Milling machine CNC system use manual

preface

Dear customers:

We are deeply honored to choose our milling machine CNC system.

Before you use our system, you must read our system use manual.

▲ In order to avoid improper operation caused by accidents, must have the corresponding qualified personnel to operate the CNC system.

▲ Please read this user manual carefully before the operation!

Special note:

The switch power supply installed on the system (in) is the special power supply provided only for the CNC system manufactured by the company.

Users are prohibited from using the power supply for other purposes. Otherwise, there will be a great danger!

Thank you for using our company's products, all the employees of the company to express their sincere thanks!

⚠ declare!

This manual describes the operation of the product as fully as possible, but due to too many possibilities involved, all allowed and not allowed operations cannot be explained. Therefore, in order to ensure the normal use of the product and the safety of personal and equipment, the operation not allowed in this manual is regarded as abnormal operation.

System damage caused by earthquake, flood, typhoon and other irresistible factors shall not be covered by the warranty.

This manual is collected by the end user. All specifications and designs are subject to change without prior notice.

⚠ warn!

Before installation, connection and programming of this product, read this manual and machine tool manufacturer in detail, and operate in strict accordance with the requirements of this manual and manual, otherwise it may lead to damage to products and machine tools, scrap of workpiece, or even personal injury.

⚠ pay attention to!

The functions of the products described in this manual are only for this product, and the CNC machine tool is installed with this product. The actual functional configuration and technical performance are determined by the design of the machine tool manufacturer, and the functional configuration

and technical indicators of the CNC machine tool shall be subject to the instructions of the machine tool manufacturer.

matters need attention

◎ Transportation and storage

1、Product packing boxes should not stack in more than six layers;

2、Do not climb, stand, or place heavy objects on the product packaging box;

3、Do not use the cable connected to the product to drag or move the product;

4、 Do not collision, scratch the panel and display screen;

5、Product packing boxes should be protected from moisture, sun exposure and rain.

◎ OOBA

1、After opening the package, please confirm whether it is the product you purchased;

2. Check whether the product is damaged during transportation;

3. Verify whether the parts are complete against the list;

4. If there is any product model inconsistency, lack of accessories or transportation damage, please contact the company in time.

◎ wiring

1、The personnel participating in the wiring and inspection must be professionals with the corresponding ability;

2、 The wiring must be correct and firm to avoid product failure or unexpected consequences;

3、Unplug the product before opening the product case.

◎ examine and repair

1、The power supply must be cut off before maintenance or replacement of components;

2、 In case of short circuit or overload, check the fault, and it can be restarted after troubleshooting;

3. Do not cut the power off of the product frequently. If the power is cut off, the power must be powered on again, at least 1 minute apart.

Safe use instructions

In order to enable you to use the system safely and correctly, all operators who write the machine tool program and operate the machine tool must fully understand the instructions provided by the machine tool manufacturer and the contents of this instructions.

catalogue

1. General warnings and attention

 **[warn]**

1. When using the new program for actual processing of parts, do not directly processing, in the case of no tools and parts, using single section feed, test run, verify the correctness of the machine mechanical movement. Without confirmation, there may be unforeseen mechanical movements, causing damage to cutting tools, machine tools, workpieces or injuries to personnel.

2. Programs, parameters, and macro variables are all stored in storage within the system. Normally, the storage content is not lost even if the power is on / off. However, in the case of abnormal operation of the system, or accidental power failure in the process of setting parameters, may restore the system parameters to the default parameters of the factory setting and even cause the loss of these data and parameters.

To avoid the above situation, please back up the data regularly and keep it properly, so as to ensure the rapid recovery of the deleted data.

3. Operation shall be fully confirmed after the correctness of the input data. If the data used is incorrect, there may be unexpected mechanical movements, causing to tools, tools, workpieces, and injuries.

4. Confirm that the set spindle speed and feed speed are appropriate. Set the maximum spindle speed and feed speed by the user parameters. If the spindle speed and feed speed are not set properly, damage may cause tools, machine tools and workpiece and personnel injury.

5. When using the tool compensation function, the compensation direction and compensation amount should be fully confirmed. If the data used is incorrect, there may be unknown mechanical movements, causing damage to cutting tools, machine tools, workpieces and personnel injuries.

6. The manufacturer has set the best values for the CNC and PLC parameters. Generally, there is no need to change. When the parameter value needs to be adjusted, it must be modified on the basis of fully understanding the meaning of the parameter. If the parameters are set correctly, it may cause damage to cutting tools, machine tools, workpieces and personnel injuries.

 **[pay attention to]**

1. Some functions are implemented at the request of the machine tool manufacturer, and when using these functions, refer to the instructions provided by the machine tool manufacturer for a detailed usage of the functions and some relevant considerations.

2. When programming, be familiar with and fully understand the contents of the operation instructions. When setting the coordinate system, non-linear insertion and positioning, and the end surface constant speed limit control programming, the appropriate command value must be set to ensure that the machine tool moves correctly, so as not to cause damage to cutting tools, machine tools, workpieces and personnel injury.

3. During manual operation of the machine tool, master the position of the tool and workpiece, and confirm whether the choice of moving shaft, direction and feed speed is

wrong. If the operation is wrong, it may damage the cutting tools, machine tools, workpieces and personnel injury.

