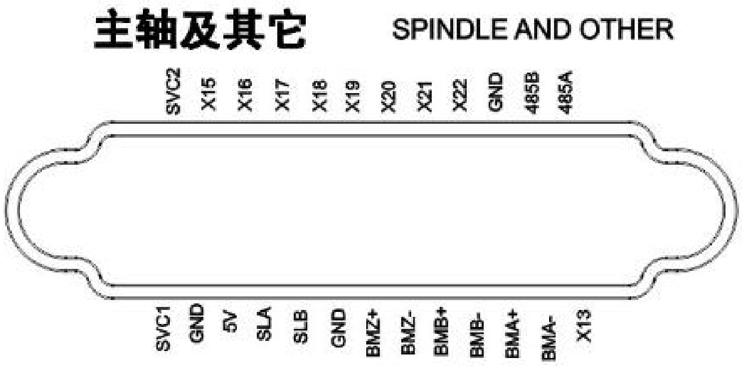
Note: 5V voltage is only common positive connection, and can not be used in other ways. It is forbidden to connect external 5V power supply.
CP + (pulse positive), CP - (pulse negative), dir + (square positive), dir - (square negative).Each axis is distinguished according to the previous name. For example, Z axis corresponds to ZCP +, ZCP -, zdir +, zdir -.

1.5 Output Interface



Y1 ~ y24 are output ports 1 ~ 24

1.6 Spindle And Other Interfaces



Svc1: analog quantity of spindle 1 (0 ~ 10V)
Svc2: analog quantity of spindle 2 (0 ~ 10V)
485io, 485a communication board

SLA, SLB: handwheel a, B signal

BMZ+,BMZ-,BMB+,BMB-,BMA+,BMA-;Spindle encoder interface.

be careful:

1. In this interface, x13 ~ X22 are reserved for the selection rate of handlebar axle, and the input terminals with the same name in the input port are connected.So there are no two x13 and others.

2.5V voltage can only be used as power supply for handwheel or spindle encoder.There can be no other usage.It is forbidden to connect external 5V power supply.

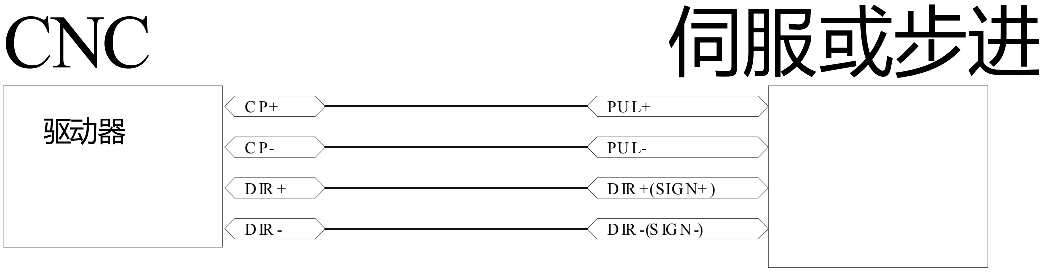
Chapter 2 Interface Connection

2.1 Drive Connection

Each driver only needs 4 wires to run. If it is a servo, please set the internal enable of the driver.

2.1.1 Driver Signal Connection

There are four signals on each axis.CP + (pulse positive), CP - (pulse negative), dir + (square positive), dir - (square negative).Each axis is distinguished according to the previous name. For example, Z axis corresponds to ZCP +, ZCP -, zdir +, zdir -.(the same as the orthogonal pulse connection, only the system and driver parameters need to be changed).



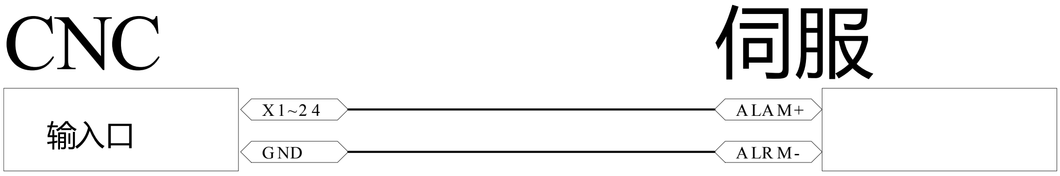
On electronic gears:

Take the screw rod as an example, without reducer.If the screw pitch is 5mm, set step or servo 5000 pulses per cycle, servo set numerator 2 and denominator 1.The ideal state is that one pulse goes one micron.So the denominator of the system is 1.If it cannot be set to 1:1, then the system molecule is the number of pulses per revolution of the screw, and the denominator is the pitch in microns.

About stepper motor debugging:

Generally, the maximum speed of three-phase stepping is 1000 rpm, and that of two-phase stepping cannot exceed 800 rpm.Some customers use 24 V two-phase stepping, so the motor has no torque.Step drivers with external power supply should use the highest voltage marked as far as possible.For speed calculation, if the maximum speed is 800 and the pitch is 5, then the maximum speed is 800 * 5 = 4000In the axis parameter, [fast speed G00] is set to 4000If there is a reducer, divide it by the reduction ratio.

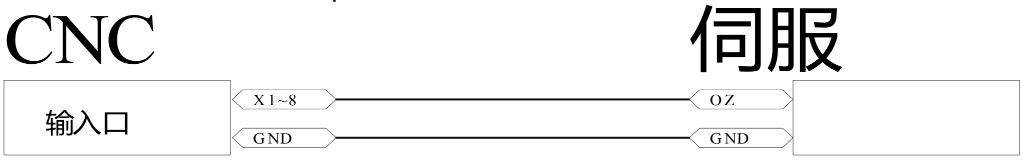
2.1.2 Driver Alarm Connection



The GND here does not have to be in the input port GND, it is recommended to connect to the negative pole of the power supply.After connection, press the [modify] key to set the corresponding port in the diagnosis as [alarm of certain axis], and set it to be normally closed (generally, servo alarm is normally closed signal).

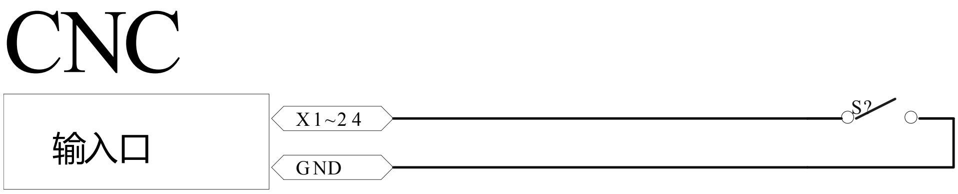
2.1.3 Driver Z-Pulse Connection

If high-precision return to zero is required, the return to zero mode in the parameter uses + ZCP, which needs to be connected with the servo Z pulse.



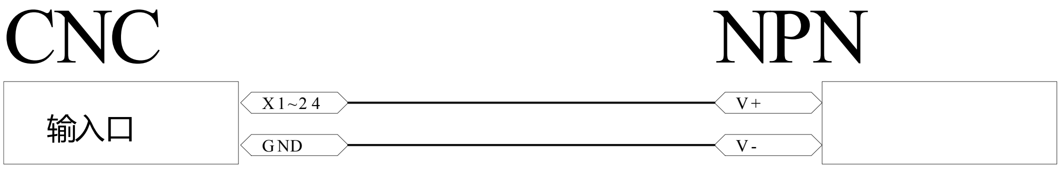
Note: the system requires that the servo has an open collector output of encoder Z signal.It can only connect input port 1 ~ 8.After connection, the corresponding port is set to [ZCP of a certain axis].

2.2 External Key, Proximity Switch And Sensor Connection



Note: if it is an electronic sensor, only NPN type is accepted, PNP cannot be used.

Two wire NPN sensor is easy to trigger.Some customers ask what kind of sensor has high precision and good stability. Maybe it is a slot type photoelectric sensor.



As for normally open and normally closed, in addition to emergency stop, other external keys should not be used as far as possible.For example, a customer has set the external [auto] (there is no connection at this time). After setting the normally closed, the external [auto] will be triggered all the time (because it is disconnected).You can't switch to other states, and you can't change it, because you need to modify it in the editing state, you can only connect a wire to the negative pole (do not let the automatic signal trigger) and then change it back.

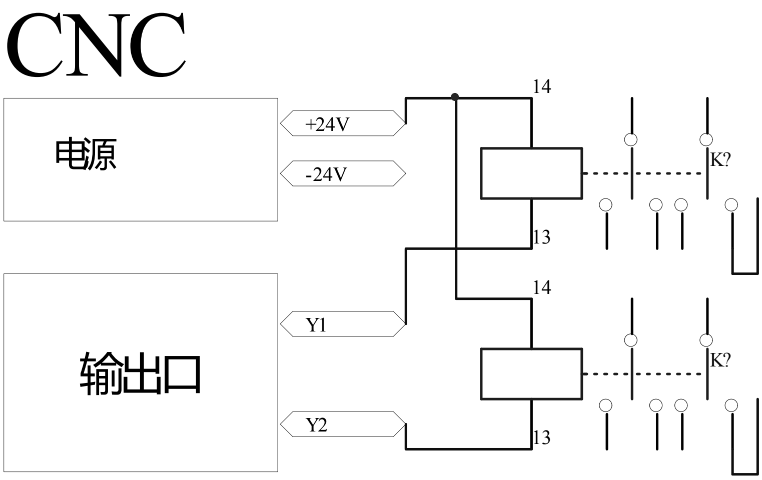
The signal of tool setting instrument can only be connected to X1 ~ 8 ports.

After the input port is connected with the signal, you can go to the [diagnosis] and trigger the button or sensor to check whether the diagnosis status changes.Switch to the editing state and press the modify key to change the corresponding function of the input port.Normally open and normally closed are set in parameters.

2.3 output connection

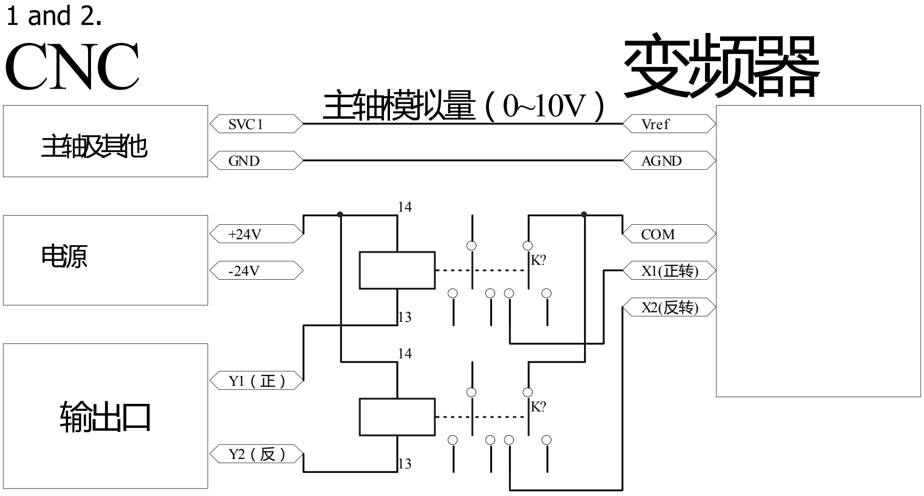
The output port of the system can only be connected with the 24 V relay. Due to the limited current, the solenoid valve, contactor and other uses must be switched through the 24 V relay.It is forbidden to connect 220 V AC relay or other electrical appliances.

The effective output voltage of the output port is 0V, and if it is invalid, it is in the state of disconnection (the result can not be measured with a multimeter, because it is suspended with uncertain leakage voltage).



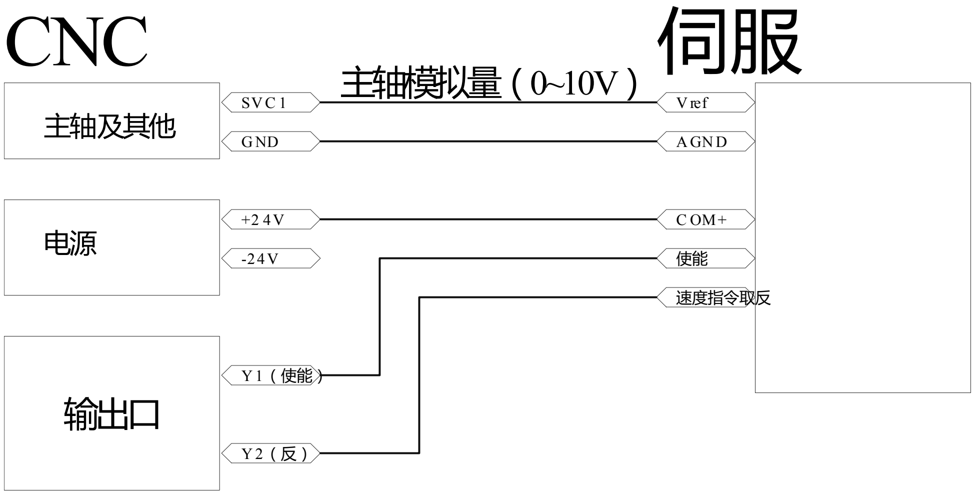
2.4 spindle connection simulation

The system supports two simulated spindles, spindle 1 and spindle 2 respectively, and signals are also distinguished by 1 and 2.



In the output diagnosis, Y1 and Y2 set spindle 1 forward rotation and spindle 1 reverse rotation respectively.Of course, not only Y1, Y2, Y1 ~ y24 can be connected, as long as the output port is set to the corresponding function.

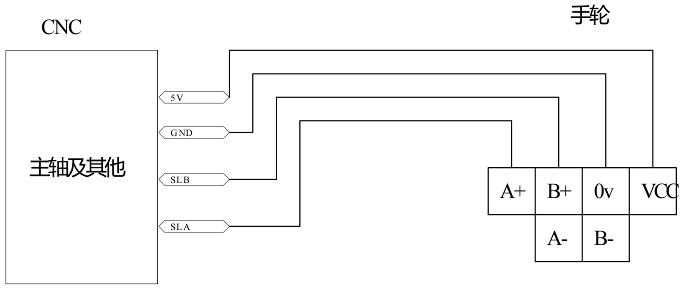
If the servo is used as the simulation spindle:



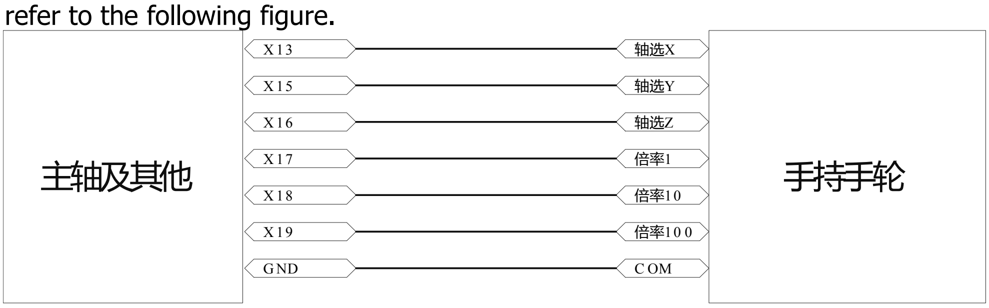
The servo cannot be enabled by soft.[reverse speed command] generally, the servo is not set in the factory. Please refer to the servo manual to set the corresponding input port as [reverse speed command] (the name is not necessarily the name, but the function is reverse command in speed mode).Not sure. Consult the servo manufacturer.

2.5 hand wheel connection

Single handwheel connection:



If it is a hand-held handwheel, the handwheel signal still refers to the figure above.Axis selection and magnification refer to the following figure.



Hand held three axis handwheel routine.Enter the input diagnosis, turn the axis selection and rate switch, the corresponding port will change, press [modify] to set the corresponding port to the correct function.

When the handwheel related function is set in the input port function, the system will automatically switch to the handheld mode, and the panel buttons will not work.Therefore, a single handwheel should be used to ensure that there is no handwheel related function in the input port (for example, one input port is [handwheel X axis]).

Chapter 3 commissioning

3.1 motor commissioning

3.1.1 motor starting

If this is the first time to use this system, please do not install the motor on the machine first, make sure that the motor can rotate before starting the machine.

After the motor is connected and powered on, if it is a servo, set the servo soft enable.The motor should be locked.The system switches to manual mode.Press the axis direction key, the motor should rotate.If not, check the cable and drive.

Servo motor: the parameter [pulse mode] of the system is set to quadrature pulse, and the driver should also be set to quadrature pulse (some servo is called AB pulse).

3.2 electronic gear

3.2.1 calculation method

Servo: take general encoder 2500 line motor as an example

The servo motor pulse per cycle is 10000 * servo denominator / servo numerator.If servo numerator 10, servo denominator 5, the motor pulse per cycle 10000 * 5 / 10 = 5000

Step: look at the breakdown table on the drive.

The screw pitch L is in microns, 1 mm = 1000 microns.5 mm pitch.

The motor should be set to 5000 pulses per cycle (similarly, the numerator 10 and denominator 5 should be set according to the above servo driver), that is, one pulse goes 1 micron.

Then the system numerator 5000, denominator 5000, can also be reduced to 1:1

If there is a reducer molecule to multiply the reduction ratio, the motor pulse per cycle can be appropriately reduced.

If the numerator of the system is 360000, the number of revolutions per minute is 360000.

3.2.2 test method

If you don't know how much distance each cycle of pulse goes, you can use the test method, and the test distance is as long as possible.

The denominator of the current system is I: J.Run at low speed ln = 100 mm (for example, not necessarily this value).Then measure the actual distance LM, then the new numerator / denominator = (I * LN) / (J * LM)Then you can continue to measure LN and LM, and then repeat the calculation until there is no error in the measurement.

If the axis of rotation needs to be used in reverse, use the scale mark to turn a circle to align. At this time, the actual LM = 360 degrees. See how much the system ln displays, and put it into the formula calculation.(do not set the rotation axis until the electronic gear is correct).

3.3 maximum speed estimation

If the command issued by the system is too high, the servo motor will give an alarm, and the stepper motor will lose step or even stall.It is very important to estimate the maximum speed and limit it. At the same time, it can be set to the maximum speed to improve the processing efficiency.

Servo motor: look at the maximum speed of the motor M, screw pitch l mm, then the maximum speed is m * L.

Stepper motor: the stepper motor should not exceed 800 rpm, and the load should be reduced.M=800.The algorithm is the same as above.

Whether the motor can reach the maximum speed depends on the system frequency limit.There are 200K, 500K system.

100k: the main reason is that some step drivers will lose pulse when they are higher than 100k.

200K: factory default.

500K: all axes must be orthogonal pulses.If one axis is not orthogonal, it will drop to 200K.

Frequency limit / motor pulse per cycle * 60 = theoretical maximum speed.

If there is a deceleration ratio a: B, the maximum speed should be * B / A

The maximum speed of each axis is set in the fast speed G00.In actual operation, the system will control each axis to be no higher than this speed.

[speed ratio] in the parameter: sometimes we need to rotate the axis according to 360 degree division control, so the speed will be very slow. For example, 3600 motor has only 10 cycles per minute, and the efficiency is low. In this case, we can set the [speed multiplier], and the final speed will be multiplied by the speed to speed up the speed.The speed in the parameter is also multiplied by the multiplier.[speed multiplier] only for parameters and G0 speed, f is not affected in the program.

Stepping motor out of step: step motor out of step, first of all, the driver voltage should not be low.Stepping motor in high speed torque depends on the supply voltage, low on the decay quickly.Second, reduce the acceleration and deceleration. It's not good to reduce the acceleration and deceleration to 10.Reduce the speed, the speed drop is very low is not good, the motor is too small, change the motor.

3.4 Limit Protection

3.4.1 hardware limit protection

In general, limit switches are installed in the positive and negative directions of each axis, which can only move within the limited range of the limit switch.When the limit switch is triggered, the system will stop the tool movement immediately (actually stop with the emergency stop acceleration and deceleration), and display the limit trigger alarm information.At this time, if you want to move the tool in reverse direction, press [reset] to release the alarm.

3.4.2 software limit protection

Software limit protection and hardware limit protection have similar functions.The limit range of the machine tool can be set as the positive and negative coordinates of the machine tool.

The practical application is a combination of a hardware limit and a software limit.

3.5 set reference point (return to mechanical zero)

The system has the function of saving coordinates after power failure, but it is not reliable for the motor to slide after power failure or power failure in motion, so it is necessary to operate power on and return to zero.

The system has four return to zero modes.Set in each axis parameter.In general, it is recommended to return to zero limit for linear axis and zero return for rotating axis (the limit switch will alarm in motion).

To return to mechanical zero, first set the input port to be used for returning to zero, otherwise an alarm will be given [zero return port mismatch].

Limit return to zero, return to zero direction forward, must have [axis positive limit] port, if additional ZCP, also have [axis ZCP] signal port.If it is a negative return to zero, there must be a [axis negative limit] port.

Zero return to zero, there must be [axis zero point] port, if additional ZCP, also have [axis ZCP] signal port.

ZCP return to zero, there must be [axis ZCP] signal port.

After returning to zero, the machine coordinate will be reset automatically.

Second, the three reference points are machine tool coordinates, unit is micrometer.

3.6 simulation spindle speed setting

The system has two simulation spindles, spindle 1 and spindle 2.If the system speed is corresponding to the actual speed, it is necessary to set [analog 10V corresponding speed] in the comprehensive parameters.The analog port of spindle 1 is svc1, and that of spindle 2 is svc2.

Spindle 1 can be configured with encoder to achieve tapping function.Set 1000 rpm, spindle 1 programming format is S1000, spindle 2 programming format is ss1000.

3.7 description of some parameters

In this system, each parameter has a detailed explanation, but some parameters are not common, which are further explained here.

3.7.1 maximum pulse frequency [in comprehensive parameters]

This parameter limits the maximum frequency of the command pulse output, which is shared by all axes, and the lowest one in the motor shall prevail.

The maximum output of 0:100 is 100kHz, some step driving quality is general. If it exceeds 100kHz, the pulse will be lost, so the limit of 100kHz should be set.

1: The maximum output is 200 kHz.

2: 500 kHz maximum output.If one axis is direction + pulse, the control will return to 200kHz, even if the parameter shows 500KHz.

3.7.2 action delay of fixed cycle chuck [in comprehensive parameters]

This parameter is used with output port [g8n chuck].In g83 and other non g80 fixed cycles, some customers need chuck to clamp the workpiece, and then release the chuck after completion.The output port [g8n chuck] can be set.

The actions are as follows: machining axis reaches r point → [g8n chuck] output → fixed cycle chuck action delay → fixed cycle start → fixed cycle end, [g8n chuck] close → fixed cycle chuck action delay → non machining axis action continues to the next hole.

3.7.3 expansion module address 1 ~ 6 [in comprehensive parameters]

The system can expand the IO port. This parameter controls the opening of the specified address IO board.

Each IO board has 12 inputs and 12 outputs.The address s (1 ~ 6) is specified by dial switch, and the name of IO port is different.

The specific calculation is: X (y) n = x (y) n + (s-1) * 12 + 24 on the expansion board

For example, the address of Y1 on S1 is 1 + (1-1) * 12 + 24 = Y25

For example, Y3 address on S6 is 3 + (6-1) * 12 + 24 = y87

Parameter setting 0: do not communicate with this address expansion board. If there is no connection address, set 0, otherwise the communication speed will be affected.

Parameter setting 1: communicate with the expansion board of this address, no alarm will be given when it is disconnected.Five consecutive communication errors are considered to be disconnected.

Parameter setting 2: communicate with the expansion board of this address and alarm after disconnection.Five consecutive communication errors are considered to be disconnected.