4. For the machine tool that needs to manually return to the machine zero, after the power supply is on, it must manually return to the machine zero, otherwise the machine tool may appear unknown action, so that the tool, machine tool, workpiece damage and personnel injury.

5. When using the hand wheel feed, if the ratio of 100 is selected, the tool and worktable will move faster, so special attention should be paid. Otherwise, it may damage the cutting tools, machine tools, workpieces and personnel.

6. Configured parameters, and recommend backup processing for later recovery.

7. The CNC system must be connected to the ground wire.

2. Warnings and attention related to the operations

 **[warn]**

1. When running the machine tool manually, we should grasp the current position of the tool and workpiece, and fully confirm that there is no error in the choice of movement shaft, movement direction and feeding speed. Incorrect operation can damage tools and workpieces or cause injuries to operators.

2. For the machine that needs to manually return to the reference point, manually return to the reference point after the power supply is connected. If the machine is operated without manually returning to the reference point, it will lead to unexpected operation of the machine. Also, the trip detection failed before the manual return to the reference point. This condition can damage cutting tools, machine tools and workpieces, or cause injuries to operators.

3. When feeding the hand wheel, if a larger ratio such as 100 is selected, the movement speed of the tool and the rotary table will be accelerated. Therefore, cutting tools, machine tools and workpieces will be damaged, or injuries to operators will be caused.

4. During thread cutting, rigid tapping, or other tapping, when the macro variable fails or when the ratio fails, it will damage the tool, machine and workpiece, or injuring the operator.

5. In principle, do not perform zero / preset operation when the machine tool is in program execution. If the zero point / preset operation is performed in the program execution, the machine tool will perform the unexpected action during the subsequent program execution. This condition can damage cutting tools, machine tools and workpieces, or cause injuries to operators.

6. Manual intervention, mechanical locking, or mirroring can cause the offset of the artifact coordinate system. Therefore, the coordinate system must be carefully confirmed before executing the procedure. If the procedure is executed without considering the displacement of the work piece coordinate system, the unexpected operation of the machine tool will result. This condition can damage cutting tools, machine tools and workpieces, or cause injuries to operators.

7. Using the soft operation panel, you can specify the operations that are not supported by the machine tool panel from the MDI panel, such as changing the mode, changing the multiplier value, assigning JOG input instructions, etc. Therefore, if you do not pay attention to operate the MDI panel key, it will lead to the machine tool unexpected operation. This condition can damage cutting tools, machine tools and workpieces, or cause injuries to operators.

8. When the RESET key is pressed, the program in the execution is stopped. RESULTS The servo shaft will also stop, but the RESET key MDI may not function due to the panel failure and other reasons. In order to ensure safety, when the emergency stop motor needs, do not press the RESET button, but use the emergency stop button.

 **[pay attention to]**

1. If manual intervention is performed during program execution, the movement path will be different when starting the machine, depending on the state. Therefore, after manual intervention, the status of manual absolute switch, parameters and absolute / incremental instructions shall be confirmed before restart the machine.

2. Air operation is usually used to confirm the operation performance of the machine tool. During empty operation, the machine runs at an empty operation speed, which is different from the feed speed specified by the program. Sometimes the machine moves move under fast movement.

3. For the instructions in the MDI mode, either the tool radius compensation or the knife tip radius compensation will not be executed. Please pay attention to the moving path. In particular, under the tool radius compensation mode (M series) or the tip radius compensation mode (T series), if an instruction is input from MDI in the automatic operation, the automatic operation. See the respective functional description for more details.

4. If the machine tool stops processing, after the processing program is modified, inserted or deleted, and then continue to perform the program, it will lead to the unexpected operation of the machine tool. It is very dangerous to modify, insert or delete the processing procedure in use. In principle, do not do it without authorization.

3: Programming warning and attention

[warn]

1. If the coordinate system is not set correctly, even if the movement instruction of the program is correct, it will lead to the unexpected operation of the machine tool. This condition can damage cutting tools, machine tools and workpieces, or cause injuries to operators.

2. Positioning with non-line interpolation method When the non-line interpolation method is used (i. e., moving in the nonlinear way between the beginning and the end point), the path of the tool must be carefully confirmed before programming. Since positioning is performed under rapid movement, if the tool touches the workpiece, it can damage the tool, machine tool and workpiece, or injure the operator.

3. Special attention should be paid to the speed of the rotation axis when compiling procedures such as polar coordinate insertion (T series) or normal direction control (M series). Inappropriate program, will make the speed of the rotating axis becomes too fast, or due to the improper installation method of the work, the work will fall off due to centrifugal force. This condition can damage cutting tools, machine tools and workpieces, or cause injuries to operators.

4. From British input to metric input, or from metric input to British input, does not convert the work piece origin bias value, various parameters and current position and other units. Therefore, the units of such data must be fully confirmed before running the machine tool. Attempts to operate with incorrect data can damage tools and workpieces, or cause injuries to operators.

5. When the current position of the work piece coordinate system of the constant control shaft is near zero, the speed of the spindle will become too fast, so the maximum speed must be correctly specified. Without properly specified maximum speed, damage to the tool, tools and workpieces, or injury to the operator.

6. For the machine tool that needs to manually return to the reference point, it must manually return to the reference point after the power supply is turned on. Travel detection fails before manual return to the test site. Note that if the trip fails, there will be no alarm to damage the tool, machine tool and workpiece or injure the operator.