The communication status can be viewed in diagnosis → more → communication diagnosis.

3.7.4 axis mode [in axis parameters]

[axis mode] 0 rotary axis, 1 linear axis, 2 digital spindle, 1 and 3 tool magazine

If it is set as the rotation axis, the system will implement 360 degree rotation control (coordinate display range 0 ~ 359.999), and the absolute programming will use the nearest mode.If the current 300 degrees, G90 A0, the control will go forward to 360 degrees instead of reversing.But the incremental mode will follow the procedure.91 a-3600 turns.

In the rotating axis mode, the absolute coordinate and the machine coordinate are always the same, and the offset and cutter compensation of the axis are invalid.The absolute coordinates can be set by setting the machine coordinates with G93.The speed of the rotating shaft is slow, and you can set [speed multiplier] to speed up the speed.

Digital spindle 1 and tool magazine are set with electronic gear according to the rotation axis.

Digital spindle 1 can replace analog spindle 1 to achieve more stable and accurate speed.It can realize the interpolation type rigid tapping, which is easier to realize the positioning.

In the tool magazine mode, the movement can only be realized in the tool magazine return to zero, t code and M06 code to control the servo magazine.

3.7.5 shaft speed ratio [in shaft parameters]

Setting the speed of the rotating axis is too slow, which will affect the efficiency. For example, when the speed of the rotating shaft is 3600, the workpiece is only 10 cycles per minute.Therefore, the machining efficiency can be improved by using [shaft speed multiplier].If the [axis speed multiplier] is set to N, then the workpiece will speed up by N times, and the acceleration will also increase by N times (when the distance is small, the acceleration will greatly affect the speed).If it is a stepper motor, set the completion rate to run several times to see if the return to 0 degree is coincident, so as to avoid losing step due to too high acceleration.In general, do not use magnification for linear axis.

[speed multiplier] only for parameters and G0 speed, f is not supported in the program.In the parameter, [return to zero low speed] is invalid for [shaft speed multiplier].

3.7.6 offset after returning to mechanical zero [in axis parameters]

This system uses limit switch to return to zero (save a zero switch).After returning to zero, if it stops near the limit switch, a slight vibration will trigger the limit alarm. Therefore, set this parameter far away from the limit switch to prevent false triggering.The unit is micrometer. The direction system will judge automatically without adding symbols.

If ZCP is added, it can be moved to the middle of two zcps (the specific distance should be tested) to prevent the pitch error caused by the position close to ZCP before ZCP detection.

3.7.7 drilling function on and off [in integrated parameters]

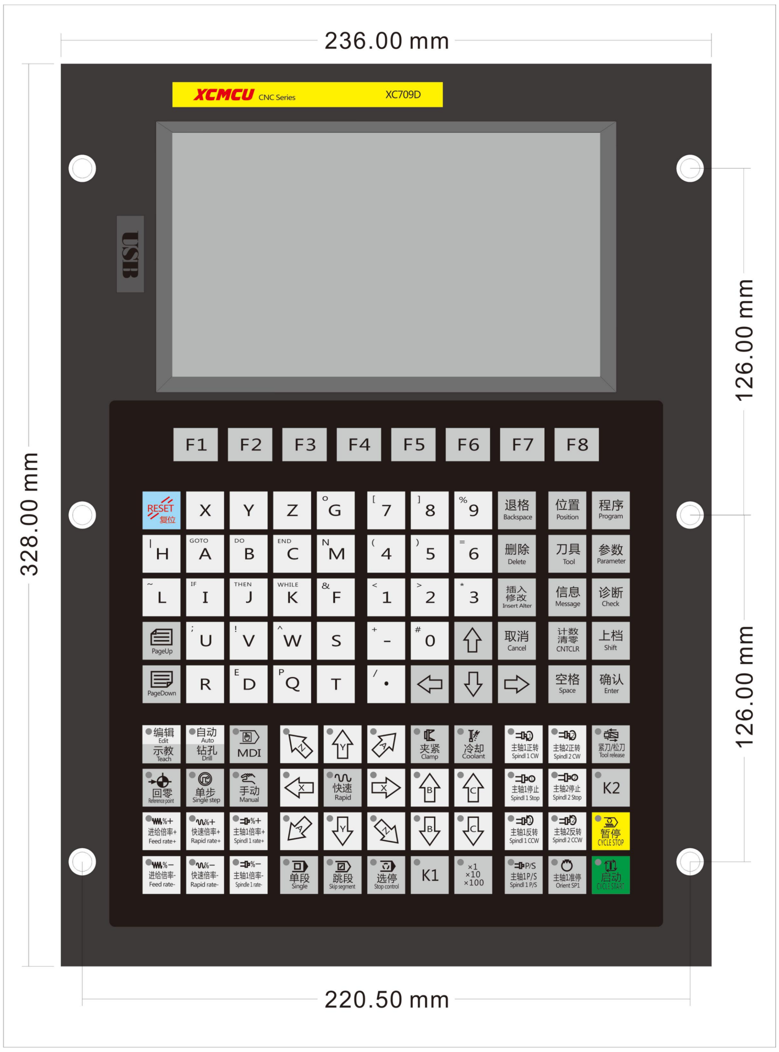
Set 0 to prohibit opening the drilling function, which is convenient for users who do not need the drilling function, and prevents the wrong operation of the drilling function.

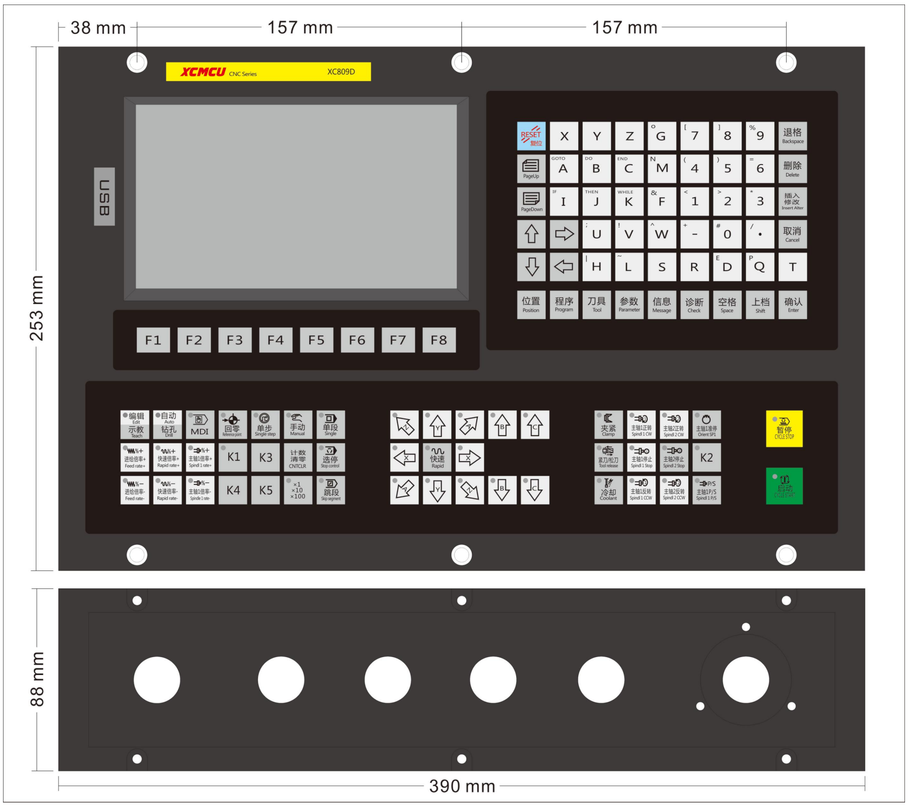
3.7.8 profile error [in comprehensive parameters]

Set the speed of the look ahead algorithm in the high speed range as much as possible.The larger the acceptable error, the higher the speed.Of course, under the given speed, the controller will try to reduce the error.

Part 3 operation instructions

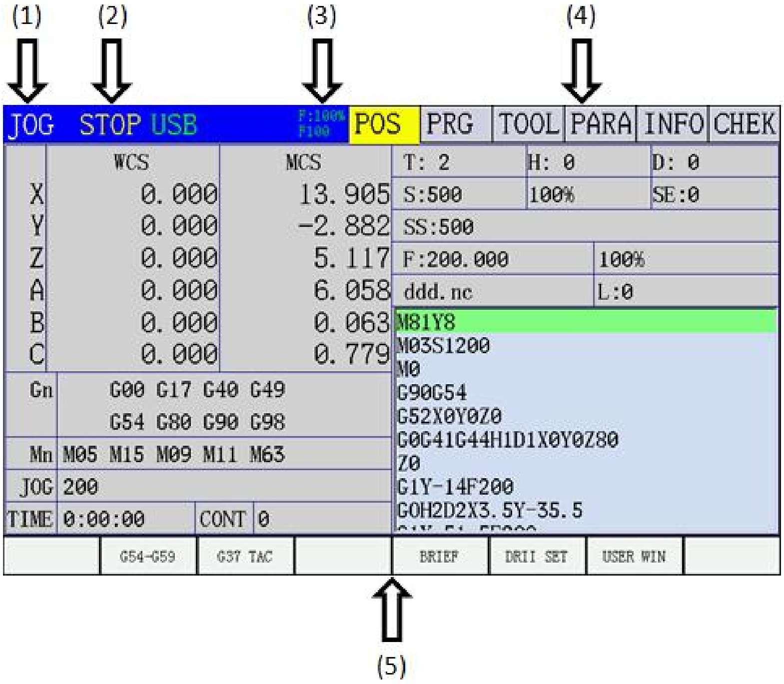
Chapter 1 Display And Interface Setting





1.1 Page Display

1.1.1 Page Layout



project	explain
(1) Working mode	Edit, teach, auto, drill, MDI, return to zero, single step, handwheel, manual
(2) Operation status	Stop, run, pause, alarm
(3) Rate indication	F: 100%, G1.F100, G0.
(4) Page name	The currently selected master page label
(5) Soft function key	Keys set by F1 ~ F8 are available on the current page

1.1.2 page display content

The system is divided into six display pages, which are [position], [program], [tool], [parameter], [information], [diagnosis].The contents 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 Program information during automatic operation	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
nformation	● 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

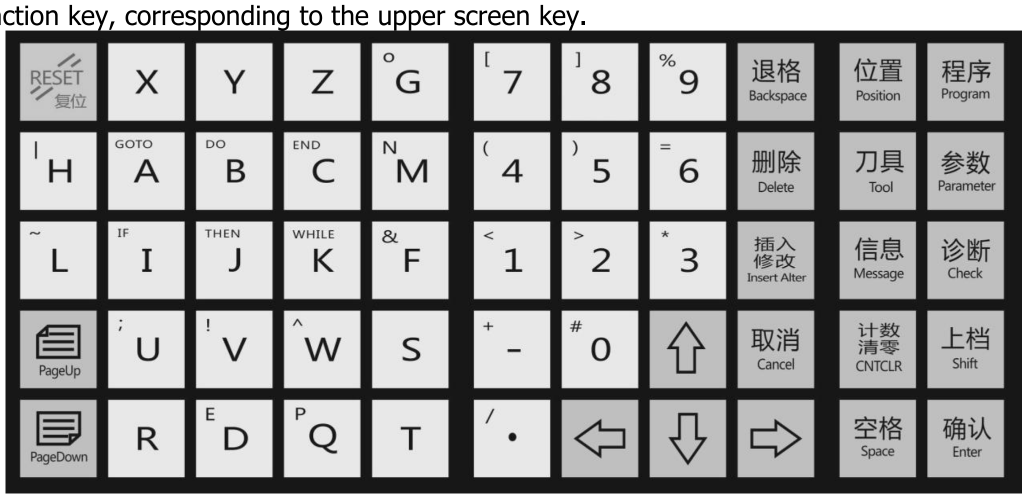
1.1.3 soft function key menu

Each main page is switched to each sub screen through the soft function key. The soft function key function is triggered by the user's pressing and lifting action. According to the operation form, it is classified as follows:

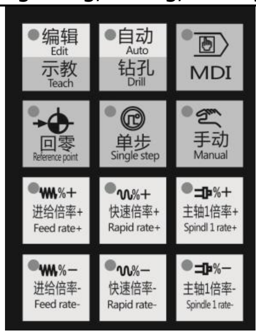
A	Highlight, do not operate the page
B	Enter the next submenu
C	Page display options or display content switching
D	Pop up window

1.2 key description

Soft function key, corresponding to the upper screen key.



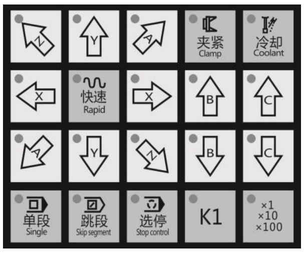
Character input area and page switch key area.



Working mode switching and magnification setting area.

The [edit teach] button is used to switch [edit state]: the indicator is always on, and [teaching state]: the indicator is flashing.

[automatic drilling] button is used to switch [automatic state]: the indicator light is always on, [drilling state]: the indicator light is flashing.



Manual control area.

[fast]: switch between manual and single step speed.When the light is off, use [manual low speed], set in the comprehensive parameters (shared by all axes), and the rate is feed rate F: 100%.Use the "F100" button to set the "fast" button on each axis.

[K1]: no key defined.

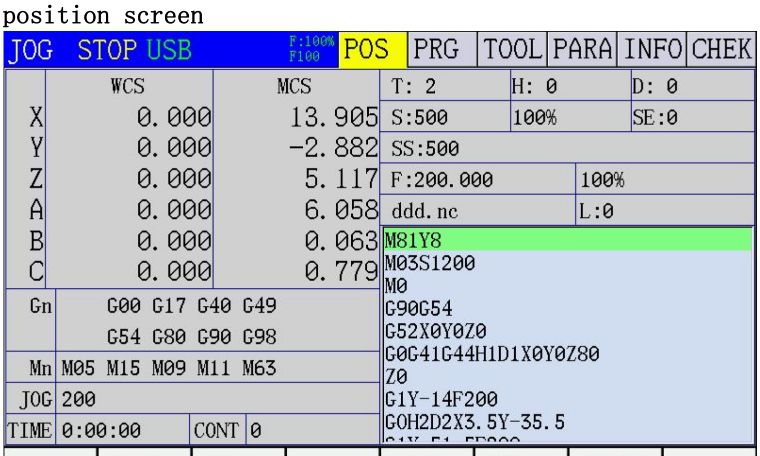
[X1 X10 X100]: adjust single step or handwheel ratio.

Spindle control and program start area.

[spindle 1p / S] spindle 1 position / speed mode switch (Digital spindle).Position mode when the light is on.

1.3 position screen

1.3.1 composition of position screen



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

S: Spindle 1 set speed se: spindle 1 encoder measured speed.

SS: setting speed of spindle 2

1.3.2 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.The machining axis can only be z-axis.

Depth: the absolute coordinates of the hole bottom.

Safe height: when multi axis drilling, the tool positioning moving height.

F: Drilling speed, tapping instruction is pitch.

Spindle speed: spindle 1 speed, can not control spindle 2

Current hole number: the number of the current hole in multi axis.If it is set at stop, the start starts with the current hole number.

Stop position: the position where XYZ stops when the whole workpiece is processed.

R: R data in fixed cycle, coordinate of fixed cycle start machining, refer to fixed cycle instruction.

Q: Q parameter in G73 and g83.

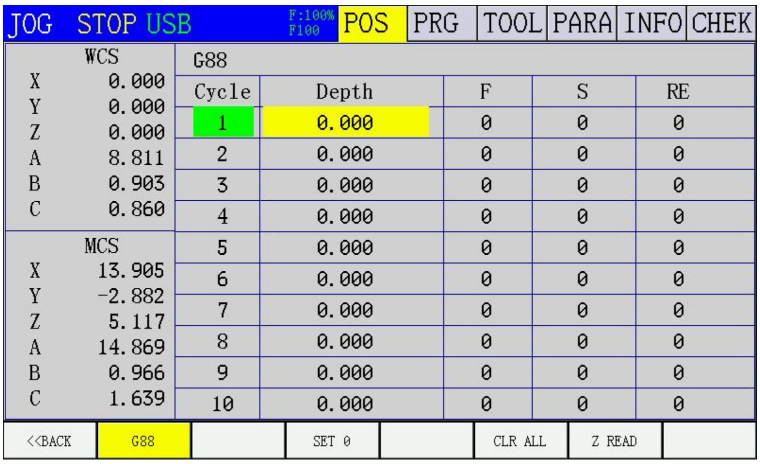
d: D data in fixed cycle, D parameter in G73 and g83.

P: Hole bottom delay P (MS) in fixed cycle.

When cooling: when cooling down.

Clamping: automatic clamping at the beginning of machining.

1.3.3 custom fixed cycle G88



G88 is a customizable fixed loop.A G88 cycle can be set to complete in 20 minutes.Different speeds, speeds and chip removal methods can be set each time.

Depth: the machining depth of the current tool, which is calculated from the R plane.If it exceeds the program specified depth, press the program specified depth and end the following cycle. If the current depth is 0, press the specified depth and end the following cycle. In other words, the tool times after 0 are ignored.

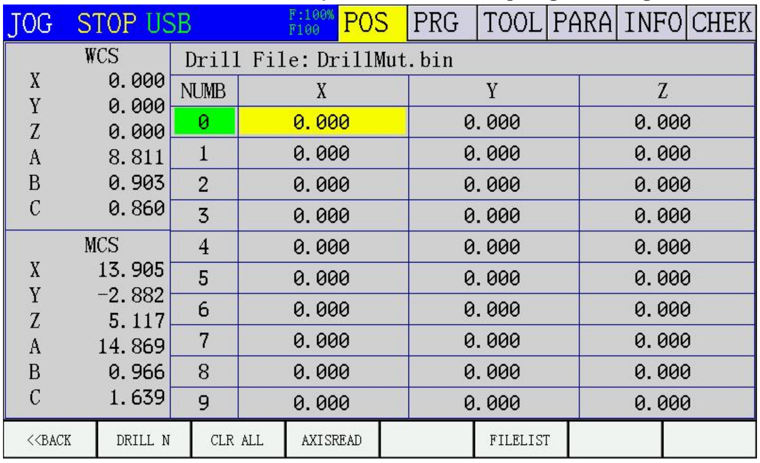
Speed: the current feed speed of the cutter. If it is 0, press the speed specified in the program.

Speed: the current tool spindle speed 1, if it is 0, according to the speed specified in the program. Cannot control spindle 2

Chip removal: 0: the current tool does not return. 1: Back to the R plane, and then quickly to the depth of g83_d。2: Back off g83_d。In case of drilling function, use special D.

1.3.4 multi hole editing interface

If it is two axes or more, multiple hole positions can be set. CAD coordinate extraction tool can be used to extract all point data without programming. However, it can only control XY position, ABC can't control it, so it is still solved by traditional programming.

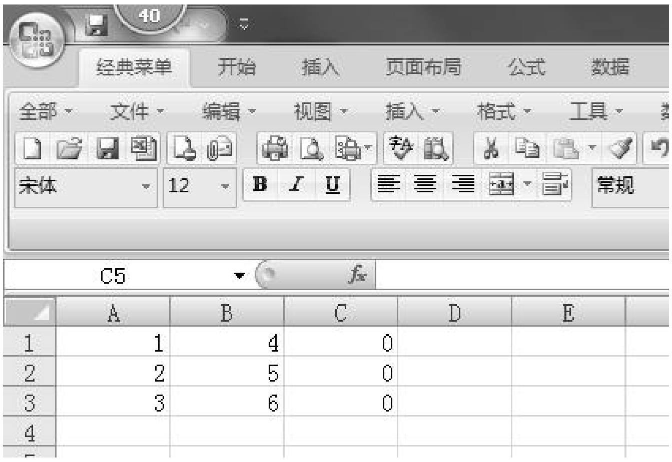


XY, non machining axis, hole position, except for the first hole, the hole with XY coordinate of 0 will be encountered later. After drilling, the data will be ignored.

Z. If the depth of Z is not pressed, the hole will not be machined by Z axis.

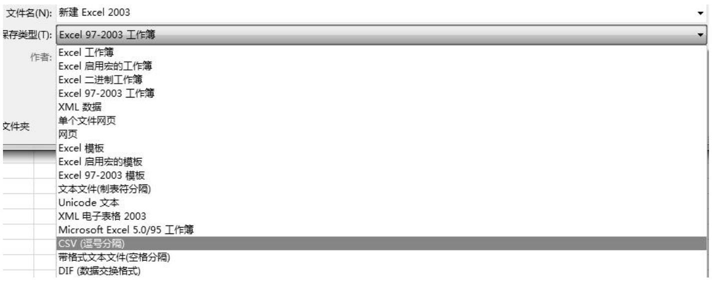
In the file directory, you can use USB flash disk to import data. If not, export it first, change it on the computer, and then import it back.

The file format is CSV. The following shows how to make a CSV file. Other projects need to use a CSV file. The key is to save the file.

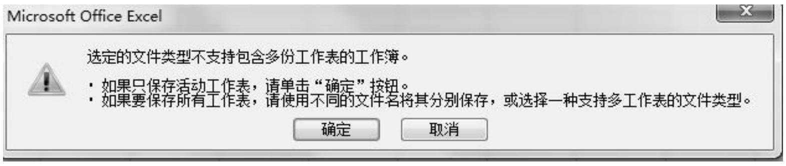


Create or open a file with Excel. Column A is the x-axis data, column B is the y-axis data (that is, the two-axis system also needs to set the number of column B, which can be 0), and column C is the z-axis (there must be several numbers). The first line can not have other things, directly start the data, if it is the data extracted by CAD, delete the data that is not data.

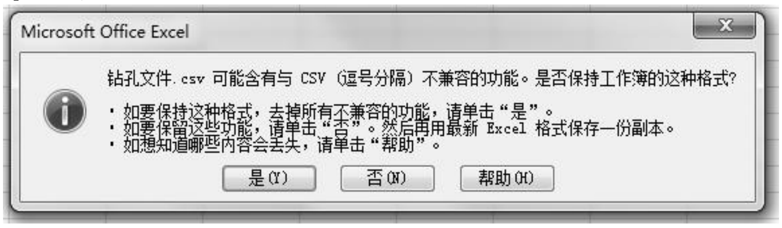
Click the icon in the upper left corner of Excel , select another format.



Select CSV and save it in the file name.

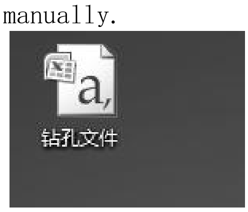


If there is this point, make sure.



Continue [yes].

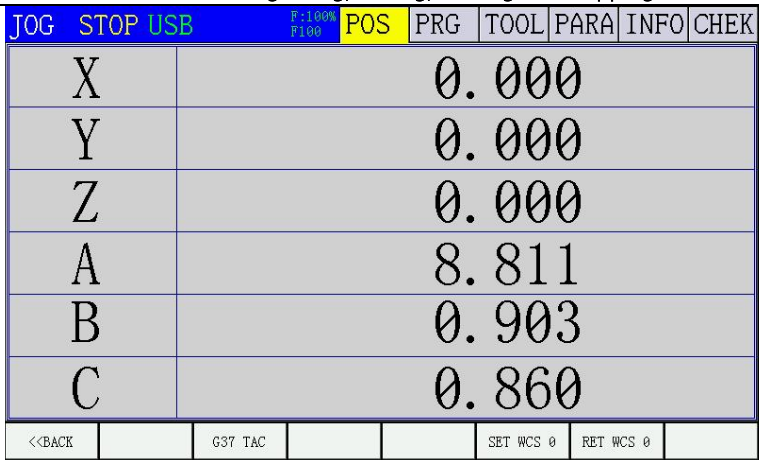
When you close excel, you will be prompted. Click No.However, if there is any modification before closing, it is better to save it manually.



The file in the directory is good. Put this in the USB flash disk and directory, and you can see and import it into the system.

1.3.5 brief display

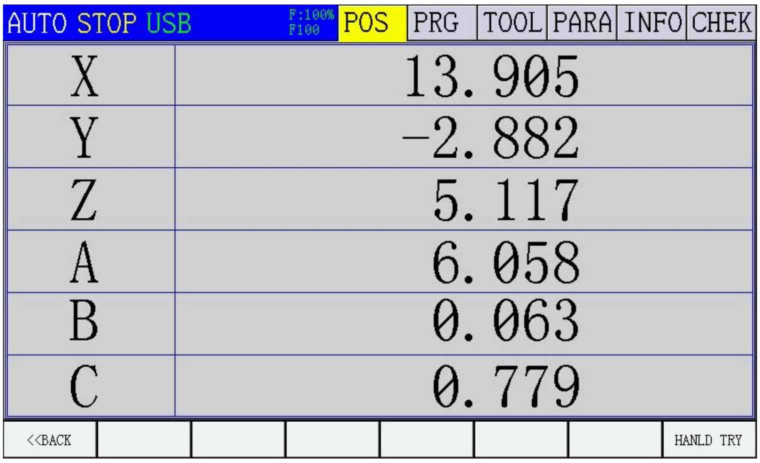
Manual mode lower interface



In manual mode, you can set the current point as the origin and quickly return to the origin. When returning to the origin, z-axis returns to [z-axis safe height] in [comprehensive parameters], and other axes return to coordinate 0.

[tool setting device tool setting], if the tool setting device port is set, press this button to start automatic tool setting.

Auto mode lower interface

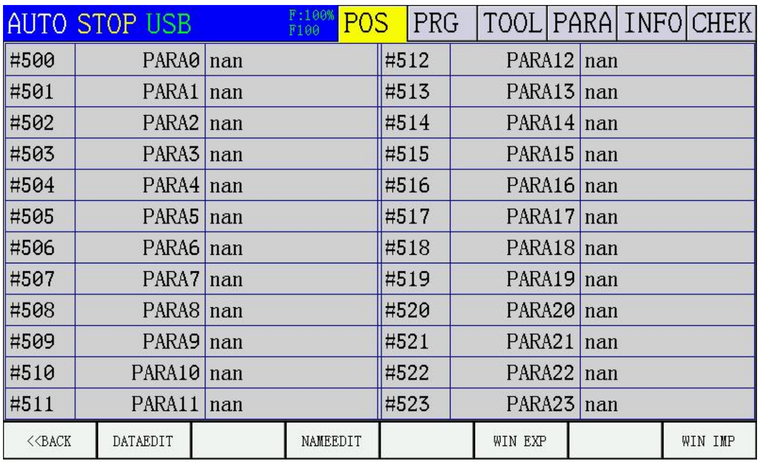


In the automatic mode, press [handwheel trial processing], and the handwheel debugging program can be used.

1.3.6 coordinate tool setting

Sketch, will be explained in detail in tool setting operation.

1.3.7 customization interface



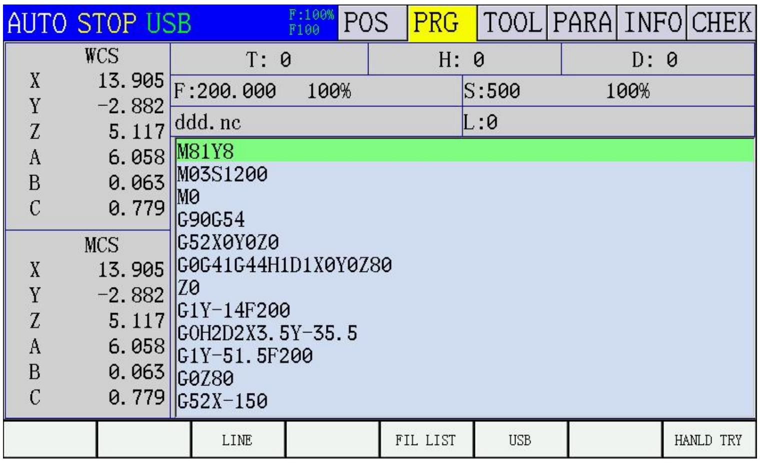
You can define 24 variable numbers that can save common variables B (#500～#599) in the first column, and name them as the second column, and the third column is the value of the variable. nan means invalid value. Use the variable name example X#500 directly in the program.

When importing the interface, use CSV file. If you are unclear, please export it from the system first, and then modify it on the computer and import it back. A parameter example.

500	Self defined parameter 1
501	Self defined parameter 2

Note that the variable number in the first column should not be added with ○J.The range is 500-599, and no other values will be displayed.

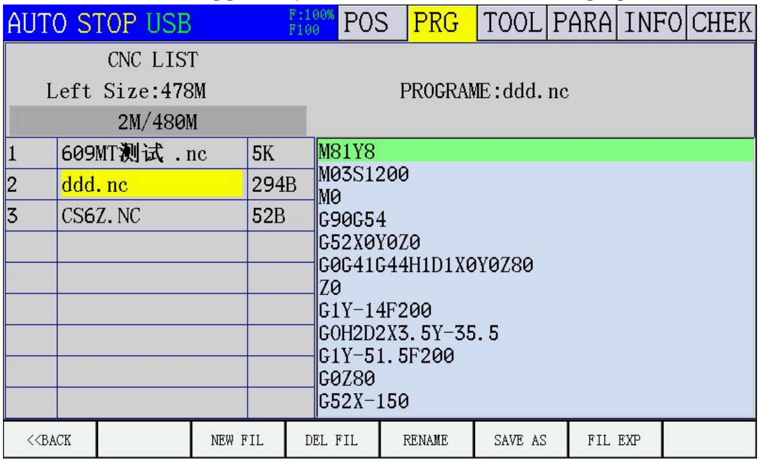
1.3.8 program interface



The system supports NC, CNC, tap and txt files.

1.3.9 local directory

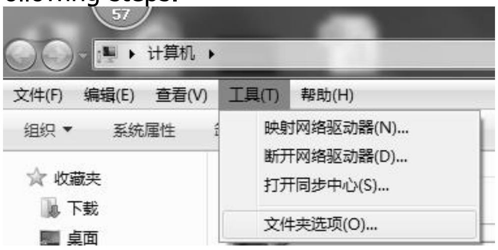
When the running state is not stopped, you cannot enter this page.



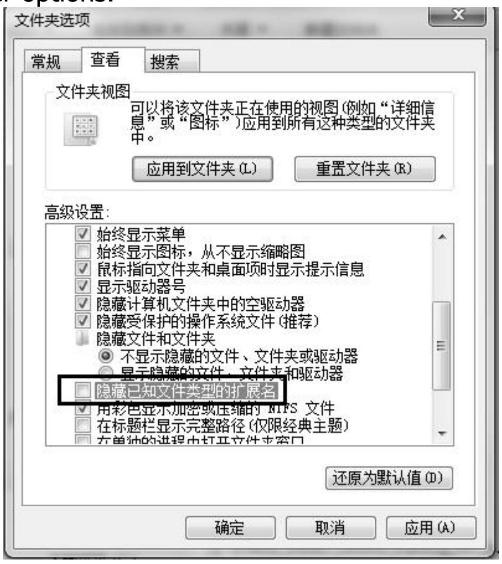
1.3.10 USB flash disk directory

Program import to enter the U disk directory, insert the U disk, enter the U disk directory (the running state is not stopped, can not enter).The system lists the files that can be imported from the USB flash disk.

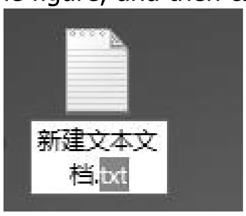
What files can be identified?Condition 1: Notepad can be opened on the computer;Condition 2: the suffix name isNC ortxt 。Some software generated code suffixes are not supported. You can directly change the suffix name to make the system recognize it.The suffix name on the computer is basically hidden. You can open the suffix name through the following steps.



Open my computer tools folder options.

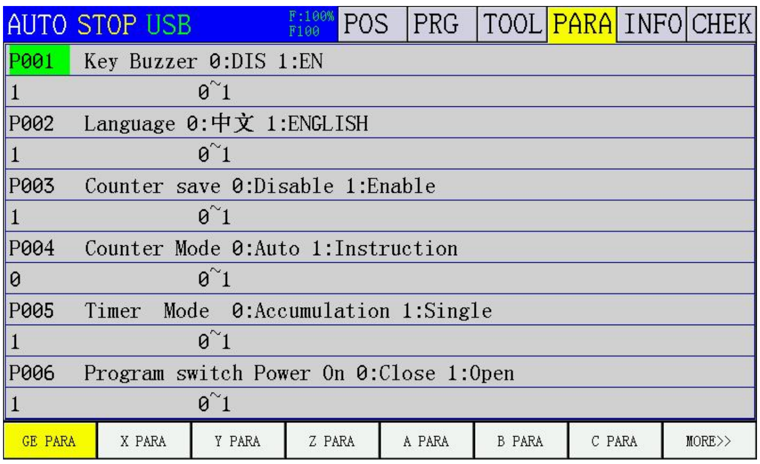


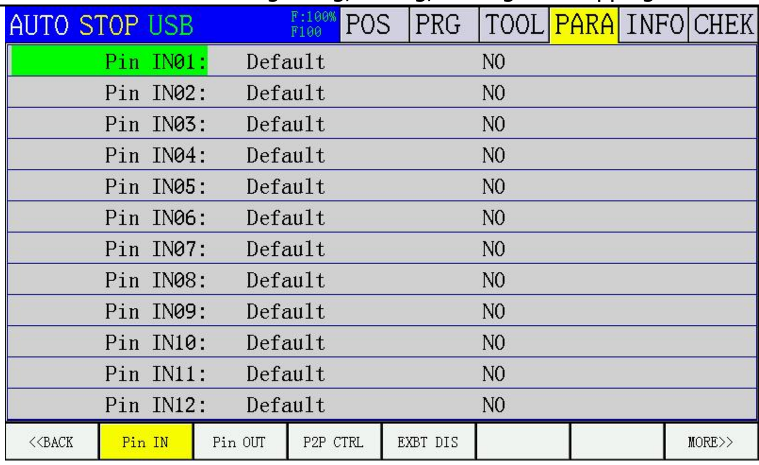
Remove the option ' √ ' from the box in the figure, and then click apply.



At this time, you can see the real suffix name of the file. Select it with the mouse and change it to ". NC".

1.3.11 parameter interface





When setting parameters, the working mode should be in the [Edit] state, and the parameter switch in the system information should be on.Press 

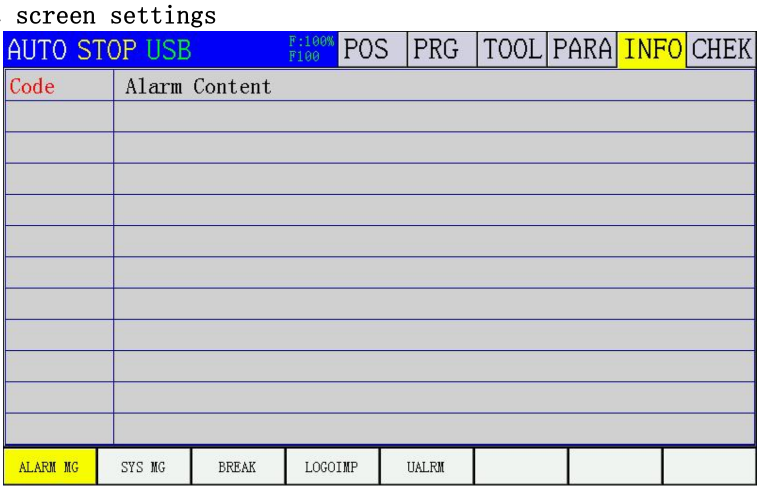
Key modification.

[external control prohibition]: it refers to the port external function of [Edit], [automatic], [MDI], [zero return], [single step], [manual].

It is mainly used when the non edit port is set to normally closed and the port is not connected with a key.In this case, you cannot switch to edit

To modify the port function, use external control disable to switch to edit. After setting the port, press external control disable again to cancel.

1.3.12 alarm and boot screen settings



In case of alarm, you can switch to the information to view the specific alarm information.Press [reset] to clear the alarm.

[breakpoint information]: the line interrupted by the program last time or the hole number interrupted when drilling.

Set the boot screen to startBMP is placed in the root directory of the U disk, and the authority is set to [manufacturer b] in the system information. Press [boot screen] to import a new boot screen.

start.BMP production: open the computer drawing software, create or open the existing map. If new

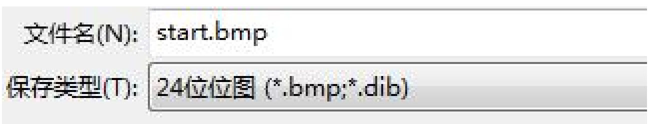
Click properties as shown in the figure.

According to the above settings: unit pixel, width 800, height 480.

To open an existing graph:

Confirm after setting according to the above figure.

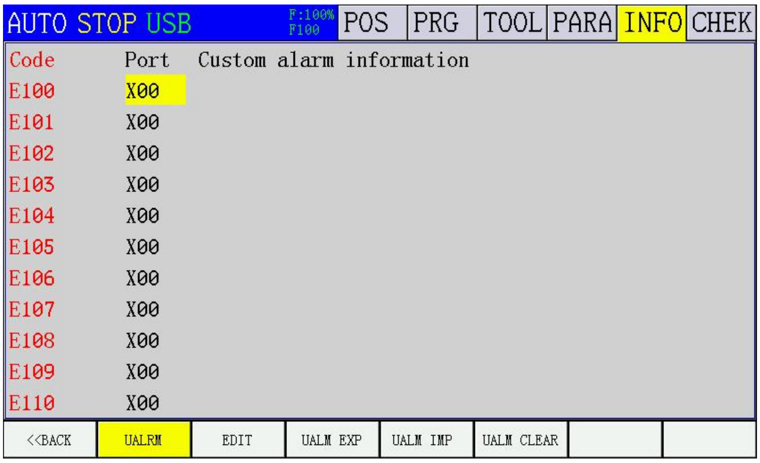
After making the picture



Name must be startBMP, put in the U disk can be imported.

The system can also support E100 ~ E121, customized alarm.Examples can be called from err [100] to err [121] in a program.

Custom alarm content

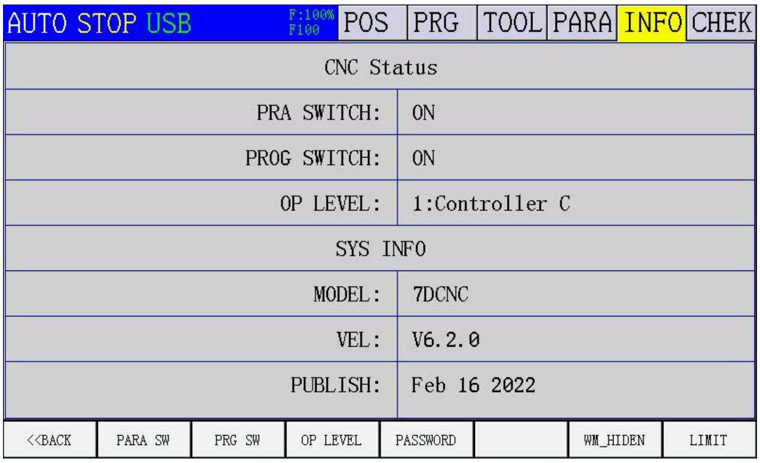


The alarm content can be imported into the CSV file, and only the customized content can be updated in the CSV file.as

E100	Custom alarm 1
E120	Custom alarm 2

Note: Custom alarms do not change with Chinese and English.

1.3.13 system information



Press the switch status and switch.Set the permissions corresponding to the permissions, F level does not need a password, C and B level default password is 888888.

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

1.3.14 trial settings

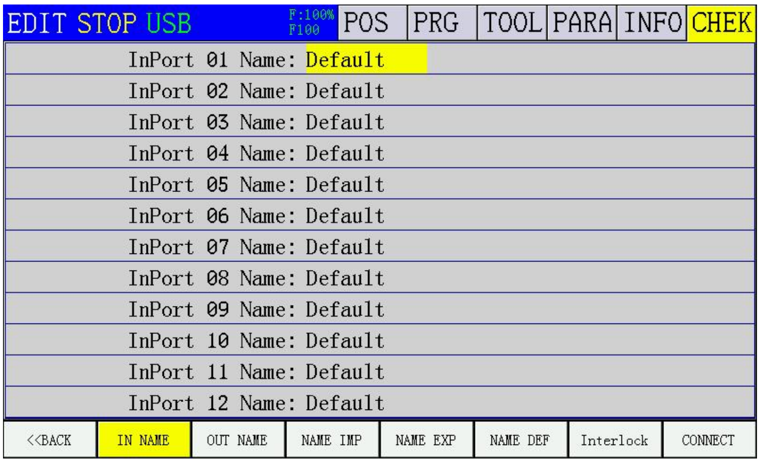
The trial setting requires [manufacturer b] level authority, and the trial time is calculated according to the power on time.The trial is cancelled. A [vendor b] level password is required.Please bear in mind that if you forget the B-level password, the manufacturer can't decrypt it.If you are a user, please contact the equipment manufacturer when the time is up. There is no back door for this function, and the manufacturer can't decrypt it.

1.3.15 diagnosis

Diagnosis can view and test the status of each port, and the port function can be changed in the editing state and parameter switch on state.

1.3.16 General port naming

This system supports port naming. Note: naming will be displayed only when the port is general.



You can import the required ports through a CSV file.You can't export it first and import it after modification.

Example format: input port of the first column: pin output port: pout.

The second column port number.

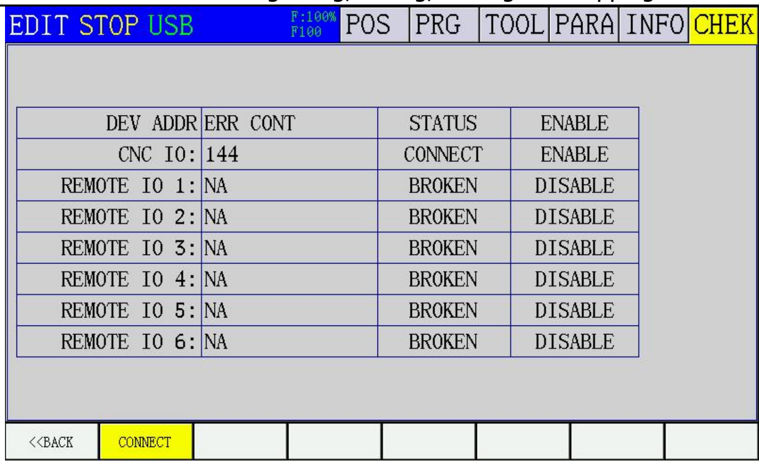
The third column name.It can be up to eight characters or four letters.

PIN	1	Input port 1
PIN	2	Input port 2
POUT	10	Output port 10

Click named import. The input and output are imported together. Only one file is needed.

Note: the user-defined name will not change with the switch between Chinese and English. To restore to the factory value, press [naming default]

1.3.17 communication diagnosis



DEV ADDR	ERR CONT	STATUS	ENABLE
CNC IO:	144	CONNECT	ENABLE
REMOTE IO 1:	NA	BROKEN	DISABLE
REMOTE IO 2:	NA	BROKEN	DISABLE
REMOTE IO 3:	NA	BROKEN	DISABLE
REMOTE IO 4:	NA	BROKEN	DISABLE
REMOTE IO 5:	NA	BROKEN	DISABLE
REMOTE IO 6:	NA	BROKEN	DISABLE

The expansion board enables and forbids the operation in the comprehensive parameters.

Chapter 2 basic operation

2.1 return to zero

Return to zero operation refers to return to mechanical zero.Press [return to zero] to switch to return to zero mode, and press the axis direction button to start returning to zero.

In front of each axis display, for example, if there is a ● mark on the zero point and a zero mark on the zero point, neither indicates that there is no zero return. If the driver alarms, the zero return flag will be cancelled.

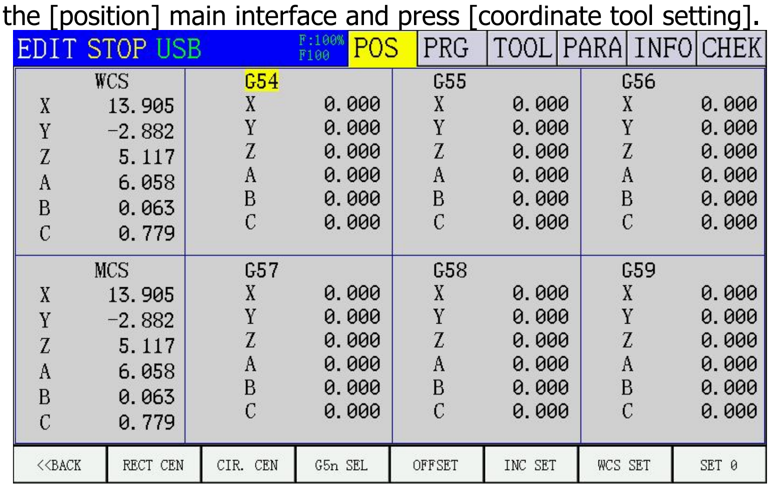
2.2 tool setting

2.2.1 tool setting in coordinate system

Before machining, the controller must set the tool, so that the machine tool corresponds to the position of the workpiece, and the coordinates in the program are based on the absolute coordinates.Machine tool coordinates are only used for limit, tool change point and other special purposes.

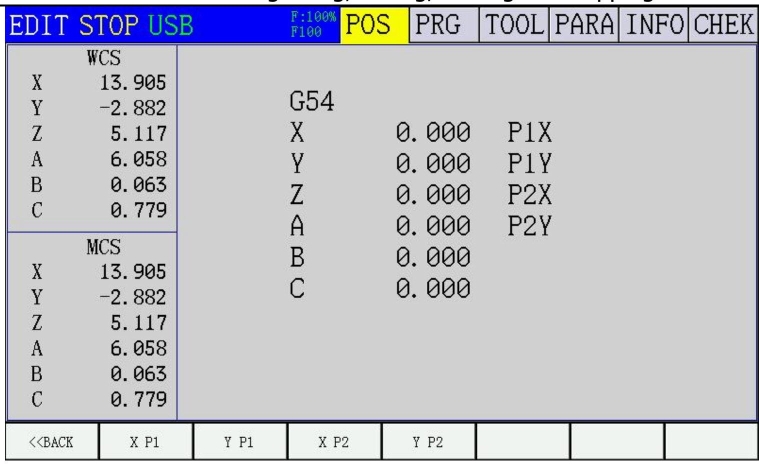
Tool setting in coordinate system refers to the deviation of 6 coordinate systems in machine coordinate system from g54 to G59.

Method: switch to the [position] main interface and press [coordinate tool setting].