[pay attention to]

1. Absolute / incremental mode

If a program written in absolute value is executed in incremental mode, or a program written in incremental value is executed in absolute mode, it will lead to unexpected operation.

2. Plane selection

The circular arc insertion / spiral insertion / fixation cycle, if the specified plane is not correct, will lead to the unexpected operation of the machine tool. See the respective functional description for more details.

3. Compensation function

If the instruction of the mechanical coordinate system or the return reference point is specified in the compensation function, the compensation will be temporarily cancelled, resulting in the unexpected operation of the machine. Therefore, cancel the compensation function mode before issuing any of the above instructions.

catalogue

The first overview article

1. Description of the instructions

This specification consists of the following sections.

I.summary

The overview describes the composition of this specification, the product profile, technical specifications, relevant instructions, and precautions for reading the instructions.

II.programming

The programming section describes the format, interpretation, limitation of the program using the NC language for each function.

III.operate

In the operation section, the manual operation and automatic operation of the machine tool, the input / output method of the data, and the editing method of the program are described.

IV.device attachment

In the equipment connection section, the system installation diagram and installation size, connector interface layout (feed motor interface, input signal interface for machine tool, spindle interface, handwheel interface, auxiliary panel interface, general output and input interface, encoder input interface, RS485 interface), connection between system and equipment, machine control I / O interface and other parameters.

V.appendix

The appendix describes the lists of parameters, instruction value range, alarm, etc.

2. Product introduction

2.1 Product Introduction

Our company independently developed the milling machine CNC system. Can be adapted to the milling machine, machining center, drilling and milling machine and other equipment.

The system adopts 32-bit high performance CPU and super-large scale programmable device CPLD, real-time monitoring and hardware insertion technology to ensure the high speed of μ m level accuracy of the system, if the absolute value driver and motor developed by our company, the effect will be more outstanding. Open PLC makes the logic control function more flexible and powerful. At the same time, it can flexibly set the name and number of axes to be displayed. The system has a perfect macro program to realize custom G code. The system supports 6 feed axes, 0.1 μ m control accuracy, which significantly improves the machining efficiency and accuracy.

2.2 Technical specifications

1. Number of control axes: the maximum number can be 6 axes [X, Y, Z, 4th, 5th, 6th].(Depending on the product configuration, this specification is subject to the actual control system product)

2. Number of linkage axes: the maximum number can be 6 axes.(Depending on the product configuration, this specification is subject to the actual control system product)

3. Display: 8 inches, resolution: 800 * 600 pixel.

4. Metric input range (G 21):-999999.999mm ~999999.999mm, minimum instruction unit 0.001mm.

5. British input range (G20):-99999.9999inch ~99999.9999inch, minimum instruction unit 0.0001inch.

6. Fast ratio: F0,25%, 50%, 100% four real-time adjustment.

7. Fast moving / cutting feed speed: up to 240m / min.

8. Feed / manual feed rate: 0~150% 16-level real-time adjustment.

9. Spindle ratio: according to the selected panel, can support 0~150% or 50%~150% two real-time adjustment.

10. Spindle constant speed limit control, tapping cycle, rigid tapping cycle.

11. Insertion method: straight line insertion, circular arc insertion.

12. Cutting feed: front acceleration and deceleration linear, rear acceleration and deceleration linear.

13. Fast movement mode: front acceleration and deceleration linear type, rear acceleration and deceleration linear type.

14. System forward-looking function: NC program can be read, so that the small line segment inserted high speed smooth, more suitable for the processing of various workpieces.

15. The speed and time of the acceleration and deceleration and whether the hand wheel is tested can be set by the parameters.

16. Rapid positioning can be selected as linear and non-linear types through parameters.

17. Support tool radius compensation, tool length compensation, pitch error compensation and reverse clearance compensation.

18. RS232: programs and parameters, support PLC and system version upgrade.

19. USB: U disk file operation, support PLC and system version upgrade.

20. Including emergency stop, hardware travel limit, software travel check, data backup and recovery and other security functions.

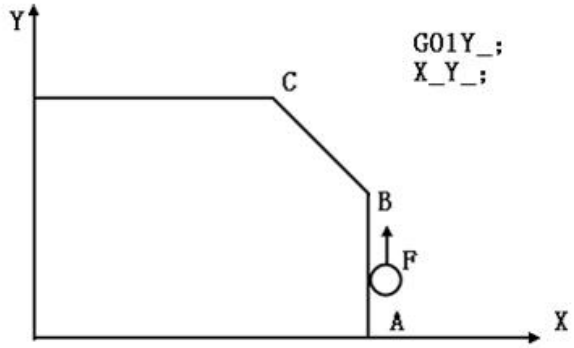
Chapter 2, Programming chapter

1 Programming overview

1.1 Insertion function

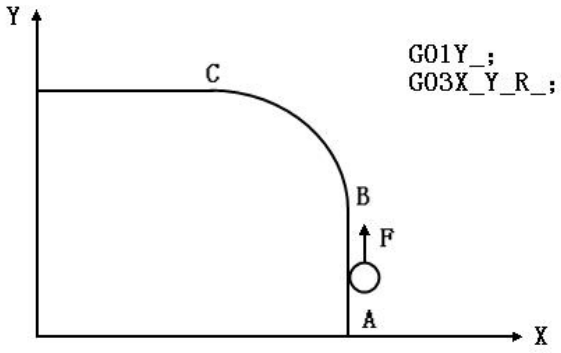
Insertion: the function of moving the tool along a straight line or arc forming the shape of the workpiece and calculating the moving position.