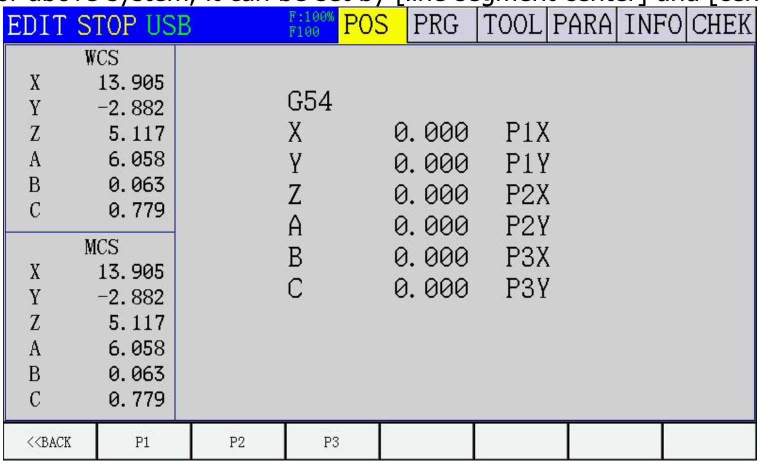
Press [set 0], select the axis, and see whether the [absolute coordinate] changes to 0. Changing zero indicates that in the selected coordinate system, the current position is 0 and tool setting is completed.G54, G55 ~ G59 commands are used for coordinate system switching.

If it is a system with two or more axes, it can be divided into two sections.



Take two points xP1 and xp2 on the X axis, and the system will automatically set the absolute coordinate 0 of the intermediate point. Note that it is not the current point coordinate, the tool will not automatically move to the end point to prevent tool collision.

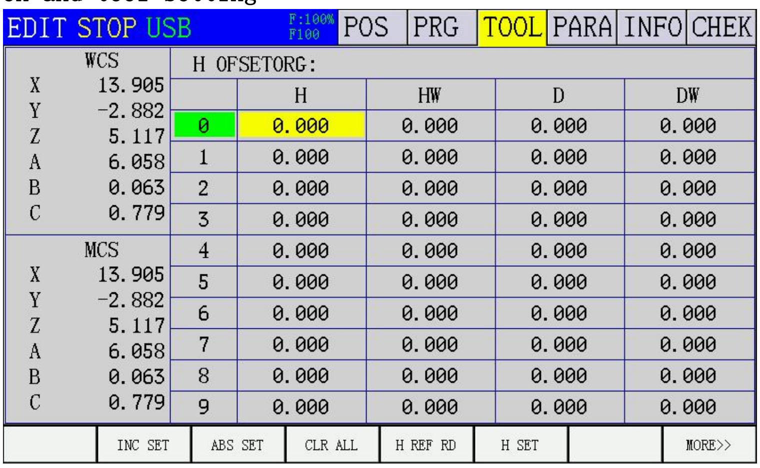
If it is a three axis or above system, it can be set by [line segment center] and [center coordinates].



Take three points on the circle in XY plane: P1, P2, P3.Three points should not be collinear. It is better to keep the three points evenly on the circle (about 120 degrees apart).

After taking three points, the system automatically sets the center of circle x0 and Y0.The current tool end point will not be prevented from moving automatically.

2.2.2 tool compensation and tool setting



Each axis of the system has 0 ~ 99 tool compensation numbers, of which 0 tool compensation is fixed and cannot be set.

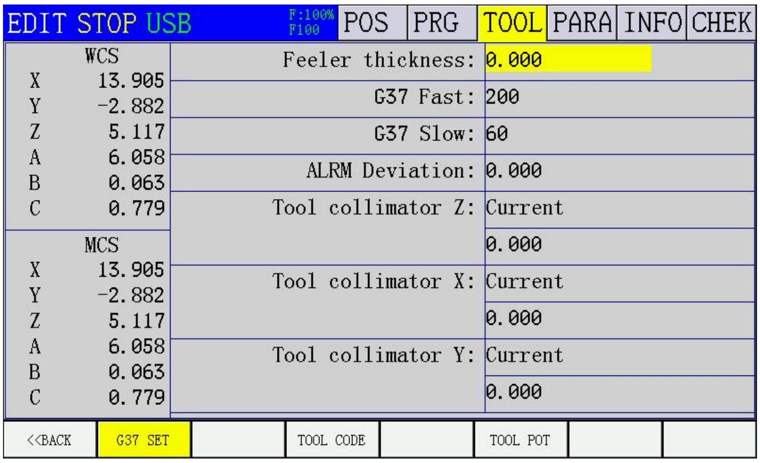
[benchmark tool setting] Application: first use the reference tool to a fixed point, [reference read in], and then change the tool to be compensated to the same point, [benchmark tool setting], then the difference between the tool and the reference tool will be calculated automatically.

After modification, the cutter compensation will not take effect immediately, and it must be run again with H command.

2.3 tool setting instrument

2.3.1 tool setting

First, connect the tool setting instrument signal to the input port X1 ~ X8, and set it as [tool setting device].And set the tool setting parameters in [tool] → [more].Hz must not be zero before tool setting, because the tool length will be written into z-axis cutter compensation according to z-axis tool complement number.

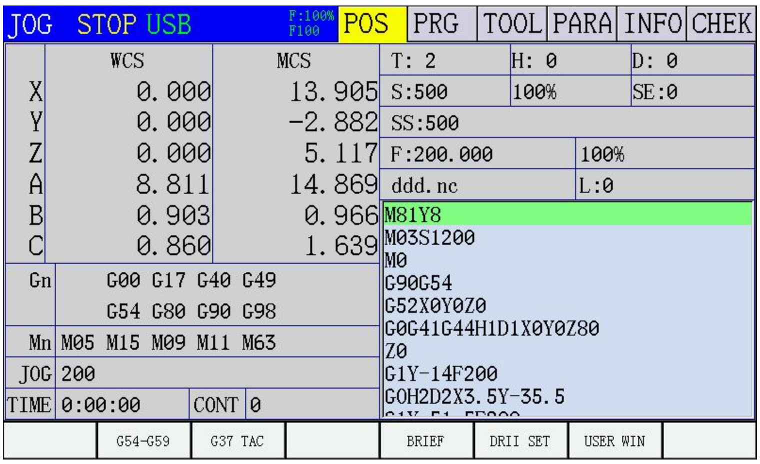


Tool setting block thickness: the distance between the tool setting instrument surface and the workpiece surface. This can be set quickly, which will be discussed later.
Fast automatic tool setting: Z axis is close to the speed of tool setting instrument.
Low speed of automatic tool setting: when the tool contacts the tool setting instrument, the disconnection point is the reading point of the tool setting coordinate, and the speed is below 60 to ensure the accuracy.
Alarm deviation: the manual mode is invalid, and the setting of 0 is invalid. The current tool complement number is not checked by the tool setting instrument, and it is also invalid.In the program, G37 is used for tool setting. If the difference between the current tool length and the last tool length is greater than the alarm deviation, the system will give an alarm.Note: This is based on the complement number of the knife.In other words, if the tool supplement number is HZ2, the system will compare the position of HZ2 last time.Therefore, each knife is supplemented with its own Z knife.
Z axis of tool setting point: fixed, that is to set the coordinate position of z-axis machine tool before tool setting. Currently, the z-axis approaches the tool setting instrument from the current position.
The same is true for XY, where the fixed point goes first to the Z axis and then to the XY axis.
This fixing point must be directly above the tool setting instrument.The direction of Z axis approaching the tool setting instrument can only be close to the negative direction.
After the tool setting is completed, the Z axis will return to the starting point of tool setting.And write the tool length into Hz to specify the tool compensation number, and update the tool compensation at the same time. (note that if the tool compensation length is changed in other ways, the tool compensation number command such as Hz1 can take effect).

2.3.2 manual tool setting of tool setting instrument

If [Z] is set in [Z] manual tool setting interface, make sure that the tool return position is set in the [Z] interface.Note that Hz cannot be 0 (use MDI to execute a Hz1 if it is 0). Press [auto tool setting] to start tool setting.

After the end of tool setting, [tool setting thickness] will appear after the automatic tool setting.



Set the tool block thickness: do not switch mode (can switch to handwheel or single step) and interface.Move the tool to the z-axis reference point of the workpiece, press [tool setting thickness], and the system will automatically set the [tool setting block thickness].The absolute coordinate of Z axis of workpiece reference point is set to 0

If G37 is used to set the tool automatically in the program, the [tool setting thickness] will not appear.

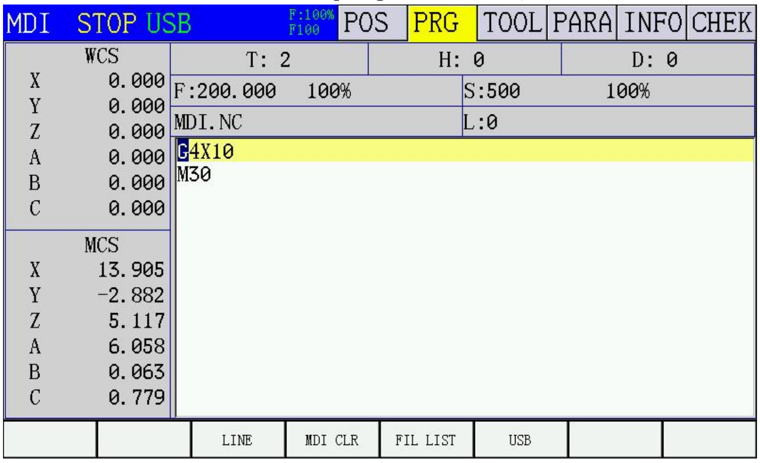
Chapter 3 Automatic Operation

3.1 MDI multi segment operation

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 operation is suitable for simple test operation, please do not use it for workpiece machining.

Operation steps

1. Press [program] and [MDI] to enter the program screen, as shown in the figure below.



2. Press the cursor key [↑] or [reset] to move the cursor to the beginning of the program.
3. Press the [start] key to start MDI operation.

Relevant explanation

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.

3.2 automatic drilling

The upper left corner of the screen displays [drilling], and the [auto] button light flashes. Press the [start] key to start the [drilling function].

3.3 trial processing of handwheel

In automatic mode, open the handwheel for trial machining in [brief display] and [program editing].Shake the hand wheel and the program runs.The handwheel stops and the program stops.

3.4 single section, skip section, selective stop

Single segment: execution of one line pause.

Skip segment: if you turn on the skip, the line beginning with '/' will be ignored. If the skip is closed, the line will be executed.Note that if you start with a '/ /', it is equivalent to a comment, and closing is ignoredIs a comment and "()" is a comment.

Select Stop: when the program executes [M01], if [Select stop] is open, the program stops and [Select stop] is closed, the program continues to run.A program is a continuous loop execution, but hope to complete a complete cycle can be manually stopped.Add M01 in front of the program cycle instruction, so that if you want to stop, you can turn on [Select stop].

3.5 subroutine calls M98

Subroutine call format M98 p1234 L1.L times, can not write.1234, subroutine name.Many people think it is written in the main program n1234, or in the main program after the o1234 write sub program.no, it isn't.1234 must be a separate program in the local directory, the name is o1234NC。Must be O + 4 digits, m98p1, then subroutine name is o0001NC。Note that the first word is because of the letter "OPQ" O, create a new file in the local directory, input the name O + 4 digits.Then edit the file to end with M99, and then edit the main program.

Routine: Call Subroutine delay 1 second.

Main program o1111NC

```
M98 P9123
M30

Subroutine o9123NC
G04X1.0
M99
```

3.6 program cycle execution

Macro program can be used in the program, if the whole program loop, you can use M99 at the end of the program.If you want to cycle a limited number of times, m99l10, cycle 10 times.

The subroutine is in the main program.Sometimes for convenience, you can put the subroutine behind the main program and call it with M98 qxxx.

```
O1111.NC
M98 Q1234
M30

N1234 / / the subroutine starts by giving the name of the subroutine with n.
G4 X1
M99
```

Chapter 4 programming instructions

Introduction to the first chapter of programming

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

1.1 absolute value instruction

The absolute value command is programmed using the coordinate value of the end position of the axis movement.That is, the coordinate position of the tool moving to the end point.As shown in Figure 2-1

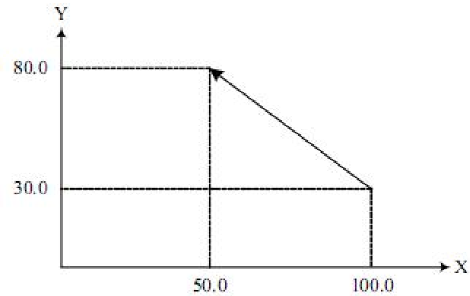


Figure 2-1 diagram of absolute value instruction and increment value instruction

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	1~6
Axis control 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 instruction, the decimal point can be in millimeters, degrees or seconds.

Note that x1.0.000 is different from other controllers.

Chapter 2 composition of procedure

2.1 procedure

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

2.1.1 program number

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

format

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

2.1.2 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 symbol. 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.

2.1.3 skip optional block

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 trip switch is on, the n101 block is skipped.

2.1.4 words and addresses

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

format

```
X1000
10: 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
Program number	O	Program number
Sequence number	N	Sequence number
Preparation function	G	Specify action state (line, arc, etc.)
Size words	X Y Z A B C U V W H	Axis movement command
	R	arc radius
	I J K	The coordinates of arc center and the middle point of G12 arc.
Feed rate	F	Feed rate designation
Spindle function	S, SS	Spindle speed designation, s spindle 1, SS spindle 2
Tool function	T	Designation of tool number
Auxiliary function	M	Machine tool auxiliary function designation
Offset number	H, HX, HY,HZ,HA,HB,HC	The offset number of each axis cutter compensation is specified, h and Hz are consistent.
suspend	P/X	Pause for a specified time
Assignment of subroutine sequence number	P	Specifies the sequence number of the subroutine
Number of repetitions	L	Number of repetitions of subroutines
parameter	P/Q/R	Fixed cycle parameters

2.1.5 base address and instruction value range

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

function	address	Mm input
Program number	O	1～9999
Sequence number	N	unlimited
Preparation function	G	0～99
Size words	X Y Z A B C U V W I J K Q R	±999999.999
Feed per minute	F	0.001～15000.0
Spindle function	S	0～9999
Auxiliary function	M	0～99
suspend	X P	0～999999.999S

The subroutine number is specified, Number of repetitions	P	1～9999
Number of repetitions	L	1～99999
Offset number	H，D	0～99

These parameters are the command range of CNC system, and have nothing to do with the actual working range of the machine tool.For example, the system can specify that the axis movement is about 100m, while the actual X-axis travel of the machine tool may be only 2m.When writing the program, you should 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:
G00X__;(G00 valid)
Y__;   (G00 valid)
G01Z__;(G01)
X__;   (G01)
```

3.1 G code list

G code	level	function
G00 *	1	Fast positioning, speed according to speed parameter G0 speed
G01		Linear interpolation, run according to the given F
G02		Clockwise circular arc interpolation, the speed is given F, if there is a non planar axis, according to the spiral interpolation operation
G03		Anti clockwise arc interpolation, the others are the same as above
G12		Circular interpolation through intermediate point
G04	0	Delay waiting, parameter x in seconds, parameter P in milliseconds, resolution of 5 milliseconds
G17 *	2	Circle interpolation plane selection XY
G18		Circle interpolation plane selection ZX
G19		Circle interpolation plane selection YZ
G28	0	Return to the parameter point (return to mechanical zero), involving parameters such as speed, return to zero direction and mode
G30		Return to the second and third reference point through the middle point
G31		Jump mechanism, if G31 input port effectively stops the current movement to the next instruction
G50	0	When the side position moves, the input port stops invalid, similar to G31, but the input port can be specified arbitrarily
G51		When the side position moves, the input port stops effectively, similar to G31, but the input port can be specified arbitrarily
G37		Z-axis automatic tool setting
G22	0	Loop instruction
G23		Loop instruction
G52	0	Local coordinate function
G53		Coordinate positioning of machine tool
G40*	3	Nose radius compensation cancelled
G41		Left nose radius compensation
G42		Right nose radius compensation
G43	4	Tool length offset in positive direction
G44		Tool length offset in negative direction
G49*		Tool length offset cancelled
G54 *	5	Workpiece 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	6	High speed deep hole machining cycle
G74		Back tapping cycle, tapping by encoder.
G80 *		Fixed 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		User defined drilling instructions, specific actions in the drilling function G88 editing.
G89		Boring cycle
G70	0	Drilling cycle of wheel circumference (for flange drilling)
G71		Circular arc drilling cycle (for flange drilling)
G72		Drilling along an angle
G90 *	7	Absolute value programming
G91		Incremental value programming
G92	0	When the workpiece is set to zero, there is no offset in the workpiece position
G93		Set the coordinates of the machine tool. If there is a soft limit, please use it carefully.
G98 *	8	Fixed loop returns to the initial plane
G99		Fixed loop return to R point

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., indicating the combination of any axis. The absolute value command is the end coordinate value of tool movement, and the increment value instruction is the tool movement amount.
Semicolon (;):Indicates the end of the segment.

Instructions

Linear interpolation positioning
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.

notes
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.

Instruction format

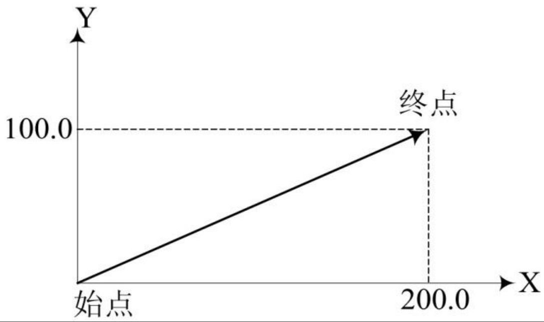
G01 IP__ F__ ;
IP: the absolute command is the end coordinate value of tool movement, and the incremental instruction is the tool movement amount.
F: tool feed rate.

Instructions

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

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

give an example




```
G91 G01 X200.0 Y100.0 F200.0
The tool moves from the starting point (0,0) to the end point (200.0100.0) at a speed of
200 mm / 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.

Instruction format

Arc of XY plane

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

Circular arc of ZX plane

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

Circular arc of YZ plane

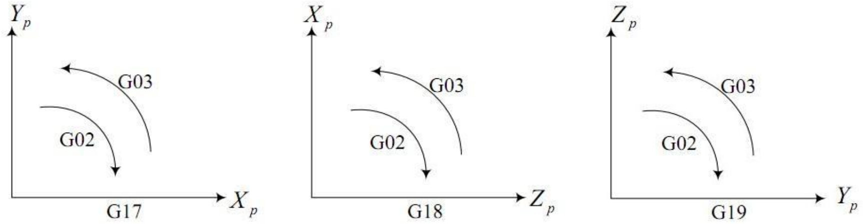
$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\_ \; Z\_\_ \quad \begin{Bmatrix} R\_\_ \\ J\_\_ \; K\_\_ \end{Bmatrix} \quad F\_\_$$

project	Specified content	command	describe
1	Plane assignment	G17	XY plane arc assignment
		G18	ZX plane arc assignment
		G19	YZ plane arc assignment
2	Interpolation direction	G02	Clockwise arc interpolation (CW)
		G03	Counterclockwise arc interpolation (CW)
3	The position or distance of an end point	Two axes in X, y, Z	The position of the end point in the absolute coordinate system
		U V W or Two axes in XYZ under G91	The distance from the starting point coordinate to the ending point coordinate
4	Center position or radius	Two axes in I J K	Distance from the starting point coordinate to the center coordinate (I J K is also calculated in increments in absolute mode)
		R	arc radius
5	Feed rate	F	Tangent speed of arc feed

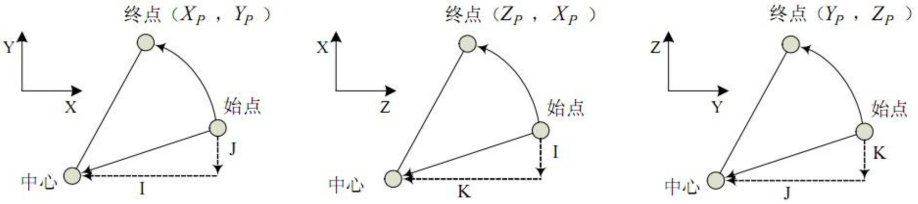
Instructions

Direction of arc interpolation

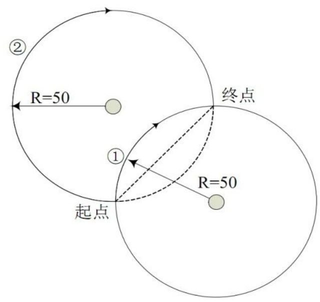
The so-called clockwise (G02) and counter clockwise (G03) means that in the right-hand rectangular coordinate system, for X_Y_The plane (Z - x - plane, y - Z - plane) from Z_The positive direction of the axis (Y axis, X axis) is shown in the following figure



The amount of movement on the arc
With the address x \Y_Or Z_Specify the end point of the arc. The absolute value is indicated under the G90 command, and the incremental value is indicated under the G91 command.The increment value is the distance from the start point to the end point of the arc.
Center of arc
The center of the arc is specified by the addresses I, J, and K, which correspond to X - u, X - u, and K, respectivelyY_、Z_、。1. The value after J and K is the vector component from the starting point of the arc to the center of the circle, and is the increment value with sign.As shown in the figure below:



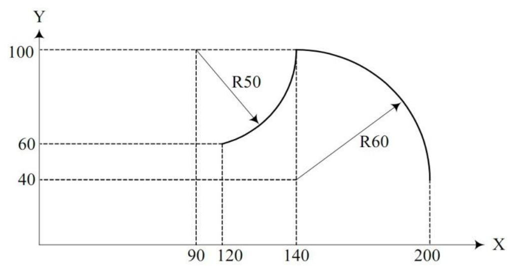
Arc radius



① When the arc is less than 180 ℃, execute G code G91 G02 X60 Y50 R50 F300;
② When the arc 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 trajectories on the graph are programmed with 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 ;
or
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300.0 ;
G02 X120.0 Y60.0 R50.0 ;
Incremental mode
G91 G03 X-60.0 Y60.0 I-60.0 F300.0 ;
G02 X-20.0 Y-40.0 I-50.0 ;
or
G91 G03 X-60.0 Y60.0 R60.0 F300.0 ;
G02 X-20.0 Y-40.0 R50.0 ;
```

Spiral interpolation

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

Instruction format

Arc of XY plane

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\} X\_\ Y\_\ Z\_ \left\{\begin{matrix}R\_\\I\_\ J\_\end{matrix}\right. F\_$$

Circular arc of ZX plane

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\} X\_\ Z\_\ Y\_ \left\{\begin{matrix}R\_\\I\_K\_\end{matrix}\right. F\_$$

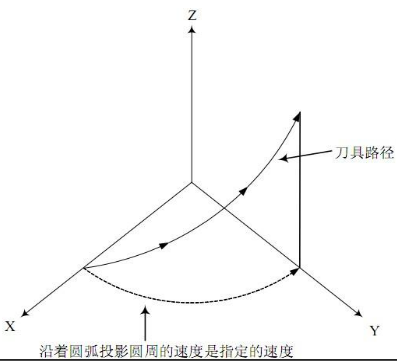
Circular arc of YZ plane

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\} Y\_\ Z\_\ X\_ \left\{\begin{matrix}R\_\\J\_\ K\_\end{matrix}\right. F\_$$

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:

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



> notes
> 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
Circular arc of ZX plane
G12 I K X Z
Circular arc of YZ plane
G12 J K Y Z
```

1. J and K are the middle point of the arc, absolute under G90 and relative under G91.
10. Y and Z are the end points of the arc, absolute at G90 and relative at G91.
This instruction is mainly used for teaching programming.
It is better to take the middle point of the arc to reduce the calculation error. Note that the starting point, middle point and end point cannot be collinear.

3.6 G04 - delay waiting

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

Instruction format