Linear insertion: the tool moves along the line as shown in Figure 2-1:



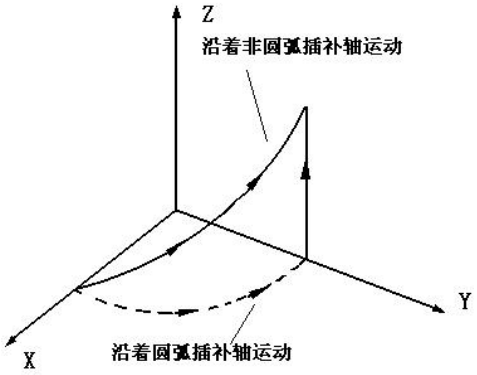
graph 2-1

Arc insertion: the tool moves along the arc, as shown in Figure Figure 2-2:



graph 2-2

Spiral line insertion: The spiral trajectory is formed by assigning a non-circular arc interpolation axis to move synchronously with other circular arc insertion axes, as shown in Figure 2-3:



graph 2-3

The instructions G01, G02 and G03 are called the preparation function, which is used to control the interpolation movement of the machine tool, and then control the machine tool to complete the interpolation movement according to the position coordinates specified in the program.

pay attention to:

Some machine tools are table moves rather than tool moves. The specification assumes the tool is move relative to the workpiece.

1.2 Feed function

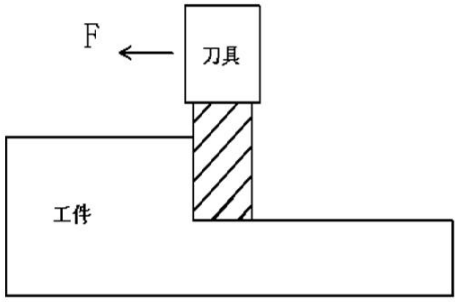
The feed, specify speed is feed, including fast movement and cutting feed. The feed speed can be specified with actual values, indicated by F.

instruction format:

F ○○○○;

The unit is generally "mm / min" or "mm / r". If the specified tool is moved at 150mm / min, the programming format is F150.0.

When the command quickly positions the G00, the system moves quickly according to the speed set by the parameter. F instruction ignore. As shown in Figure Figure 2-4:



graph 2-4

1.3 Spindle function

The speed of the tool relative to the workpiece is called the cutting speed. The CNC machine tool uses the spindle speed to command the cutting speed. The function of the relevant spindle speed is called the spindle function.

instruction format:

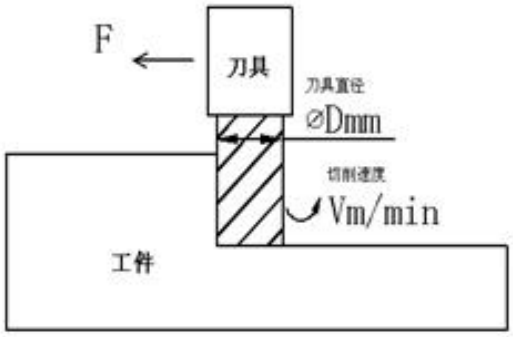
S ○○○○;

The spindle speed is generally selected according to the cutting speed, and the formula is: Vc

$$N = \frac{1000 \times Vc}{\pi D}$$

Where: D is the tool diameter and Vc is the cutting speed.

The operation panel of the CNC system has the spindle speed adjustment (ratio) switch, which can adjust the spindle speed during the machining process. As shown in Figure Figure 2-5:



graph 2-5

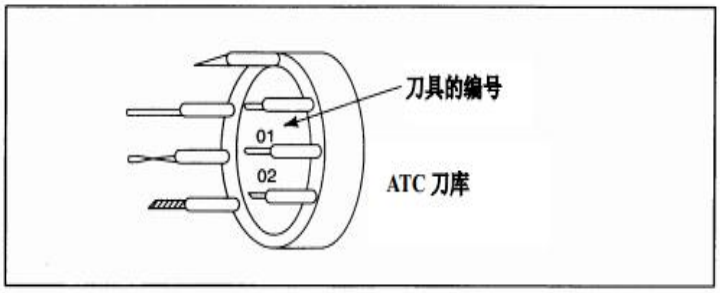
1.4 Tool function

When drilling, boring, milling and other processing, proper tools must be selected. Give each tool a number, and when instructing a different number in the program, select the corresponding tool.

The T code is usually used with the M06. When number 01 is designated as a drill bit: the tool is placed at position 01 of the knife library, the tool can be selected by instruction T01, and this function is called the tool function.

instruction format:

T○○;



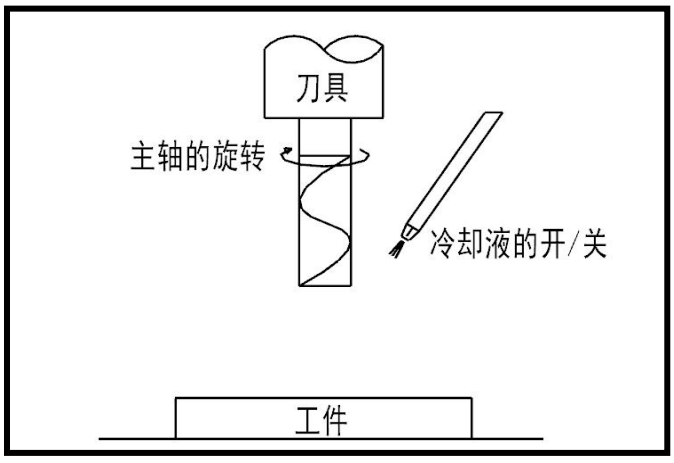
graph 2-6

1.5 Auxiliary function

The function of instructing the start and stop operation of the machine parts is called the auxiliary function.