```
G04 X__ ;
or
G04 P__ ;
10: Delay waiting time setting (decimal can be used).
P: Delay waiting time setting (decimal is not allowed).
```

Instructions

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

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

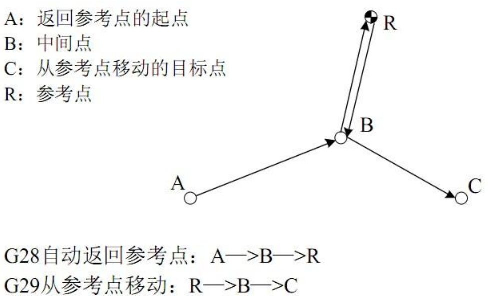
notes
The 1 x / P command unit is independent of the minimum unit of the linear axis or the axis of rotation.
If P and X instructions are omitted, it can be regarded as accurate stop.
3 execution of G04 instruction will automatically disable read ahead and buffering.

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.



Command format

G28 IP__;
G28: automatic return to reference point instruction.
IP__:The coordinate of the intermediate point passed by when automatically returning to the reference point, specified by absolute or incremental value.

Instructions

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

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.

There are several axes in G28, and the execution order is Z - > xyabc

example
N1 G28 X40.0 ;Middle point (40.0)
N2 G28 Y60.0 ;Intermediate point (40.0, 60.0)

notes
1 after power on, if the manual return to the reference point is not carried out once, the movement from the intermediate point to the reference point is the same as when returning to the reference point manually when G28 is commanded.
When changing the workpiece coordinate system, the intermediate point will also move to the new coordinate system.

3.7.2 G30 - return to second and third reference point

Instruction format

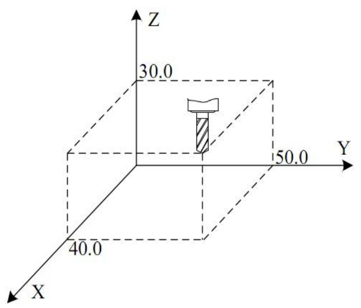
G30 IP__P_;
G30: automatic return to reference point command.
IP__:The coordinate of the intermediate point passed by when automatically returning to the reference point, specified by absolute or incremental value.
P_ 2 or 3

Routine: X axis returns to the second reference point: g91g30x0.X axis returns to the third reference point: g91g30x0p3

The position of the axis in the reference point.

3.8 coordinate system function

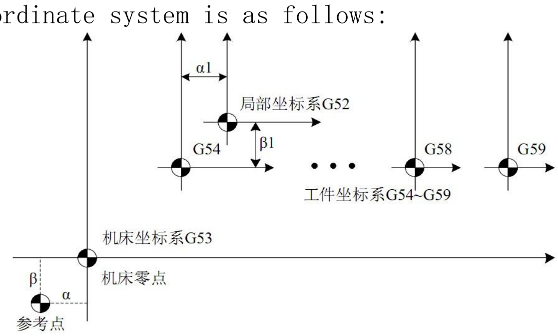
When the machine tool is working, the tool moves to the specified position according to the coordinate specified by the machining program, and the coordinate value is specified by the axis components of the coordinate axis.As shown in the figure below is the tool position specified by x40.0 y50.0 z30.0.



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

The zero point of the machine tool is a fixed reference point set by the machine tool manufacturer for the machine tool.The coordinate system with the zero point of the machine tool as the coordinate origin is called the machine 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.
The workpiece coordinate system is based on the sub coordinate system of the machine tool coordinate system, and its position in the machine coordinate system can be set and changed.
The coordinates of the workpiece in its local coordinate system and 3 can be changed based on the workpiece coordinates.
The relationship of each coordinate system is as follows:



Usually, after the system is started, the user needs to reset the machine coordinate system.
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__; IP__:Absolute coordinates of target point in machine coordinate system

Instructions

Because 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 tool change position), it is more convenient to use g53 command.

2 g53 is a modeless G code, which is valid only in the current program segment.

The 3 g53 command must be absolute.If it is an incremental command, an alarm is 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

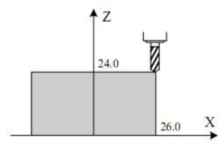
Instruction format

G92 IP__ ; IP__:The coordinates of the specified point in the current coordinate system.

The workpiece coordinate system is established by making the specified coordinate value (IP \It becomes the absolute coordinate value of the point on the current tool (such as the tool tip) in the set workpiece coordinate system.
1. Under the condition of tool length compensation, when G92 is used to set the coordinate system, the specified coordinate value (ip_uuuuuuuuuuu) is setIs the position before tool compensation.

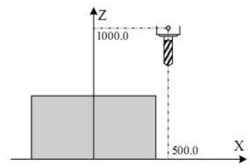
2 for tool radius compensation, the compensation will disappear temporarily when G92 command is used.
Generally, please set the workpiece coordinate system before specifying tool compensation.
give an example

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 command reference point moves to the designated position, the tool length compensation must be added. The compensation value is the difference between the reference point and the tool tip.

3.8.2.2 automatic setting of workpiece coordinate system

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

G92 XαYβZγ;

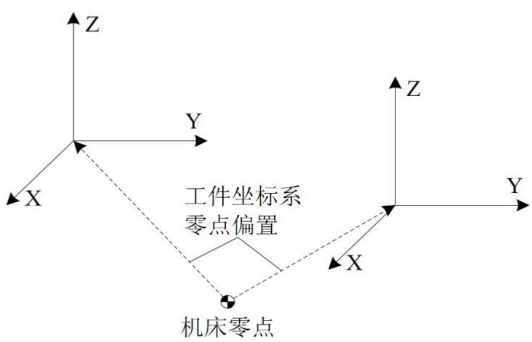
3.8.2.3 select workpiece coordinate system (G54-G59)

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

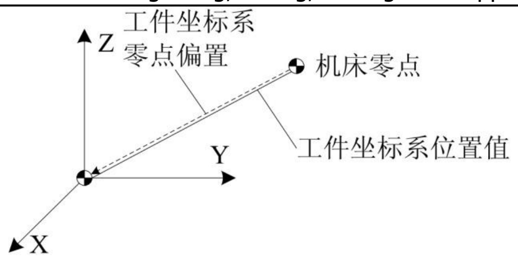
G54	Workpiece system 1
G55	Workpiece coordinate system 2
G56	Workpiece coordinate system 3
G57	Workpiece coordinate system 4
G58	Workpiece coordinate system 5
G59	Workpiece coordinate system 6

Instructions

The six workpiece coordinate systems are set according to the distance (workpiece zero offset) from the machine zero point to the respective coordinate system zero point, as shown in the figure below.



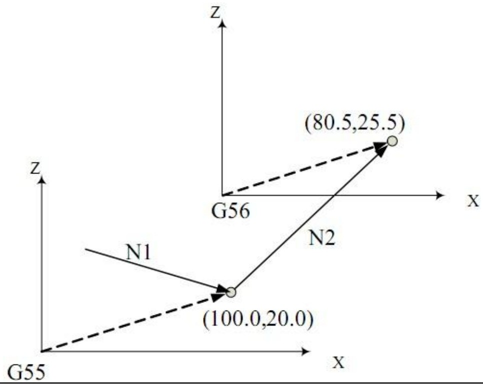
After the zero point is offset, the zero point is returned as shown in the figure below.



3 if the working coordinate system is selected, it is generally set to zero in the parameter. If the setting value is not zero, after returning to the parameter reference point, all workpiece coordinate systems will offset the set value of the parameter.

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



Processing program N10 G55 G00 X100.0 Z20.0 ; N20 G56 X80.5 Z25.5 ;

3.8.3 move workpiece coordinate system with G92

Instruction format

G92 IP_ ; IP__:Specifies 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.

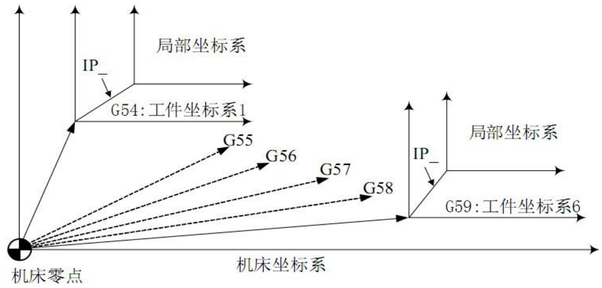
Instruction format

G52 IP_ ;Set local coordinate system …… G52 IP0;Cancel 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.



notes When an axis returns to the reference point automatically or manually, the zero point of the local coordinate system of the axis is consistent with the zero point of the workpiece coordinate system, that is, the local coordinate system is cancelled.This is in line with the instruction G52α; (α:Return to the axis of the reference point). 2. The local coordinate system does not change the workpiece coordinate system and the machine coordinate system. It depends on whether the coordinate of 3 is reset or not. When G92 is used to set the workpiece coordinate system, the local coordinate system is cancelled.If the coordinate values of all axes are not commanded, the local coordinate system of the axis without coordinate values is not cancelled, but remains unchanged. 5 G52 temporarily cancels tool radius compensation. After the execution of 6 G52 program segment, the absolute coordinates immediately display the coordinates in the local coordinate system.

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 invariant (ZX plane)

In addition, 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 simplify 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
G73	Intermittent feed	——	Rapid feed	High speed deep hole machining cycle
G74	Cutting feed	Spindle forward rotation	Cutting feed	Counter tapping cycle
G80	——	——	——	Cancel fixed 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	——	custom	drill
G89	Cutting feed	——	Cutting feed	Boring

Instruction format

G__ IP__ R__ Q__ P__ F__J__L__;

Address description

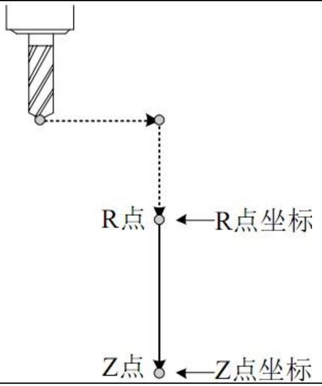
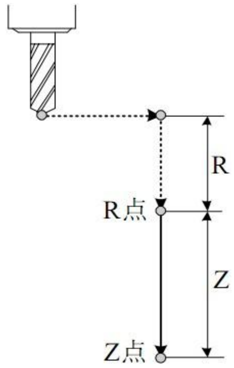
Specified content	address	explain
Hole processing method	G	Fixed cycles G73, G74, g80 ~ g89 were selected.
Hole position data	Direction axis address of non hole machining in IP	The position of the hole is specified with absolute value or increment value. The control is the same as that of G00 positioning. Example x100y100
Hole machining data	IP medium hole processing direction address	Or use the absolute value of R from the bottom of the hole as shown in the figure below.In action 3, the feed rate is the speed specified by F. in action 5, according to the different hole processing methods, it is the speed of rapid feed or command with F code.For example, Z-20
	R	As shown in the following figure, the distance from the initial point plane to the R point is specified with an increment value, or the coordinate value of the R point is specified with an absolute value.The feed rate is rapid feed in both actions 2 and 6.
	Q	Specify the amount of each cut in G73 and g83, or the translation amount (increment) 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 of G04.
	F	Specifies the cutting feed rate.G74, g84 pitch designation.
	J	Machining axis assignment, J0: X, J1: y, J2: Z,

		J3: A, J4: B, J5: C, other values or no default Z axis is specified
	L	Go to the position of L70G71 is used. It's used to bypass certain points.The number of cycles in G91 mode: g91x10l10.

Instruction interpretation

Absolute programming and relative programming

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

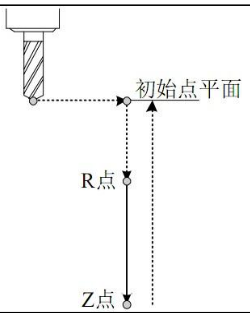
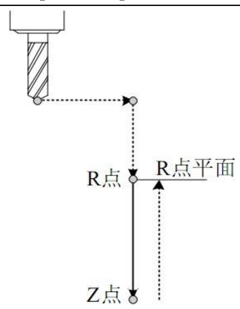
G90 (absolute value instruction)	G91 (increment value instruction)
	

Return point plane

① The instruction g98 returns to the initial point plane.

② The instruction G99 returns the r-point plane.

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

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

Hole processing method

The optional fixed cycle instructions for hole machining include: G73, G74, G76, g80 ~ g89, all of which are modal g codes.

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

Once the hole processing method and data are instructed, they remain valid until the G codes (g80 and 01 group G codes) for canceling the fixed cycle are specified. Therefore, when the same hole processing is carried out continuously, it is not necessary to specify the hole processing method and data in each program segment.At the beginning of the fixed cycle, all the necessary hole machining data are assigned, and in the subsequent fixed cycle, only the changed data need to be specified.

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

The cancellation of fixed cycle

Fixed loop can be cancelled by using group 01 code or g80 in the same group as fixed loop.01 group G code includes: G00, G01, G02, G03.

3.9.2 G73 - high speed deep hole processing cycle

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

Instruction format

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

For example, G73 x10y10z-20r2q5f100j2 drilled a hole with a depth of - 20 at position x10y10.R2: feed from Z2, Q5: 5 per feedJ2: machining axis Z axis, the cutter is installed on the Z axis.

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_ Q_ F_ J_; IP_ (non hole machining shaft) hole position data IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value) R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value) Q_:For pecking tapping, each tapping (spindle in (m29) position mode is effective), and back off with g73d F_:Metric pitch.Value range: 0.001 ~ 500.00mm J_:Machining shaft

Case G74 z-20r2f1.0;Z-20: hole bottom coordinate - 20 R2: start tapping from coordinate 2, leaving a distance of 2mm.F1.0: pitch 1.0 mm

Interpolation tapping m29s1000 / / it is still necessary to set the spindle speed, and the interpolation speed is determined by the spindle speed

G74 Z-20R2Q5F1.0 ; 5mm each time

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

For example, g81 x10y10z-20r2f100j2 drilled a hole with a depth of - 20 at position x10y10.R2: feed from Z2, J2: machining axis Z axis, the cutter is installed on Z axis.

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.;  Locate, drill 1 hole and return to R
X150.; Locate, drill 2 holes and return to R
G98 Y150.;     Locate, drill 3 holes and return to initial plane
G80 G00 X0. Y0. Z0.; Cancel the fixed 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_ (non hole machining shaft) hole position data
IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
P_:Hole bottom pause time (unit: 0.001 s)
F_:feed rate
J_:Machining shaft

For example, g82 x10y10z-20r2f100j2 drilled a hole with a depth of - 20 at position x10y10.R2: feed from Z2, J2: machining axis Z axis, the cutter is installed on Z axis.

Instruction interpretation

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.

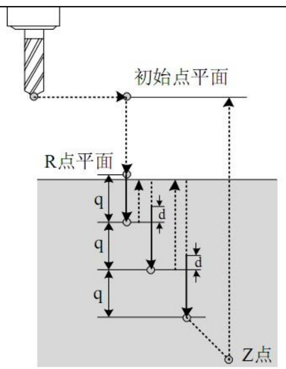
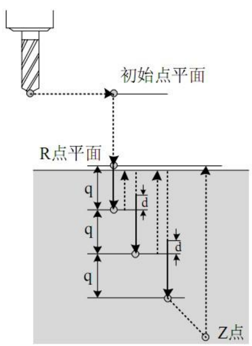
3.9.6 g83 - deep hole machining cycle

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

For example, g83 x10y10z-20r2f100j2 drilled a hole with a depth of - 20 at position x10y10.R2: feed from Z2, J2: machining axis Z axis, the cutter is installed on Z axis.

G83(G98)	G83(G99)
	

Instruction interpretation

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.

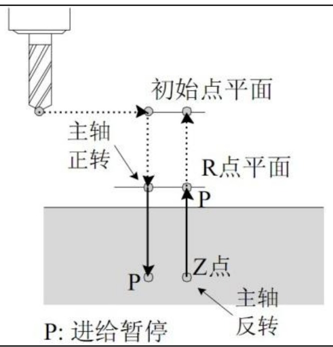
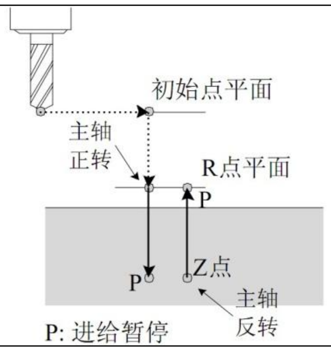
Instruction format

G84 IP_ R_ Q_ F_ J_; IP_ (non hole machining shaft) hole position data IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value) R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value) Q_:For pecking tapping, each tapping (spindle in (m29) position mode is effective), and back off with g73d F_:Metric pitch.Value range: 0.001 ~ 500.00mm J_:Machining shaft

Example g84 z-20r2f1.0 Z-20: hole bottom coordinate - 20 R2: start tapping from coordinate 2, leaving a distance of 2mm.F1.0: pitch 1.0m

Interpolation tapping m29s1000 / / it is still necessary to set the spindle speed, and the interpolation speed is determined by the spindle speed

G84 Z-20 R2 Q5 F1.0 ; Each time the attack is 5mm, this mode does not refer to the encoder, so the encoder can not be connected

G84(G98)	G84(G99)
	

Instruction interpretation

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_ (non hole machining shaft) hole position data
IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
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_ (non hole machining shaft) hole position data
IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
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_ (non hole machining shaft) hole position data
IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
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_ (non hole machining shaft) hole position data
IP_ (hole machining axis) distance from R point to hole bottom (increment value) or coordinate of hole bottom (absolute value)
R_:The distance from the plane of the initial point to the R point (incremental value), or the coordinates of the R point (absolute value)
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.

3.9.13 circular drilling of G70 wheel (Group 00)

Format G70 I_J_ L_

I radius (+ CCW / - CW)

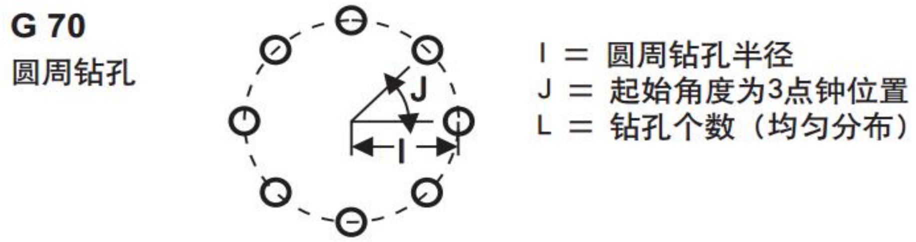
J starting angle (0 to 360.0 degrees counter clockwise from horizontal position; 0 at 3 o'clock position)

L number of holes evenly distributed on the circumference

This modeless G code must be associated with one of cycles G73, G74, or g81 - g89

Together.Either a tapping function must be activated in each position.

Note: the circle is centered on the current xy and returns to the center after machining.The machining must be the z-axis.



Example G0 Z50

    G0 X0 Y0 ;Locate to the center of the circle.

G83 Z-20R2Q5F100L0 ;Activate g83, enable l0, set drilling parameters, but do not drill (at the center of the circle at this time)

G70 I20 J0 L6 ;Radius 20, drilling 6 holes evenly.

    G80 ;Cancel drilling.The above program plus spindle cooling, etc., can facilitate the processing of flange.

3.9.14 circular arc drilling of G71 wheel (Group 00)

Format G70 I_J_K_L_

I radius (+ CCW / - CW)

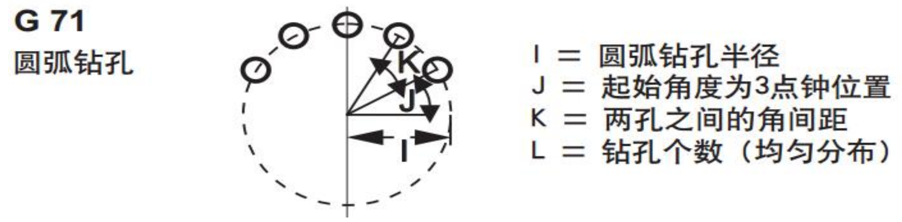
J starting angle (angle counter clockwise from horizontal position)

Angle spacing of K holes (ten or one angle)

L number of boreholes

This modeless G code is similar to the G70, but it is not limited to a full circle.G71 belongs to group 00, so it is Non modal.A cycle must be activated so that drilling or tapping functions can be performed at each location.

Note: the circle is centered on the current xy and returns to the center after machining.The machining must be the z-axis.



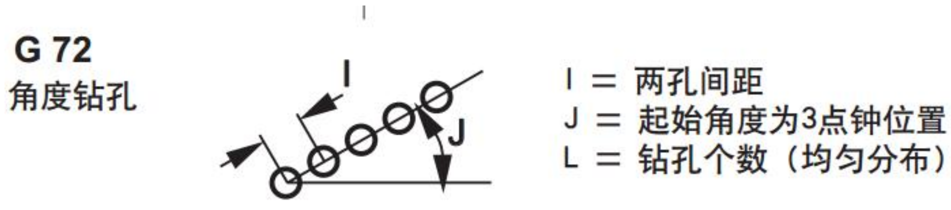
Refer to G70

3.9.15 G72 drilling along an angle (Group 00)

72 GI format_J_ L_

I distance between holes (+ CCW / - CW)

Angle of J line (angle counter clockwise from horizontal position)
L number of boreholes
This modeless G code drills "L" holes in a straight line at a specified angle.It operates like the G70.
For the G72 to work properly, a cycle must be activated so that drilling or tapping functions can be performed at each location
Note: the starting point is the current XY, and it will return to the starting point after processing.The machining must be the z-axis.



3 L of 10

If 10 is used in the fixed cycle, it means that only the position setting parameter is taken this time, and the drilling is not true. It can be used to bypass the workpiece.See G70 for routine

In relative programming mode, the loop can be realized by L
For example:
G81 G99 z-0.5 r0.1 f6.5 (drill a hole in the current position)
G91 x-0.562 L9 (drill 9 holes every 0.562 apart in the negative X direction)

3.11 G22-G23 cycle execution

This instruction pair can realize program loop.

```
      G22L3
      ..
Circulation content
      ..
     G23
```

The intermediate program is executed three times
It can also be nested, but no more than 4 levels.