Usually, the function is specified with the M code. The movement instruction and the auxiliary function instruction can be executed simultaneously in the same program segment. M code has only one valid in a segment.

For example, when instructing M03, the spindle rotates in a clockwise direction at the specified spindle speed.



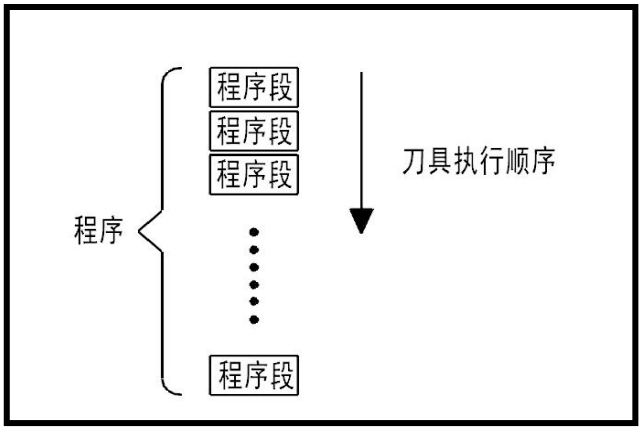
graph 2-7

2 Program structure

A set of instructions sent to the CNC for running the machine tool is called a program.

Following the specified instructions, the tool moves along a line or arc at a certain speed and the spindle motor rotates or stops following the instructions.

In the program, instructions are given in the actual order that the tool is moved.



graph 2-8

2.1 Procedures

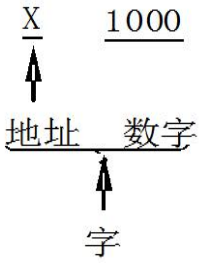
A program is composed of a number of program segments, a group of single step of sequential instructions is called the program segment, the program segment is the basic unit of the processing program. The program segment consists of one or more words and ends with a program segment end character (EOB, ISO code LF, EIA code CR, ";") on the screen. As shown in Table 2-1:

Table 2-1 Procedure examples

O0001;	Program number
N01 G91 G28 Z0;	Relative mode of the Z-axis back to the
N02 G28 X0 Y0;	Relative mode X and Y return to zero
N03 T01 M06;	Change the no. 01 knife
N04 G90 G54 G00 X0 Y0 S1000 M03;	Absolute programming mode, with the origin of G54 coordinate system as the programming origin, quickly locate to
N05 G43 Z100.0 H01 M08;	Establish the tool length compensation,
N06 G98 G81 X0 Y0 Z-5.0 R3.0 F120;	Establish drilling fixed cycle with a depth of 5 mm
N07 X12.5 Y-12.5;	Drill the second hole
N08 G80;	Fixed cycle cancellation
N09 M05 ;	Spindle stop
N10 M09 ;	Coolant close
N11 M30;	The program ends and returns to the

2.2 words and address

The word is the basic element of a program segment. It consists of the address character and the value behind it (sometimes with + and-symbols before the value).



The address is a letter in the English letter (A ~Z). It specifies the meaning of the subsequent value.

Table 2-2 Basic address

function	address	meaning
Program number	O	Program number
SN	N	SN
g function	G	Specify action states (line, arc, etc.)
Size word	X ， Y ， Z	Axis move (rotation) instruction
	A ， B ， C	
	R	Arc radius
	I ， J ， K	The center coordinate of the arc
feed rate	F	Feed speed is specified
Spindle function	S	Spindle speed specified
tool function	T	Specify the number of knives
additional function	M	Specification of ON / OFF for control machine tools
suspend	P ， U ， X	The designation of the pause time
tool offset	H ， D	Specifies the tool offset number

Program number specified	P	Specifies the subprogram number
number of replication	P	Number of repetitions of the subroutine
parameter	P ， Q ， R	Specifies the serial number of the repeat part of the program, etc

2.3 Basic address and instruction value range

The basic address and instruction value ranges are shown in Table 2-3.

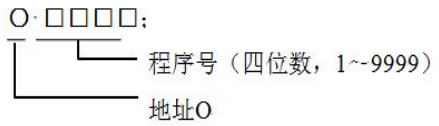
Table 2-3 Basic address and instruction range

function	**address**	**Directive range**
Program number	O	1～9999
SN	N	1～9999
g function	G	0～99
Size word	X ，Y ，Z ，I ， J ， K ， R	± 999999.999 mm
Each bonus	F	From 1 to 15,000 mm / min
Spindle function	S	0～9999
tool function	T	0～9932
additional function	M	0～999
suspend	X ， U ， P	0~9999.999 seconds
Program number is specified, and the number of repetitions is given	P	1～9999

tool offset	H ， D	0～399
SN	P ， Q	1～9999

2.4 Program number and program segment

The number used to distinguish each program segment is called the serial number, and the number used to distinguish each program is called the program number. The serial number is expressed by Nxx, and the program number is composed of O and the subsequent four digits.



The program starts with the program number and ends with M30 or M99. The basic program segment structure is shown in Table 2-4:

Table 2-4

1	2	3	4	5	6	7	8	9	10	11
N	G_	X_	Y_	Z_	I_ J_ K_	S_	F_	T_	M_	；
SN	standard have exploit ability	a note of the scale in gongchepu cun character				host axle exploit ability	advance To exploit ability	knife With exploit ability	assist help exploit ability	bear bind tally

2.5 Main program and subprograms

When the processing mode with the same function appears in the program, the same mode can be compiled into a program that becomes a subroutine. The original program became the main program. In general, the machine tool operates according to the instructions of the main program. However, the control goes to the subprogram in the main program. Control returns to the master program when the subprogram encounters an instruction returned to the master program.

Subprogram writing format:

O0000; child program No

……

……

M99; Subprogram call ends

Subprogram calling format:

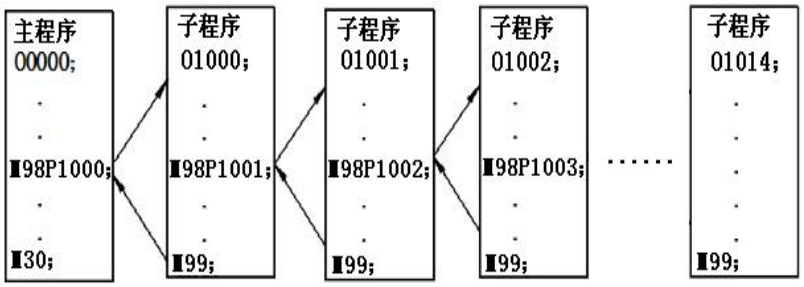
M98 P□□□OOOO ；

perhaps:

M98 POOOOL□□□；

M30；

Where □□□ is the number of calls, and O 0000 is the program number. When the number of times is not specified, the subprogram calls it only once when the master program calls the subprogram. Up to 15 levels can be nested, as shown in Figure Figure 2-8.



graph 2-9

2.6 End of the procedure

In the execution program, the program ends if the program end code is detected: M02, M30, or M99.

If the M02 code, the program ends but does not return to the beginning. If M30, to return to the beginning of the program (automatic mode).

If M99, M02, M30 are at the end of the subbroutine, return to the program calling the subbroutine and continue to execute the following program segment.

In order to meet the processing needs, you can insert a pause in the program segment. When the M00 code is running, the program pauses and press the cycle start to continue the next line of instructions. M01, the function is the same as M00, but only M01 is valid when the operation panel selection stop switch is on.

3. Basic knowledge of programming

3.1, and the control axis

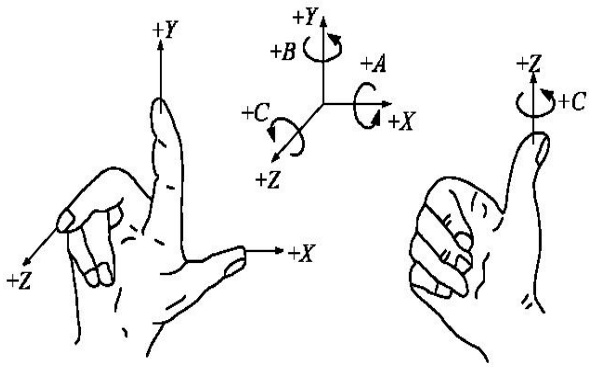
3.1.1 Control axis number

Table 2-5 System Control Shaft

Basic control axis	6 Axis (X, Y, Z, 4,5,6)
Basically	6 Axis (X, Y, Z, 4,5,6)

3.1.2 Axis name

The names of the 3 basic axes are X, Y, and Z. The coordinate axis of the tool motion parallel to the main axis of the machine tool (transfer the cutting force) is the Z axis, and the direction of the tool away from the workpiece is the positive direction (+ Z). The X axis is horizontal and perpendicular to the Z axis and parallel to the assembly face of the workpiece. When the Z axis is horizontal, look along the back end of the tool spindle to the workpiece, and the right direction is the positive direction of X. The above positive direction is the tool relative to the workpiece movement. After determining the positive direction of the X and Z axes, the positive direction of the Y axis can be determined according to the right cartesian coordinate system of the right hand, that is, in the Z-X plane, from + Z to + X, the right helix should advance along the + Y direction. The right-hand cartesian coordinate system is shown in Figure 2-10:



graph 2-10

3.2 Set the unit

3.2.1 Minimum set unit, minimum mobile unit and maximum set unit

The setting unit is combined by the minimum setting unit (input unit) and the minimum moving unit (output unit).

(1) Minimum set unit

The minimum sets the minimum unit of the tool movement given in millimeters (mm), inch (inch), or deg (angle).

(2) Minimum mobile unit

The minimum moving unit is the minimum unit of the machine movement. The minimum unit is given in millimeter (mm), inch (inch), or deg (angle). There are three set units (Table 2-6), available through NO.1804 Parameter for selection setting.