```
      G22L3
        G22L5
         ..
Circulation content
         ..
        G23
     G23
```

3.12 G31 - jumping function

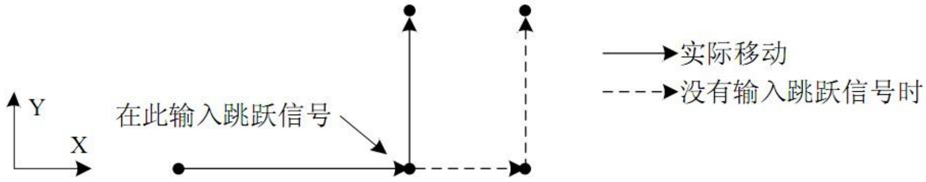
After G31, linear interpolation like G01 can be performed by command axis movement.During the execution of this instruction, if a jump signal is input, the program segment stops the rest and starts to execute the next segment.This function is mainly used to control the end of processing by external signal,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

Instruction format

G31 IP__ F__ ;
G31: jump instruction, non modal, valid only in this block.
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 ○ɔ.



……
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

3.14 G37 automatic tool setting

G37 instruction can use the tool setting instrument to automatically adjust the tool. Note: Hz cannot be 0Because the offset of the tool should be saved in the corresponding supplement number.

Instruction format

```
G37 IP_
IP: move axis.
```

The controller first quickly locates the specified axis, and then moves according to the automatic tool setting process. After positioning, the controller still moves to the fixed point first. After the tool setting is completed, the control automatically updates and uses the new tool deviation.

For example:

HZ1
G37 Z100

3.15 G10 modification of coordinate system and tool compensation

Instruction format

G10 L2 modify coordinate system

```
G10 L2 Pn
L2 specifies to modify the coordinate system
P1 ~ 6, corresponding to g54, G55, g56, G57, g58, G59
```

Case G90 G10 L2 P1 x2;Set g54 X-axis offset to 2
G91 G10 L2 P1 X2 ;Set g54 X-axis offset to increase by 2

Grinder application:

N100 G1 Z0 F100 / / down to the workpiece surface
G1 x-50 / / processing
G10 L2 P1 w-0.1 / / modify the z-axis coordinate system. The next time you arrive at Z0, it will be 0.1 more than this time. Power failure is also effective.
G0 Z10 / / tool lift
G0 x0 / / back to X axis

G10 L10 modifying tool length and tool compensation

G10 L10 Pn R_
L10 specifies the modification of tool length and tool compensation
P1 ~ 99, corresponding to h cutter supplement number
R_ Modify value

G10 cases were g1r2;Set H1 cutter compensation to 2.000. Note that the modification will not take effect immediately

H1 ;Call the No.1 cutter compensation to make the new one effective.

G10 L11 modify tool length tool compensation wear value

G10 L11 Pn R_
L11 modify the wear value of tool length and tool compensation
P1 ~ 99, corresponding to tool complement number
R_ Modify value

For example, G10 L11 P1 R2;Set H1 tool compensation wear 2.000, note that the modification will not take effect immediately

H1 ;Call the No.1 cutter compensation to make the new one effective.

G10 L12 modifying radius cutter compensation

G10 L12 Pn R_
L12 specify modification radius cutter compensation
P1 ~ 99, corresponding to D cutter supplement number
R_ Modify value

For example, G10 L12 P1 R2;Set D1 radius cutter compensation to 2.000. Note that the modification will not take effect immediately

D1 ;Call the No.1 cutter compensation to make the new one effective.

G10 L13 modify radius tool compensation wear value

G10 L13 Pn R_
L13 specify the tool compensation wear value of modified radius
P1 ~ 99, corresponding to tool complement number
R_ Modify value

For example, G10 L13 P1 R2;Set D1 radius tool compensation wear 2.000, note that the modification will not take effect immediately

D1 ;Call the No.1 cutter compensation to make the new one effective.

Chapter 4 Auxiliary Functions (M Code)

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

4.1 Overview

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

name	function
M00	Program pause, press "cycle start" program to continue
M01	Select stop. If the stop light is on, the program will stop
M02	Program stop
M03	Spindle 1 forward rotation
M04	Spindle 1 reverses
M05	Spindle 1 stop
M06	Start the M06 code and wait for completion
M07	Start magazine return to zero code
M08	Coolant on
M09	Coolant off
M10	The chuck is clamped
M11	Chuck released
M13	Spindle 2 forward transmission
M14	Spindle 2 reverse transmission
M15	Spindle 2 stop
M19	Spindle positioning
M20	broach
M21	Loose knife
M30	Program ends, program returns to start
M29	Spindle P / s position mode
M62	Start spindle speed monitoring (encoder required).For example: M62 S1000, if the spindle speed is lower than 1000 rpm in the open state, the program will stop, and encoder support is required. 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	Waiting for input port, output port or auxiliary relay valid example: M71 X12 input port;My171 output port;M71 Z1 auxiliary relay;
M72	Invalid jump of input port, output port or auxiliary relay
M73	Input port, output port or auxiliary relay jump effectively
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
M80	Output port or auxiliary relay closing example: M80 Y12
M81	Example of output port or auxiliary relay: M81 Y12
M82	Output port or auxiliary relay output is closed for a period of time, e.g. M82 Y12 P1000 (MS)
M83	The output port or auxiliary relay output is closed after one input port is valid
M84	Or an auxiliary output of my12 is invalid
M85	The output port or auxiliary relay output waits for an input port to be valid, and does not close Example: M85 Y12 x13
M86	The output port or auxiliary relay output wait for an input port to be invalid and not to be closed Example: M86 Y12 x13
M98	Call the subroutine.Note that the subroutine name format is oxxxxNC, X is a number
M99	Subroutine or macro program return.If used in the main program, the program loops from the beginning

4.2 M code description

4.2.1 moo program suspension

Instruction format

M00 (or M0);

Command function

After executing the M00 command, the program stops running, and the word "pause" is displayed. After pressing the cycle start key, the program continues to run.

4.2.2 M01 program selective stop

Instruction format

M01 (or M1);

Command function

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

4.2.3 M02 - end of procedure

Instruction format

M02 (or m2);

Command function

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

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

4.2.5 M04 - spindle 1 reversal
Instruction format
M04 (or M4);
Command function
Control spindle 1 to reverse.

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

4.2.7 M08 / M09 - coolant on / off
Instruction format
M08 (or M8);
M9 or M9;
Command function
The M08 command opens the coolant.
The M09 command turns the coolant off.

4.2.8 M10 / M11 - clamping / loosening
Instruction format
M10;
M11;
Command function
The instruction of M10 is clamping.
M11 command is release.

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

4.2.10 M14 - spindle 2 reversal
Instruction format
M14
Command function
Control spindle 2 to reverse.

4.2.11 M15 - spindle 2 stop
Instruction format
M15

Command function

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

4.2.12 M19 - spindle orientation

Instruction format

M19

Command function

Cancel M19 with M05.

4.2.13 M20 / M21 broach, loose knife

Instruction format

M20

M21

Command function

It can't be used in general program, only in t code and M06 code.

4.2.14 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.

4.2.15 m29 - spindle P / s Switching

M29 digital spindle position control.Cancel position control with M05

4.2.16 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.

4.2.17 M63 - cancel speed monitoring

Instruction format

M63

Command function

Cancel speed monitoring

4.2.18 M64 counter plus one

Instruction format

M64

Command function

Add one to the workpiece count.

4.2.19 M65 - counter clear

Instruction format

M65

Command function

The workpiece count value is cleared.

4.2.20 M70 - wait for input port, output port, auxiliary relay invalid

Instruction format

M70 Xxx Pxx Exx; Input port
M70 Yxx Pxx Exx; Output port
M70 Zxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
When the specified port is valid, the program waits.
When the specified port is invalid, the program goes down.
Pxx limit time, Ms.If not, there is no time limit.
If the alarm number exxx is issued.If the time-out and no e, directly next.
Only one port can be specified at a time.
Case M70 x12 P1000 E100;Wait for input port 12 to be invalid. If the time-out is 1 second, alarm No. 100 will be sent.

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

Instruction format

M71 Xxx Pxx Exx; Input port
M71 Yxx Pxx Exx; Output port
M71 Zxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
When the specified port is invalid, the program waits.
When the specified port is valid, the program goes down.
Pxx limit time, Ms.If not, there is no time limit.
If the alarm number exxx is issued.If the time-out and no e, directly next.
Only one port can be specified at a time.
Case M71 x12 P1000 E100;Wait for input port 12 to be valid. If the time-out is 1 second, alarm No. 100 will be sent.

4.2.22 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.

4.2.23 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.

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

Instruction format

M70 Xxx Lxx Pxx Exx; Input port
M70 Yxx Lxx Pxx Exx; Output port
M70 Zxx Lxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
The specified port waits for a valid signal and then an invalid signal.
LXX specified the number of times, not specified 1 time
Pxx limit time, Ms.If not, there is no time limit.
If the alarm number exxx is issued.If the time-out and no e, directly next.
Only one port can be specified at a time.
Case M74 x12l3;Wait for the third falling edge of the input port.

4.2.25 M75 - waiting for input, output, rising edge of auxiliary relay

Instruction format

M71 Xxx Lxx Pxx Exx; Input port
M71 Yxx Lxx Pxx Exx; Output port
M71 Zxx Lxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
The specified port waits for an invalid signal and then a valid signal.
LXX specified the number of times, not specified 1 time
Pxx limit time, Ms.If not, there is no time limit.
If the EXX time-out, send XX alarm.If the time-out and no e, directly next.
Only one port can be specified at a time.
Case M75 X12 L3 L3000 E100;Wait for the third rising edge of the input port.No alarm was detected in 100 seconds.

4.2.26 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.

4.2.27 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.

4.2.28 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.

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

Instruction format

M83 Yxx Xxx Pxx Exx; Output port
M83 Zxx Xxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
Pxx limit time, Ms.If not, there is no time limit.

If the EXX time-out, send XX alarm.If the time-out and no e, directly next.
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.

4.2.30 M84 - output port, auxiliary relay output is closed after one input port is invalid

Instruction format

M84 Yxx Xxx Pxx Exx; Output port
M84 Zxx Xxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
Pxx limit time, Ms.If not, there is no time limit.
If the EXX time-out, send XX alarm.If the time-out and no e, directly next.
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.

4.2.31 M85 - output port, auxiliary relay output waits for an input port to be valid, it will not be closed, and the next paragraph will be executed

Instruction format

M83 Yxx Xxx Pxx Exx; Output port
M83 Zxx Xxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
Pxx limit time, Ms.If not, there is no time limit.
If the EXX time-out, send XX alarm.If the time-out and no e, directly next.
For example: M83 Y12 x13;Explanation: open port 12 and wait for x13 to execute the next segment effectively
Only one port can be specified at a time.

4.2.32 M86 - output port, auxiliary relay output, wait for one input port invalid, do not close, execute the next section

Instruction format

M84 Yxx Xxx Pxx Exx; Output port
M84 Zxx Xxx Pxx Exx; Auxiliary relay

Command function

Xyzxx: No. 01 ~ 96.
Pxx limit time, Ms.If not, there is no time limit.
If the EXX time-out, send XX alarm.If the time-out and no e, directly next.
For example: M84 Y12 x13;Explanation: open port 12 and wait for x13 to execute the next segment
Only one port can be specified at a time.

4.2.33 M87 output port, auxiliary relay output, waiting for an input port l rising edge, closing the output, mainly used for tool selection

Instruction format

M87 Yxx Xxx Lxx Pxx Exx; Output port
M87 Zxx Xxx Lxx Pxx Exx; Auxiliary relay

Command function

Yzxx: No. 01 ~ 96.
XX: input port
LXX specified the number of times, not specified 1 time
Pxx limit time, Ms.If not, there is no time limit.If the time limit is exceeded, the output will also be turned off.
If the EXX time-out, send XX alarm.If the time-out and no e, directly next.
For example: M87 Y12 x13 L5 p5000 E100;Explanation: open the output port 12, wait for the 5th rising edge of x13 (counting 5 knives) to close the output. It is required to complete in 5 seconds, otherwise alarm No. 100 will be sent.

Only one port can be specified at a time.

4.2.34 M98 / M99 - subroutine call and subroutine return

Instruction format

M98 P####Ln;
M98 Q####Ln
M99;

Command function

1. P: subroutine calls the characteristic character, which cannot be omitted.Q: The subroutine n in the main program.
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
N0010 M03 S1000 ;
……
N0080 G0 X10 ;
N0090 M98 P0005 ;
N0100 G0X30 ;
……
N0150 M30 ;

Subroutine o0005
N0010 G01 X10 F100 ;
……
N0060 G0 Z30 ;
N0070 M99 ; Subroutine return
```

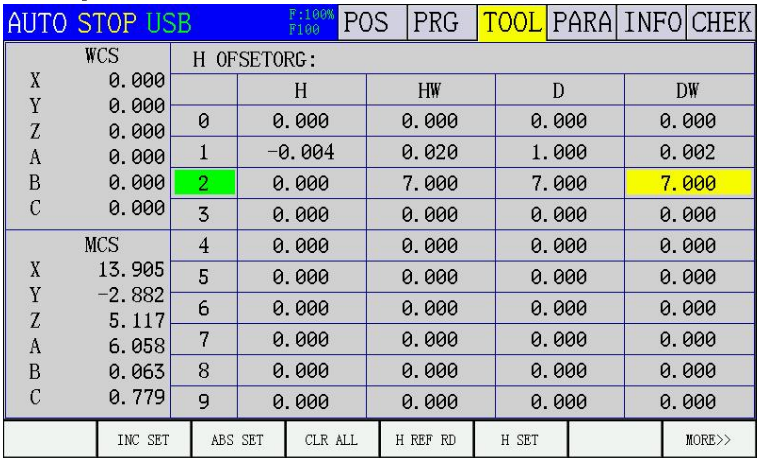
Chapter 5 tool compensation function

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.

Z-axis length offset and XY plane radius cutter compensation are available

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 re axis, the whole direction compensation is needed.



5.2 tool length compensation (G43, g44, G49)

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.

Instruction format

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

Instructions

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.

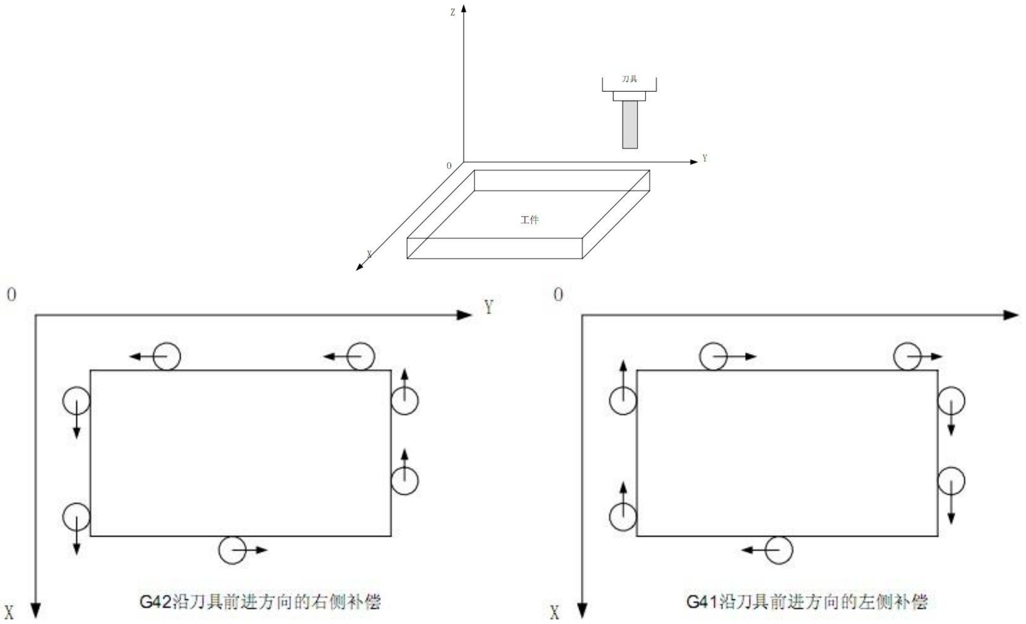
Offset number

The offset number can specify h00-h99.In the LCD / MDI panel, the offset corresponding to the offset number h01-h99 can be set in the offset memory in advance.The tool length compensation value corresponding to offset number H00 is always 0 and cannot be set.

The allowable input range of tool length compensation value is [- 9999.999, 9999.999].

5.3 tool radius compensation (tool compensation C function)

The tool radius compensation function is a function that can make the tool move on the path which is offset by a tool radius value relative to the programming path.With this function, users only need to write NC program according to the shape of the part, and do not need to consider the factors such as tool radius. The system automatically calculates the compensation vector and tool center path according to the specified compensation number to complete the machining process.



Instruction format

■ tool starting (tool radius compensation starts)

```
G00/G01 G41/G42 IP_ D_ ;
  G41: tool radius compensation left
  G42: tool radius compensation right
  IP_:Command value of axis movement
  D_ :Tool radius compensation offset number
```

Cancel tool radius compensation

```
G40 IP_;
or
D00 ;
  G40: tool radius compensation cancelled
  IP_:Command value of axis movement
  D00: Specifies the offset number 00
```

Instruction interpretation

■ G40, G41 and G42

Use G40, G41, G42 to command the cancellation and implementation of tool radius compensation vector

G code	group	function
G40	03	Cancel tool radius compensation
G41	03	Tool radius compensation (left)
G42	03	Tool radius compensation (right)

G40, G41 and G42 are the G codes of group 07, which are combined with G00, G01, G02 and G03 commands to define tool motion mode, radius compensation type and compensation direction.

Compensation amount (d code)

D code is used to specify the offset number corresponding to the compensation amount.D code is modal.

The compensation amount can be set up to 99.The number ranges from D01 to d99.(D00 means canceling tool radius compensation)

The compensation value is set in advance, which corresponds to the three digits after the D code specified in the program.

Compensation cancellation status

When the system is initially powered on, reset or executed M02, M30 commands, the system control is in the state of tool compensation cancellation.

When the compensation is cancelled, the size of the compensation vector is always 0, and the tool center path is consistent with the programming path.

At the end of the program, it must end with a compensation cancellation state.

Compensation start

In the compensation cancellation state, when the program segment meeting the following conditions starts to execute, the system enters the compensation mode.

* contains G41 or G42 commands, or controls entry into G41 or G42 mode.

The offset number of tool compensation is not 0.

The movement of any axis (except I, J, K) on the compensation plane cannot be zero.

During the compensation process, there should be no waiting for input, reading and writing system variables and other instructions. The number of instructions without movement of compensation axis (XY) should not be more than 5.Otherwise, there is an error compensation radius.

In the compensation start program section, the arc command G02 and G03 cannot be instructed, otherwise an alarm will be generated.In the compensation start section, the system reads two program segments continuously. The first program segment reads in and executes, and the second program segment enters the tool compensation buffer.

In single block mode, two segments are read in, the first segment is executed, and then stopped.

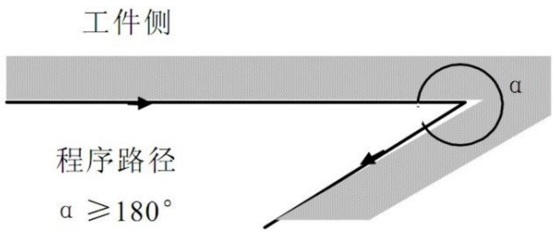
During continuous execution, two program segments are usually read in advance, so there are three program segments in CNC, one is the program segment in execution, and the next two segments enter the buffer.

5.3.1 tool radius compensation offset path

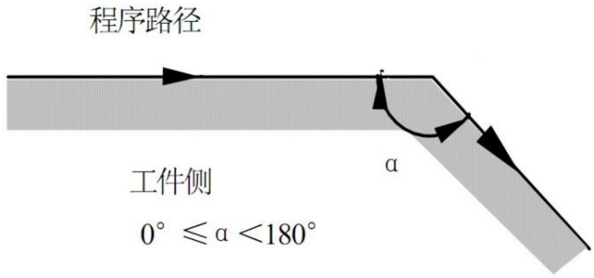
5.3.1.1 inside and outside

When the tool nose radius compensation is carried out, the corner of the two programming tracks is different, and the tool nose compensation track is also different.Therefore, when the angle between the intersection point of two moving program segments is greater than or equal to 180 ° on the workpiece side, it is called "inside", and when it is between 0 ~ 180 ° it is called "outside".It is shown in the figure below.

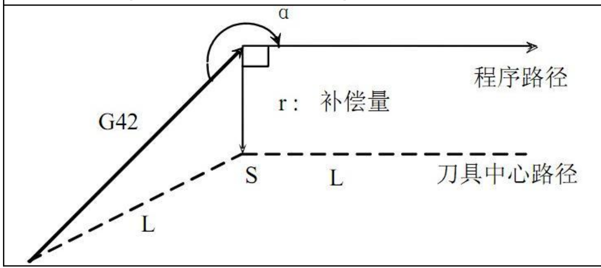
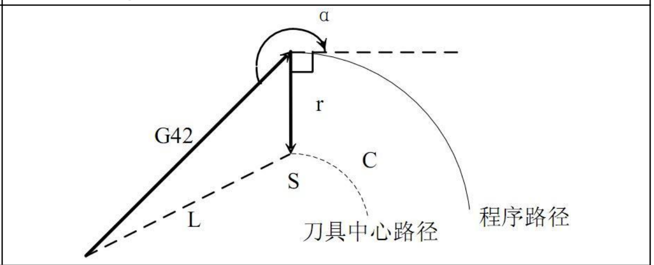
Inside:

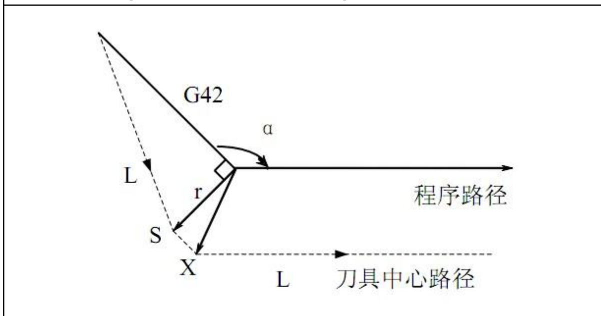
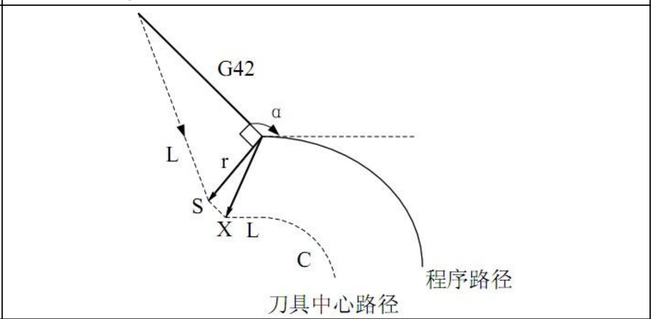


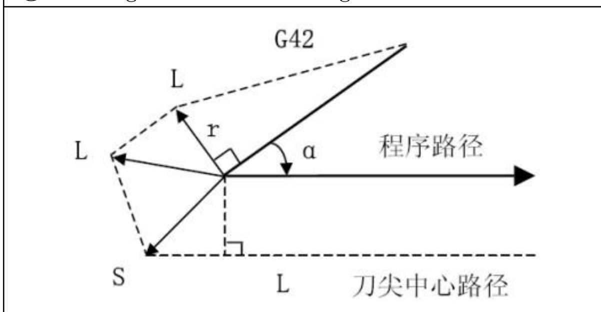
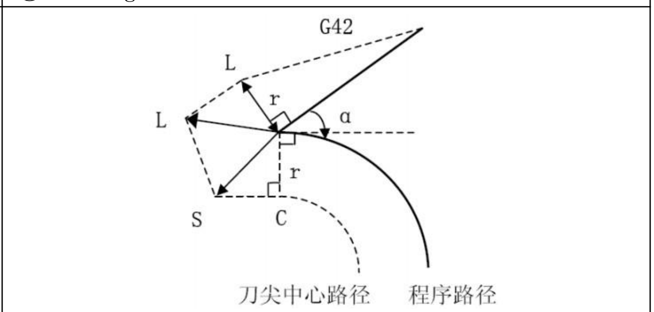
outside:

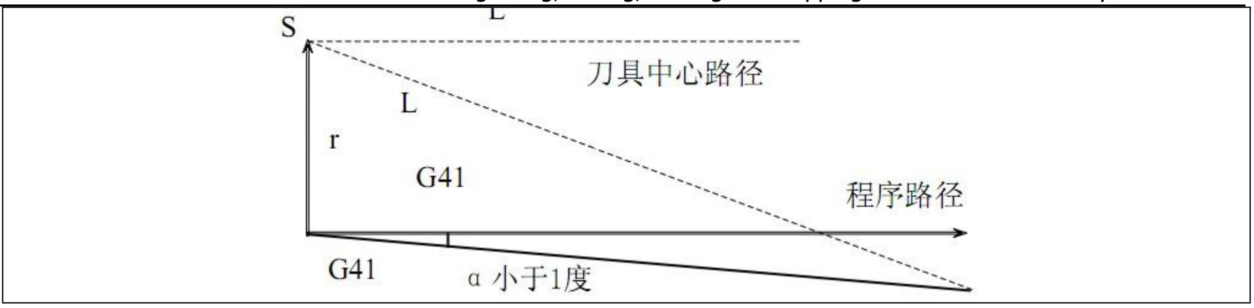


5.3.1.2 tool compensation establishment

(a) Move along the inside of the corner (a ≥ 180 °)	
① Straight line - straight line	② Straight line arc
	

(b) Moving along the outside of the obtuse angle (180 ° > a ≥ 90 °)	
① Straight line - straight line	② Straight line arc
	

(c) Move along the outside of acute corner (a < 90 °)	
① Straight line - straight line	② Straight line arc
	
(d) Move along the outside of the corner with an acute angle less than 1 degree.(a<1°)	

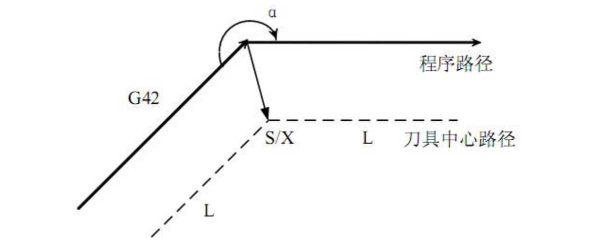


5.3.1.3 knife compensation

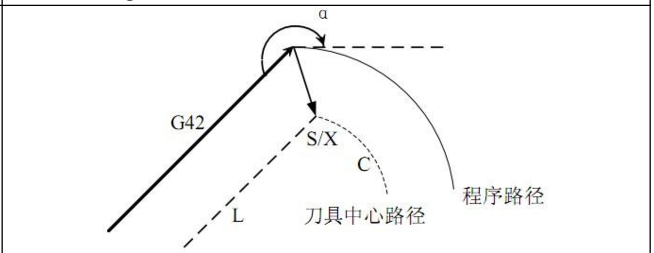
The offset trajectory from the establishment of cutter compensation to the cancellation of cutter compensation is called tool compensation.The specific tool compensation is shown in the following figure:

(a) Move along the inside of the corner (a ≥ 180 °)

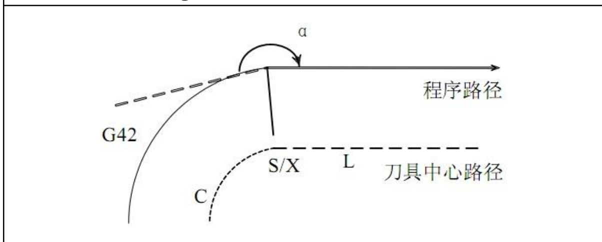
① Straight line - straight line



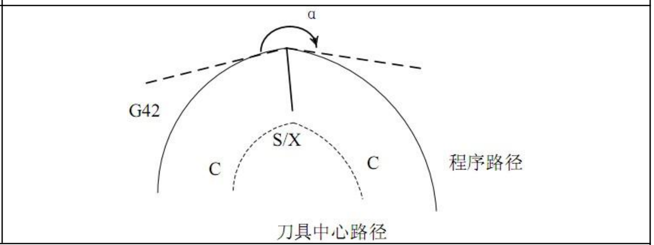
② Straight line arc



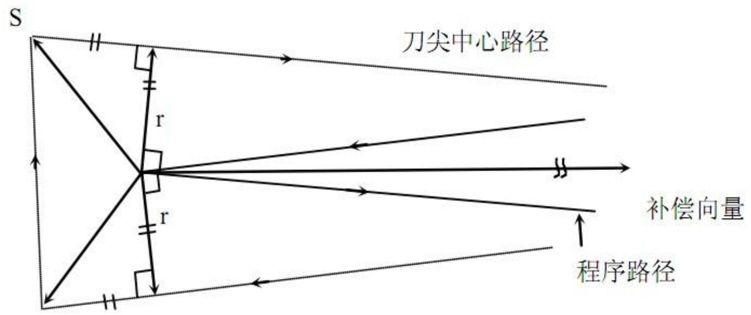
③ Arc straight line



④ Circular arc arc



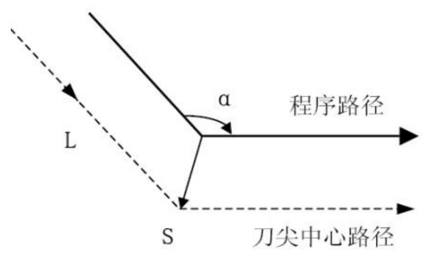
Inner processing and compensation vector amplification less than 1 degree



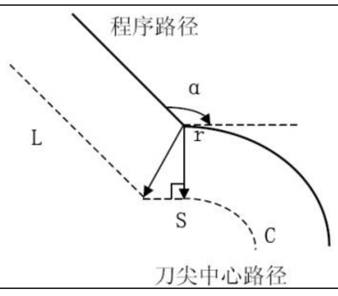
In the same way:
① Arc straight line
② Straight line arc
③ Circular arc arc

(b) Moving along the outside of the obtuse angle (180 ° > a ≥ 90 °)

① Straight line - straight line

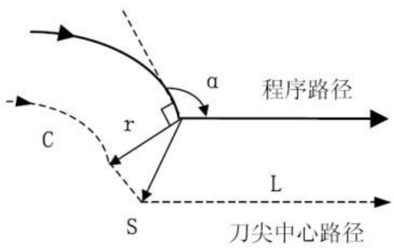
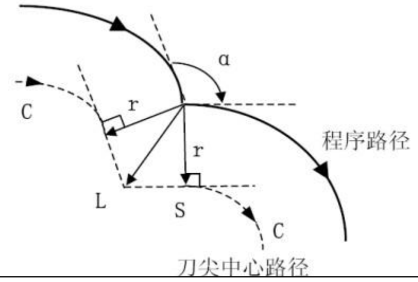


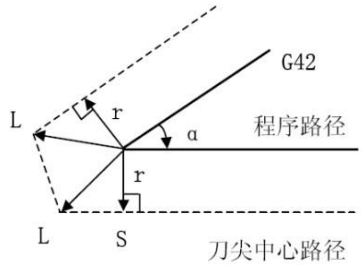
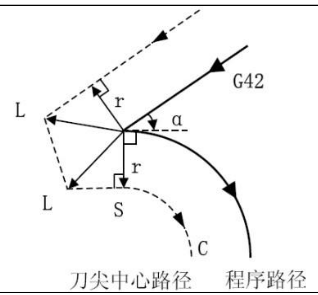
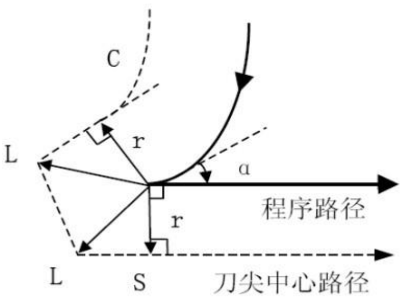
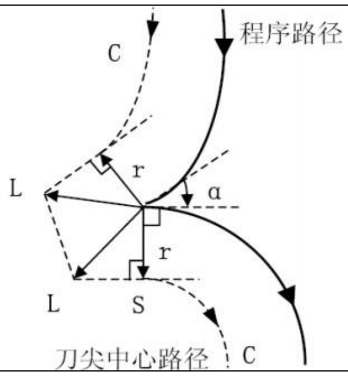
② Straight line arc



③ Arc straight line

④ Circular arc arc

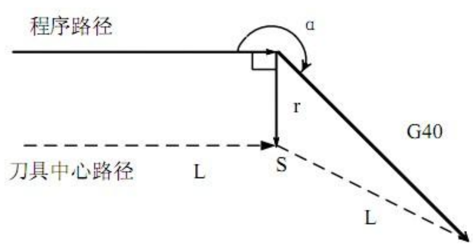
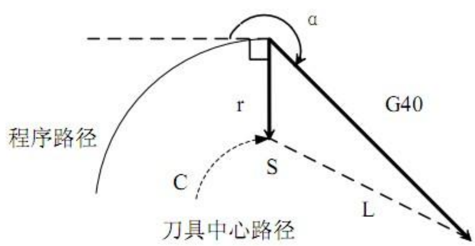
(c) Move along the outside of acute corner (a < 90 °)	
① Straight line - straight line	② Straight line arc
	
③ Arc straight line	④ Circular arc arc
	

5.3.1.4 tool compensation cancellation

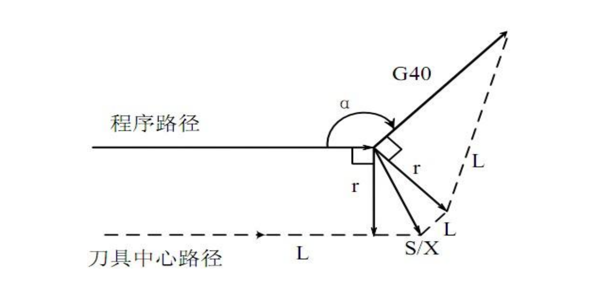
Under normal conditions, it is necessary to use the instruction G40 to cancel cutter compensation C.

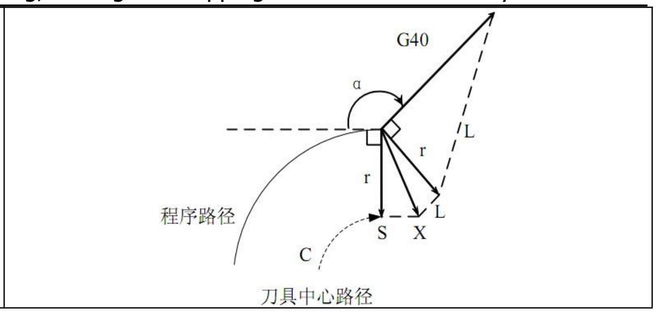
When the tool compensation is cancelled, the movement command is not arc command (G02 / G03).If the command arc system will generate an alarm and stop the movement.

The following figure shows the cancellation of cutter compensation:

(a) Move along the inside of the corner (a ⩾ 180 °)	
① Straight line - straight line	② Arc straight line
	

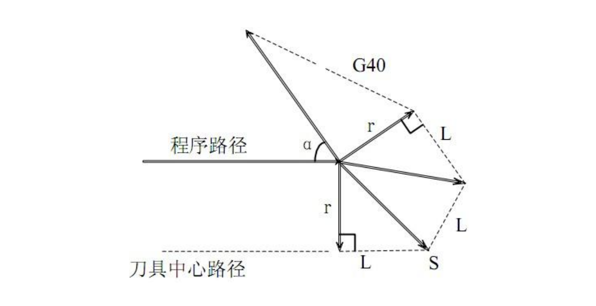
(b) Moving along the outside of the obtuse angle (180 ° > a ⩾ 90 °)	
① Straight line - straight line	② Arc straight line



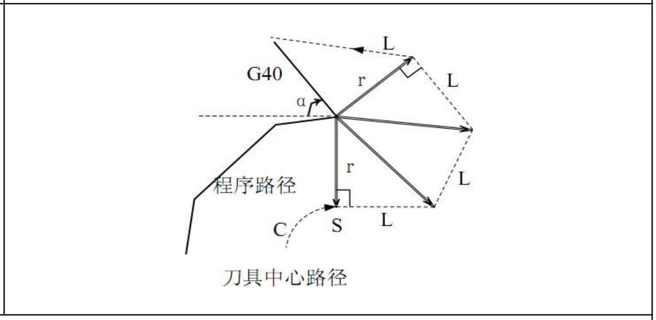


(c) Move along the outside of acute corner (a < 90 °)

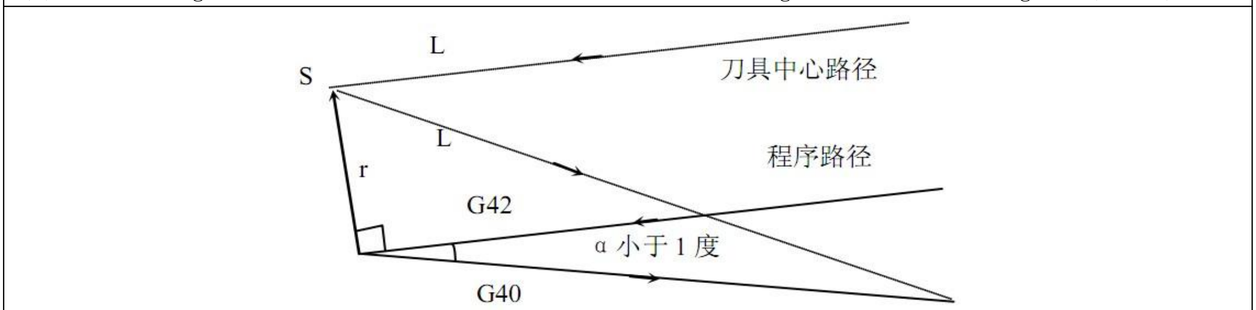
① Straight line - straight line



② Arc straight line



(d) Move along the outside of the corner with an acute angle less than 1 degree.(a<1°)



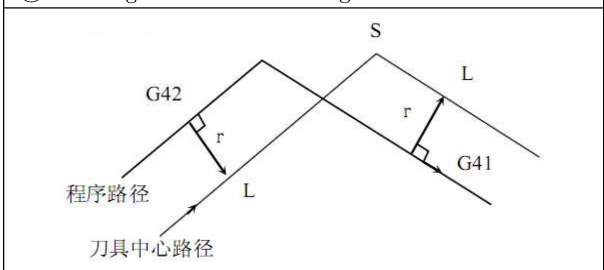
5.3.1.5 change of compensation direction during tool compensation

Tool diameter compensation g codes (G41 and G42) determine the compensation direction, and the symbols of compensation amount are as follows:

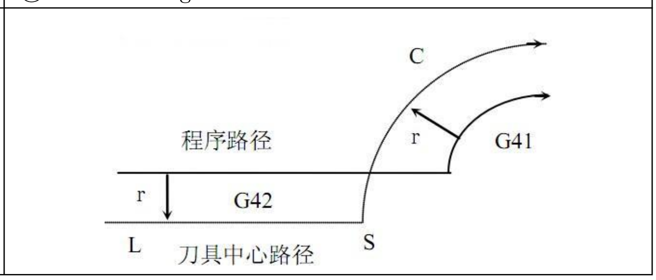
G code	Symbol of compensation quantity	
	+	-
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

In special cases, the compensation direction can be changed in the compensation mode.However, it is not allowed to change the program section at the beginning.When the compensation direction is changed, there is no concept of inside and outside for all conditions.The following compensation is assumed to be positive.

① Straight line - straight line

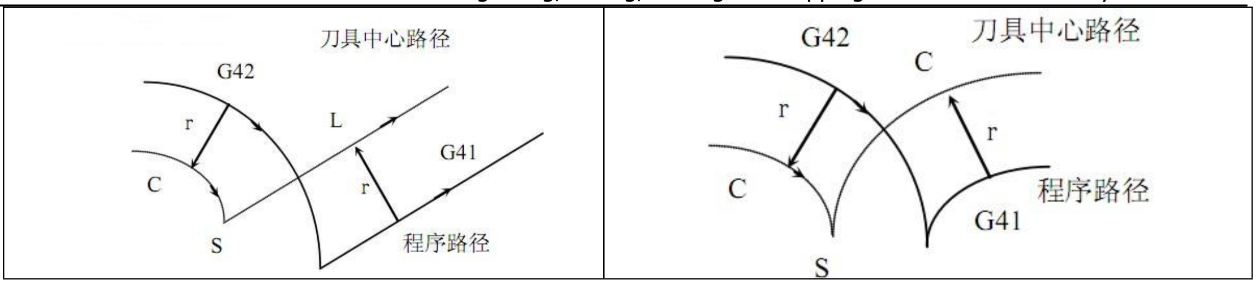


② Arc straight line



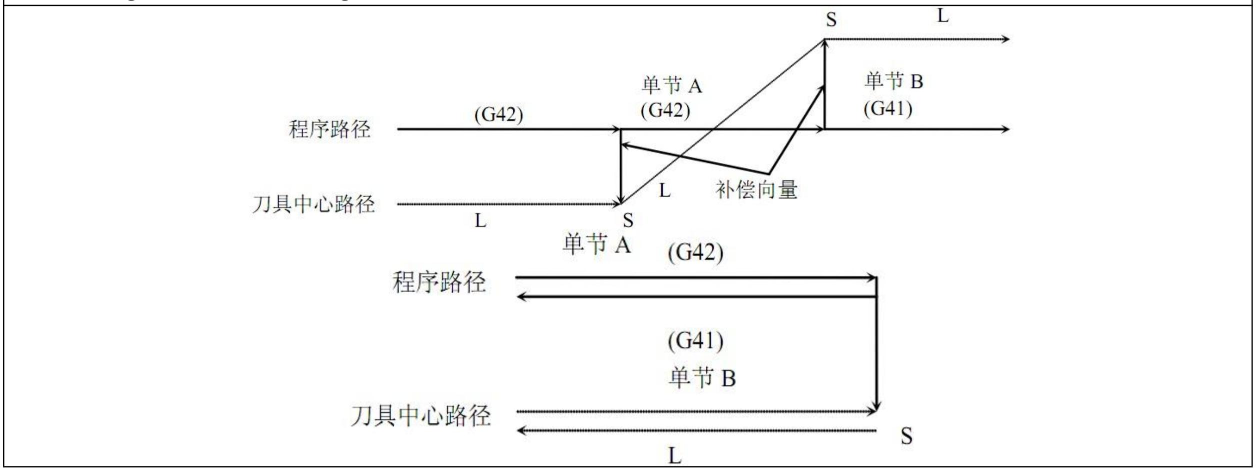
③ Arc straight line

④ Circular arc arc

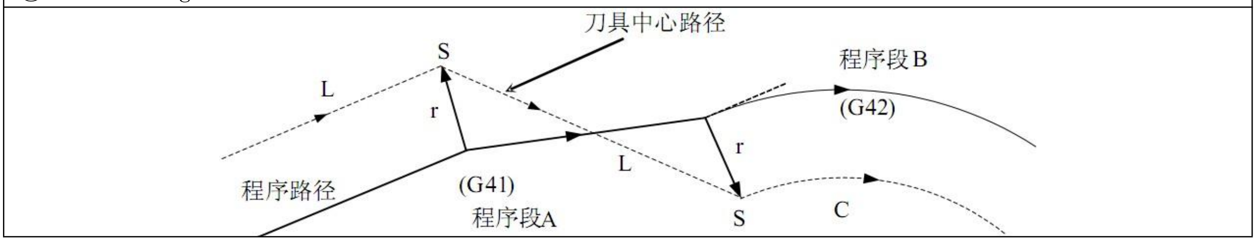


If the compensation is executed normally, but there is no intersection point
If it is not necessary to offset the direction of the program segment G41 to the program segment 42, use the program segment gb42 to offset the program segment.

① Straight line - straight line



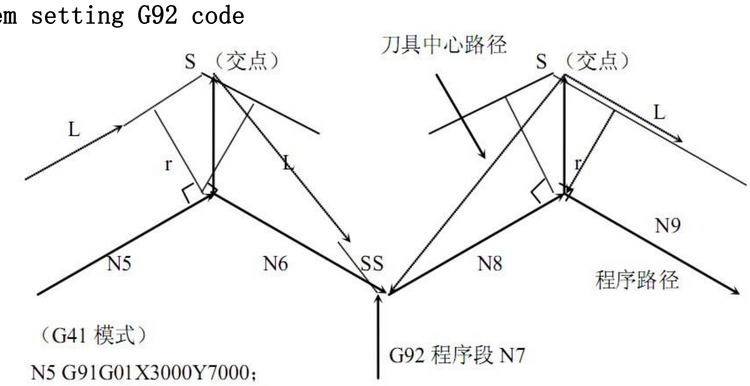
② Arc straight line



5.3.1.6 tool compensation temporarily cancelled

In the compensation mode, if G92 and G28 codes are specified, the compensation vector will be temporarily cancelled. After the code is executed, the compensation vector will be automatically restored.The tool moves directly from the intersection point to the command point where the compensation vector is cancelled. When the compensation mode is restored, the tool moves directly to the intersection point.

Coordinate system setting G92 code

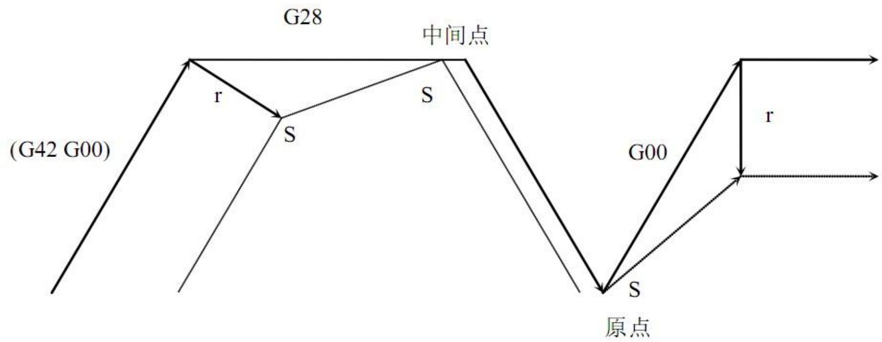


（G41 模式）
N5 G91G01X3000Y7000;
N6 X-3000Y6000;
N7 G92X1000Y2000;
N8 G01X4000Y8000;

Note: SS refers to the point where the tool stops twice in one-way block mode.

The G28 automatically returns to the reference point

In the compensation mode, if G28 is instructed, the compensation will be cancelled at the intermediate point, and the compensation mode will resume automatically after the reference point returns.



Special circumstances

When the inside corner machining is less than the tool tip radius

In this case, the inside offset of the tool will lead to excessive cutting.After the start of the previous block or the corner movement, the tool movement stops and an alarm is displayed.

When machining a step smaller than the tool tip radius

When the program contains a step smaller than the tool tip radius and the step is an arc, the tool center path may form a movement direction opposite to the program path.At this point, the first vector is automatically ignored and moved directly to the end of the second vector.In single block mode, the program will stop at this point. If it is not in one-way block mode, the loop operation will continue.If the step is a straight line, the compensation is performed correctly without an alarm.But the uncut part will remain.

When G code contains subroutines

Before calling the subroutine, the system must be in compensation cancel mode.After entering the subroutine, the C cutter compensation can be re established, but it must be the compensation cancellation mode before returning to the main program.Otherwise, an alarm will appear.

When the compensation amount is changed

① Usually, when changing tool in cancel mode, the value of compensation is changed.If the compensation mode is changed, only after the new compensation mode is changed.

② The positive and negative of compensation and the path of tool tip Center

If the compensation is negative (-), G41 and G42 are exchanged with each other in the program.If the tool center moves along the outside of the workpiece, it will move along the inside and vice versa.