Table 2-6

Set the name of the unit	Minimum setting unit	Minimum moving unit
IS-A	0.01mm	0.01mm
	0.001inch	0.001inch
	0.01deg	0.01deg
IS-B	0.001mm	0.001mm
	0.0001inch	0.0001inch
	0.001deg	0.001deg
IS-C	0.0001mm	0.0001mm
	0.00001inch	0.00001inch
	0.0001deg	0.0001deg

The minimum mobile unit is metric or English system, according to the machine tool, prior through the parameters (No.1804) is set for selection.

The minimum set unit is metric input or English input, which can be switched through G code (G20, G21) or set parameters.

(3) Maximum setting unit: do not set the instructions that exceed the maximum set unit.

Table 2-7

Set the name of the unit	Minimum moving unit	Maximum mobile travel unit
IS-A	0.01mm	±999999.99mm
	0.001inch	±99999.999inch

	0.01deg	±999999.99deg
IS-B	0.001mm	±999999.999mm
	0.0001inch	±99999.9999inch
	0.001deg	±999999.999deg
IS-C	0.0001mm	±99999.9999mm
	0.00001inch	±9999.99999inch
	0.0001deg	±99999.9999deg

The minimum mobile unit is metric or English system, according to the machine tool, prior through the parameters (No.3804) The setting is selected.

The minimum set unit is metric input or English input, which can be switched through G code (G20, G21) or set parameters.

pay attention to:

The unit of the rotation axis cannot be converted by British / metric.

The metric and English systems cannot be mixed together.

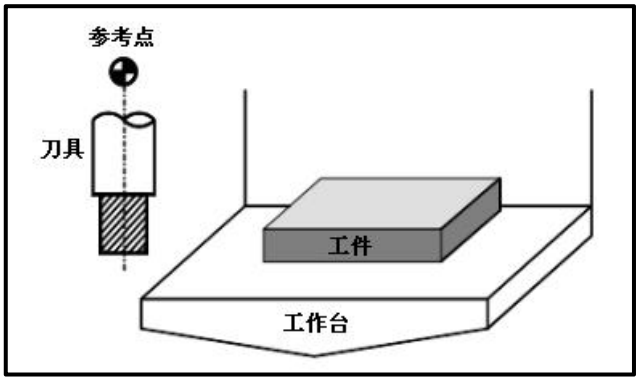
3.3, and the coordinate system

3.3.1 Point of Reference

On the CNC machine tool, there is a special position, usually change the knife or set the coordinate system, this position is called the reference point. The reference point is a fixed point under the machine tool coordinate system set by the machine tool plant.

Returning the functional tool with a reference point can be easily moved to the position.

In general, the reference point of the CNC system coincides with the machine tool zero point.



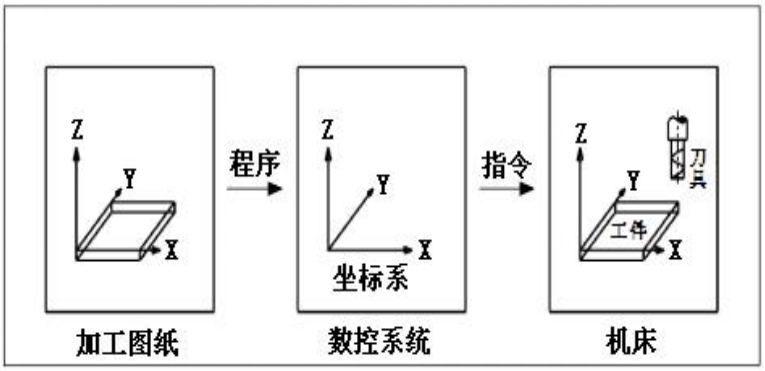
graph 2-11

3.3.2, Machine tool coordinate system

A specific point on the machine tool as the processing reference is called the machine tool zero (the machine tool origin is set by the machine tool manufacturer for each machine tool). The coordinate system set by the zero point of the machine tool as the origin is called the machine tool coordinate system, and its position is usually at the maximum limit of each coordinate axis. After power up, perform a manual return reference point to set the machine coordinate system. Once the machine coordinate system is set, it remains unchanged until the power is turned off.

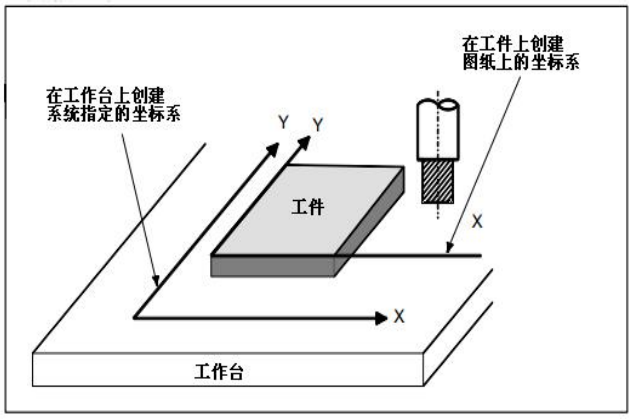
3.3.3. Work piece coordinate system

The work coordinate system is the coordinate system used by programmers in programming and processing, and it is the reference coordinate system of the program. The position of the work coordinate system takes the machine tool coordinate system as the reference point. Generally, six working coordinate systems can be set in a machine tool. Take a certain point on the workpiece pattern as the origin of the working coordinate system, which is called the origin of the workpiece. During processing, the workpiece is installed on the machine tool along with the fixture. At this time, the distance between the origin of the workpiece and the origin of the machine tool is measured. This distance is called the workpiece origin offset, as shown in Figure 2-11:



graph 2-12

When the workpiece is installed on the workbench, the relative relationship between the two coordinate systems is formed.