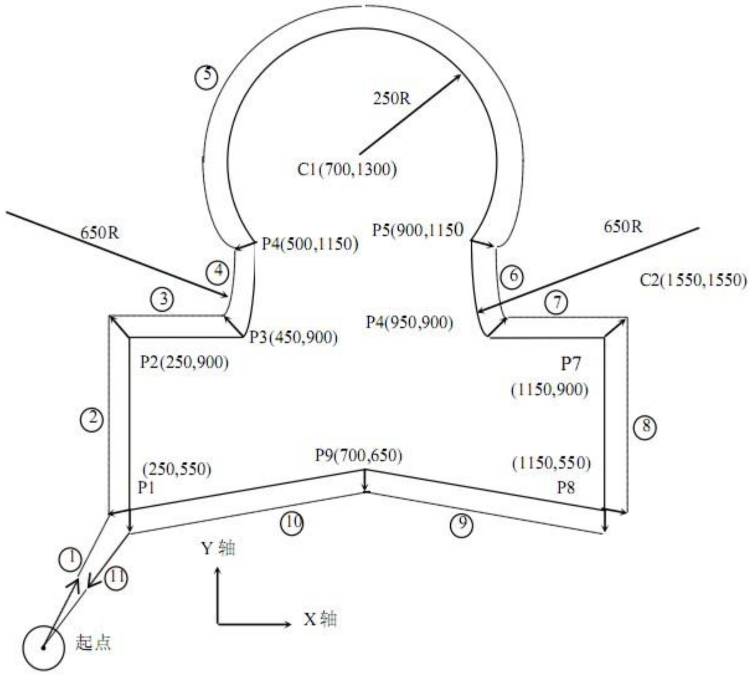
This is shown in the following example.In general, the compensation is (+).When the tool path is shown in (a), if the compensation is negative (-), the tool center moves as (b), and vice versa.

Also note that when the offset symbol changes, the tool nose offset direction also changes, but the assumed nose direction remains unchanged.So don't change the sign of the offset.

The end point of the programmed arc is not on the arc

When the end point of the arc in the program is not on the arc, the tool movement stops and the alarm message is displayed.

5.3.2 application examples



Program: (the compensation amount is preset with the compensation number)

N0 G92 X0 Y0 Z0; The tool is positioned at the start position x0 Y0 Z0
N1 G90 G17 G00 G41 D07 X250.0 Y550.0; Starting the tool, the tool is offset to the tool path with the distance specified by D07
N2 G01 Y900.0 F150; Processing from P1 to P2
N3 X450.0; Processing from P2 to P3
N4 G03 X500.0 Y1150.0 R650.0; Processing from P3 to P4
N5 G02 X900.0 R-250.0; Processing from P4 to P5
N6 G03 X950.0 Y900.0 R650.0; Processing from P5 to P6
N7 G01 X1150.0; Processing from P6 to P7
N8 Y550.0; Processing from P7 to P8
N9 X700.0 Y650.0; Processing from P8 to P9
N10 X250.0 Y550.0; Processing from P9 to P1
N11 G00 G40 X0 Y0; To cancel the offset mode, the tool returns to the start position x0, Y0

Chapter 6 User Macro Program

6.1 Definition

User macro program allows users to use basic program language features such as variable, arithmetic operation, logic operation, bit operation, condition transfer, cycle control and program call, which makes programming more convenient, flexible, easy and fast.It can greatly improve the generality of the program. Only by assigning values to different main programs, the same subroutine can be called to process the same type of parts.

Main program user macro program Ontology

Program segment 1; Program segment 2; …… …… Macro program instruction; …… ……		Complete a function A set of instruction s for

6.2 variables

Instruction format

#i ;

For example: ◌ɔ1, ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ◌ɔ.

Instructions

(1) type of variable: variable can be divided into four types according to variable number.

Variable number	Variable type	function
#0	Null variable	The variable is always empty and no value can be assigned to it.
#1～#99	local variable	Local variablesThe main program, sub-A program, sub-b program and t code have their own local variables.The local variables of main program, sub-A program and sub-b program are cleared every time the program starts.T code local variables are cleared only on power up.
#100～#199 #500～#599	Common 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 variables	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。

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 the assignment with < null variable >, the < null variable > is the same as "0" in other cases.

When #1= < empty >	When #1=0
#2=#1 ↓ #2= < empty >	#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 expression, < empty > in EQ and NE is different from "0"

When #1= < empty >	When #1=0
#1 EQ #0 ↓ found	#1 EQ #0 ↓ false
#1 NE #0 ↓	#1 NE #0 ↓

false	false
#1 GE #0 ↓ found	#1 GE #0 ↓ false
#1 GT #0 ↓ false	#1 GT #0 ↓ false

(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 function	Read-write function
#1001~#1096	input port	read only
#1101~#1196	delivery outlet	read and write
#1201~#1296	Auxiliary relay	read and write
#1301~#1312	Input 8bit to read, #1=#1301 reads x01 ~ x08 at a time, # 1 = # 1302 reads X09~X016… at a time …	read only
#1401~#1412	Output 8bit read and write, #1401=0, one-time Y01~Y08 reset,	read and write
#1501~#1512	Auxiliary relay 8bit reads and writes, #1501=0, and Z01~Z08 is reset at one time.	read and write

6.3.2 Return to zero mark

Macro variable number	Macro variable function	Read-write function
#1601~#1606	Read the current axis back to zero: 0 has not returned to 1, and 2 has returned to zero.	read only

6.3.3 Macro variables of tool compensation system

Macro variable number	Macro variable function	Read-write function
#2001~#2006	Read the knife complement number of current axes (XYZABC)	read only
#5081~#5086	Read and write the value of the knife complement number of the current axis (XYZABC)	Read-write (read-only for knife 0)

6.3.4 Other system variables

Macro variable number	Macro variable function	Read-write function
#2007	Upper spindle tool number	read only
#2008	Tool set number of current tool	read only

	magazine	
#3091	Workpiece counter	read and write
#5041~#5044	Absolute coordinates of each axis	read only
#5061~#5064	Coordinate of each axis machine tool	read only
#5060	Current coordinate system 54~59	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 constants or variables composed of functions or operators. The variables #j and #k in the expression can be assigned with constants. The variables on the left can also be assigned with expressions.

<table>
<tr><td>function</td><td>format</td><td>remarks</td></tr>
<tr><td>assignment</td><td>#i=#j ;</td><td>Assignment operation.</td></tr>
<tr><td>addition
subtraction
multiplication
division</td><td>#i=#j + #k ;
#i=#j - #k ;
#i=#j * #k ;
#i=#j / #k ;</td><td>Arithmetic operations.
If J = = I, the simplified symbol (+ =, - =, * =, / =) can be used. If ○ɔ I = ○ɔ I + ○ɔ K, it can be simplified as ○ɔ I + = ○ɔ K.</td></tr>
<tr><td>And
Exclusive or
or
Shift left
Shift right</td><td>#i=#j & #k ;Or ○ɔ I = ○ɔ J and ○ɔ K;
#i=#j ˆ #k ;Or ○ɔ I = ○ɔ J XOR ○ɔ K;
#i=#j | #k ;Or ○ɔ I = ○ɔ J or ○ɔ K;
#i=#j << #k ;
#i=#j >> #k ;</td><td>Bit operation. This operation will cast a floating-point number to an integer. Bit operations operate as binary integers.
If J = = I, the simplified symbol (& =, ˆ =, |, =,,) =, can be used. If ○ɔ I = ○ɔ I & ○ɔ K, it can be simplified as ○ɔ I - = ○ɔ K.</td></tr>
<tr><td>be equal to
Not equal to
greater than
Greater than or equal to
less than
Less than or equal to</td><td>#i=#j == #k ;Or ○ɔ I = ○ɔ J EQ ○ɔ K;
#i=#j != #k ;Or ○ɔ I = ○ɔj ne ○ɔ K;
#i=#j > #k ;Or ○ɔ I = ○ɔj GT ○ɔ K;
#i=#j >= #k ;Or ○ɔ I = ○ɔj Ge ○ɔ K;
#i=#j < #k ;š š š š š#########;
#i=#j <= #k ;Or ○ɔ I = ○ɔ J Le ○ɔ K;</td><td>Relational operations.
The result is a 32-bit unsigned integer 0 (false) or 1 (true).</td></tr>
<tr><td>square root
absolute value

rounding
Round up
Rounding down
Natural logarithm
exponential function</td><td>#i=SQRT[#j];
#i=ABS[#j];
#i=FABS[#j];
#i=ROUND[#j];
#i=FUP[#j];Or =CEIL[#j];
#i=FIX[#j];Or ○ɔ I = = floor [# J];
#i=LN[#j];Or ○ɔ I = log [○ɔ J];
#i=EXP[#j];</td><td></td></tr>
<tr><td>sine
Arcsine
cosine
Cosine inverse
tangent
Arctangent</td><td>#i=SIN[#j];
#i=ASIN[#j];
#i=COS[#j];
#i=ACOS[#j];
#i=TAN[#j];
#i=ATAN[#j]</td><td>Trigonometric function. When specified in angle, such as 90 ° 30 'table 5 degrees.
Constants or expressions can be used instead of ○ɔ J.</td></tr>
</table>

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).

(2)ARCSIN # i=ASIN[#j]

When ○ɔJ exceeds the range of - 1 to 1, an alarm is given.
The constant can replace the variable ○ɔJ.

(3)ARCCOS # i =ACOS[#j]

The output range of the results is from 180 ° to 0 ° 。
When it exceeds the range of - 1 to 1, an alarm is given.
The constant can replace the variable ○ɔJ.

(4) natural logarithm ○ɔI = ln [#j]

When the opposition number (○ɔj) is 0 or less than 0, the alarm will be given.
The constant can replace the variable ○ɔJ.

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

(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 ○ɔ1 = round [○ɔ2], where ○ɔ2 = 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.
A goto statement (unconditional transfer).
If statement (conditional transfer: if. Then.).
A 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: Sequence number (1 ~ 99999)
```

give an example

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

6.5.2 conditional control (if statement)

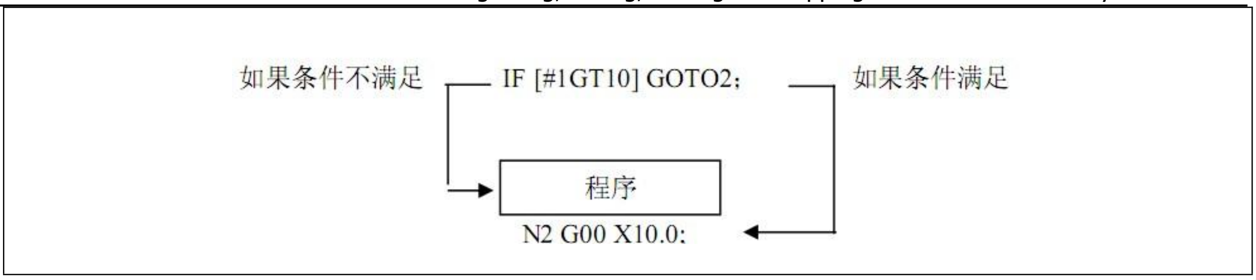
Command format 1

```
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 ○ɔ1 is greater than 10, it is transferred to the block of sequence number N2.



Command 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 ◌ↄ1 is equal to the value of ◌ↄ2, 0 is assigned to the variable ◌ↄ3;If they are not equal, the order goes down and the assignment statement after then is not executed.

Instructions

You must use conditional expression, conditional expression, or conditional expression.
Conditional operators are shown in the following table.

operator	meaning
EQ	Equal to (=)
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 integers 1 to 10.

```
09600
#1=0;  Store the initial value of sum variables
#2=1;  The initial value of the addend variable
N1 IF[#2 GT 10]GOTO2; When the addend is greater than 10, it is transferred to N2
#1= #1+#2;     Calculate sum
#2= #2+1 ;     Next addend
GOTO1;         Go to N1
N2 M30; End of procedure
```

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.

Command format 1

While [conditional expression] do m; …… END m ; m: Specifies the label (1 ~ 1023) of the loop execution range.

Instructions

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 is executed, #1 is added from 1 to 100, #2 gets the sum of 1 + 2 + 3 +. + 100.

Command format 2

```
DO m ;
……
END m ;
```

m: Specifies the label (1 ~ 1023) of the loop execution range.

Instructions

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 #1 and #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 ;
……
```

The 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 #500

Procedure o0001NC

```
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
G91G10L2P1Z-#502 G90 ;Compensation of z-axis cutter coordinate system
N10 Z0 ;Z axis to machining position
G1X100F100 ;Start machining of x-axis
X10
G0Z10 ;Z axis lifting
#500+=1 ;Processing count plus one
M05 ;Stop spindle
M30
```

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 ;Spindle on
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 above Z axis
#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 Three Axis Circle Drilling

```
    Main program o0001NC
G90G54G98HZ1
    M03S1000
    G0 x0y0 / / to the center of the circle
    G73Z-20R2Q5J2F500L0;Deep hole processing instruction G73, l0: do not hit Center
    G70I50J0L3 ;Radius 50 3 / 3 drilling
    G80
M05
    M30
```

7.4 Three Axis Rectangular Array Drilling

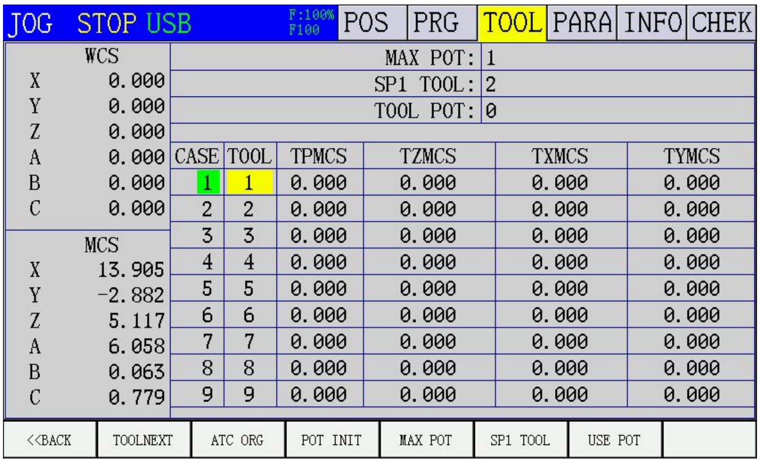
Define panel parameters first
501 x number of holes
502 number of Y holes
503 x spacing
504 y spacing
505 hole depth

Drilling sequence, in line zigzag

```
Main program o0001NC
G90G54G98HZ1
M03S1000
G0Z10
#2=#501-1 ;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 5 DEBUGGING AND USE OF TOOL MAGAZINE

The tool magazine of this system is realized by special program. The related interfaces are in the tool pocket table and tool magazine code. Enter from tool → more → more.



Pocket Watch

Note that the bottom button is only displayed and valid in the corresponding mode.[tool change] [tool magazine return to zero] is displayed in [manual] mode.

[number of tool sets] is the maximum number of tools loaded in the tool magazine.

[spindle tool] the current tool on the spindle.

[current tool cover] some needs.Such as hat type, some do not need, such as straight row magazine.

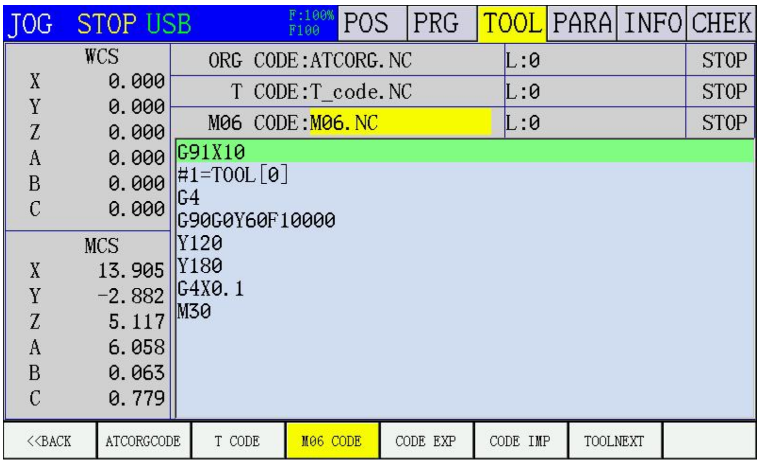
[tool] the ATC of the tool in the tool sleeve varies randomly, and the others are basically fixed.

[TPMCS] the position of the tool sleeve on the motor in the servo magazine.

[tzmcs] [txmcs] [tymcs] some tool libraries such as straight row magazine need to take the tool at the corresponding tool pocket position.Setting these parameters is convenient for programming.

If [tool change] is valid manually, the control will send out t [spindle tool + 1], if it is greater than [number of tool sets], take 1.Wait for the T code to execute before issuing M06.

Manual zero control [effective code return]



Tool magazine code

Return to zero code (M7): used for special operations such as tool magazine return to zero.

T Code: set the current tool number to be changed.It is used for ATC tool magazine rotation, and the tool number is basically set under other tool magazine.T code cannot be large

At T99.Otherwise, an alarm will be given.

M06 code (M6): used to write tool exchange program.

To edit the tool magazine code, in addition to opening the [program switch], it also requires [C-level authority].After debugging, the power on authority should be set to 0.prevent

The workers made random changes.In addition, the tool magazine code pause should also be cancelled.

These three codes are not subroutines.They are parallel with the main and auxiliary programs, and can only be started by the main and auxiliary programs.They can't inspire each other

Move.That is, these three programs can not contain M7, t, M6 code.

The absolute coordinates in the zero return code, t code, and M6 code are machine coordinates, independent of all offsets.

Code group, since you can't see the G code group of T code, please start the program with G90 to ensure that it is absolute programming every time.In group G, the main and side effects were not affected

Preface.It is absolutely not related to writing the three programs.

G code support: G0 G1 g53 G28 G30 G93, others should not be used in tool magazine code.

Some macro functions are used in the tool library code

TNMAX[]	The format tnmax [0] is used to obtain the number of tool sets.0 is meaningless, only meets the parsing format requirements
TOOL[]	Format tool [0], get t code and set tool number.0 is meaningless, only meets the parsing format requirements
POT[]	Format pot [0] to obtain the current tool set number.0 is meaningless, only meets the parsing format requirements
POTT[]	Format pot [0] ~ pot [99] to obtain the tool number in the specified tool pocket.Pot [0], tool number on spindle
TPOT[]	Tpot [0] ~ pot [99] to obtain the tool pocket on which the tool number is located. If there is a repetition, return the first found one.No report [program error] found
TPMCS[]	TPMCS [1] ~ TPMCS [99], return to the tool pocket position of servo magazine.For example, the axis of servo magazine is a, a TPMCS [1]
TXMCS[]	Txmcs [1] ~ txmcs [99], return the tool pocket to X-axis position
TYMCS[]	Tymcs [1] ~ tymcs [99], return the tool pocket to the y-axis position
TZMCS[]	Tzmcs [1] ~ tzmcs [99], return the tool pocket to z-axis position

Note: TPMCS, txmcs, tymcs, tzmcs, all need to be set by human, and they should be set as machine tool coordinates in the tool magazine code.Not necessarily

Yes, if it is used to make a read-only parameter in the main program.

M code for tool pocket table should be updated in tool magazine code

M26 Pn Lm	[0] is set as [0] tool sleeve.P0 is the tool number on the spindle.
M27 Pn	Set the current tool cover number to n [1 ~ 99]

There is no special tool magazine input and output port function, only one [tool release output].You can refer to the input / output name in [diagnosis] and import it into the tool magazine

To the common function name convenient memory.Program with M.

Taking the servo tool magazine as an example, this paper explains how to compile the three codes.

Set the a axis to the [tool magazine] mode.This mode can only be activated in these three codes.When setting TPMS, manual or hand wheel rotation is required, which can be set temporarily

Set as the axis of rotation.

TPMS should be set first and tzmcs [1] should be set, which is the tool change point of Z axis.Tzmcs [2] tool change lifting point

Input 1: forward in place

Input 2: back in place

Output 1: magazine forward, break back

Return to zero code:

G91G28A0;Motor return to zero

G90G0A10;No, if it's not in the desired position after returning to zero.

G93 A0;Set 0, the purpose is the same as above, the motor return to zero is not in the ideal position, go to the ideal position, set as machine coordinate 0.

#90=1;Set the return to zero flag, and judge whether there is zero return in t code.

M30

T code

IF[#90==1] GOTO10; Zero return, zero crossing

G91G28A0;This indicates that the motor does not return to zero and starts to return to zero

G90G0A10;No, if it's not in the desired position after returning to zero.

G93 A0;Set 0, the purpose is the same as above, the motor return to zero is not in the ideal position, go to the ideal position, set as machine coordinate 0.

#90=1;Set return to zero flag

N10 M30;There's nothing to do except go back to zero.At the same time, the target tool number is transmitted

M06 code

G90

IF[TOOL[0] == POTT[0]] GOTO 100 ;Set the tool to be consistent with the spindle tool, directly to the end

M05 ;Prevent spindle from moving

M19 ;Spindle positioning

G0 Z [TZMCS[1]] ;Z tool change point

IF[POTT[0]==0] GOTO10 ;If there is no knife on t0, take the knife directly without returning it

G0 A [TPMCS[TPOT[POTT[0]]]] ;A position of tool sleeve to spindle tool number.

M85 Y1 X1 P3000 E100 ;The magazine moves forward and is in place.Three seconds, No. 100 alarm

G4 X0.5

N10 M21 ;Loose knife

G4 X0.5

IF[TOOL[0] ==0] GOTO 20 ;The spindle does not take the tool

G1 Z [TZMCS[2]] F100 ;Z axis lifting

G0 A [TPMCS[TPOT[TOOL[0]]]] ; A to t codes specify the location of the tool pocket for the tool number.

M27 P [TPOT[TOOL[0]]] ; ;Update current tool pocket

G1 Z [TZMCS[1]] F100 ;Z axis down

G4 X0.5

N20 M20 ;Tight knife

G4 X0.5

M80 Y1

M71 X2 P3000 E100 ;Wait for the tool magazine to return to position, and if it fails to reach the designated position for 3 seconds, alarm No. 100 will be sent out

M05 ;Cancel spindle positioning

M26 P0 L[TOOL[0]] ;Set the spindle tool number.In this case, it is not necessary to update the other tool pocket tables.

N100 M30

Because M06 needs to use t code to specify the tool.

In the main program format to use t01m06, t code before M06.

T code and M instruction are not followed by movement related instructions.

For example, t01m06 X02.However, the system will ignore the move command of mxg106 immediately after the move command

In this example, due to the servo magazine, the motor is automatically planned nearby.If it is an ordinary motor, how to plan nearby?The planning procedure is as follows,

#1= POT[0] ;Get current pocket

#2= TPOT[TOOL[0]] ;Get the target pocket

#3= TNMAX [0] ; Get the number of sets

#4=1 ; #4 1 forward - 1 reverse

#5=0 ; #5 number of turns

IF[#2 < #1] GOTO 10 ; Target less than current, jump

#5= #2-#1 ;Here, the target is greater than the current, forward rotation, and calculate the number of forward rotation.

IF[#5*2 <= #3] GOTO 100 ;The number is no more than half a circle. The planning is completed

#5= #3-#5 ;If the number is more than half a circle, reverse it. If you subtract the original quantity by one circle, it is the reverse number

#4=-1 ;Reverse direction

GOTO 100 ;Planning completed

N10 #4=-1 ;If the target is smaller than the current, the direction is reversed

#5=#1-#2 ;Here the target is larger than the current, turn forward.

IF[#5*2 <= #3] GOTO 100 ;The number is no more than half a circle. The planning is completed

#5= #3-#5 ;If the number is more than half a turn, turn forward. If you subtract the original number by one circle, it is the number of positive turns

#4=1 ; Positive direction

N100 ; The following is to control the actuator according to the direction and quantity.