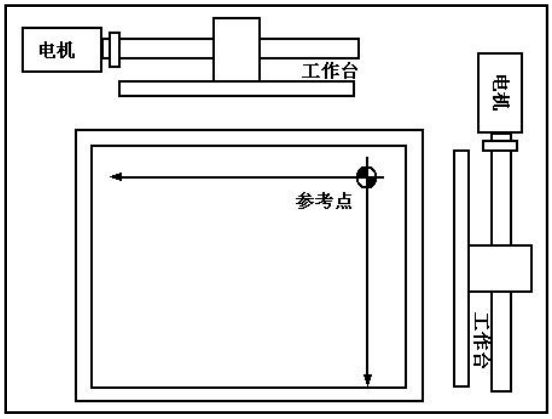


graph 2-13

The tool is moved on the coordinate system specified by CNC, according to the "instruction program written by the coordinate system on the machining drawing", and processes the workpiece into the shape required on the drawing. Therefore, in order to correctly process the workpiece according to the drawings, the two coordinate systems must be set in the same position.

3.4 Maximum itinerary

Install a finite bit switch at each end of the machine shaft to prevent the tool from moving out of the endpoint. The range where the tool can move is called the stroke. As shown in Figure 2-14:



graph 2-14

In addition to setting the trip with the limit switch, you can also pass the soft limit parameter NO.1225-1230 to set the maximum forward stroke of each axis, parameter NO.1234-1239 to set the maximum negative stroke for each axis.

3.5 Coordinate value and size

There are two ways to move the tool; absolute and incremental value instructions.

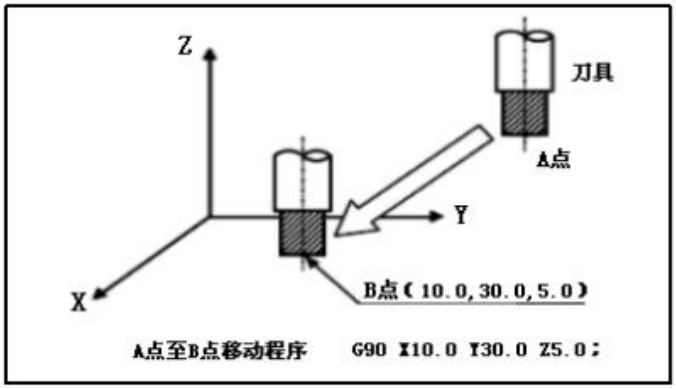
In the absolute value instruction, X_Y_Z_ is the coordinate value of the programming endpoint; while in the incremental value instruction, X_Y_Z_ is the programming movement distance.

G90 and G91 are used for programming the instruction at absolute or incremental values, respectively.

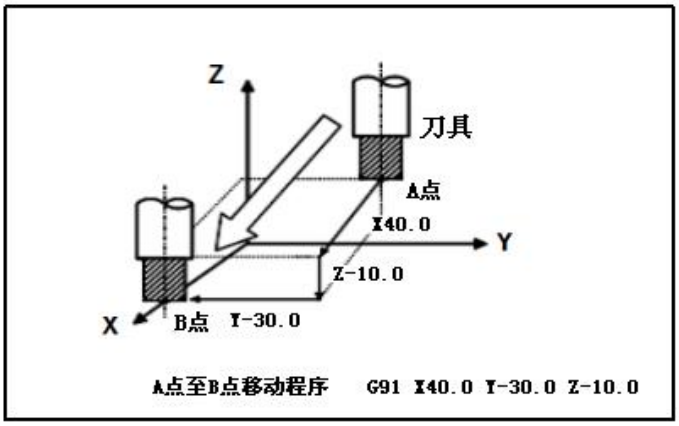
When the system is reset, the G90 / G91 mode is adjustable, through the user parameter NO.3803 Setting-up.

Absolute instruction: G90 X_ Y_ Z_

Increment instruction: G91 X_ Y_ Z_



graph 2-15



graph 2-16

instruction format:

G20; inch input

G21; mm input

The G code must be specified in a separate program segment at the beginning of the program before setting the coordinate system.

After specifying the G code for the British / Metric conversion, the unit of input data switches to the minimum British or metric input increment unit of the increment system IS-B or IS-C.

The units of the input angular data remained the same.

By imperial / Metric conversion, the following numerical units will change:

The feed speed specified by the F code;

The instruction value of the relevant position;

Offset amount of work piece origin;

Tool offset amount;

The value of the hand pulse generator per moment degree;

The amount of incremental feed movement;

Some parameter values.

explain:

1．G20 and G21 during be execution.

2． When the imperial input (G20) switches to the metric input (G21) or switches to the opposite direction, the tool compensation value must be preset according to the minimum input increment unit.

3．Through the user parameter NO.3805 Set the mode after charging to G20 or G21.

3.7 decimal al programming

Digital values can be entered with decimal points. A decimal point can be used when entering the distance, time, or speed. The following address can specify the decimal point: X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, and F.

Programming command	actual numerical value
Z15.0；	15mm
Z.015；	0.015mm
Z15；	15mm
F10；	10mm/r 10mm/min

4. Preparation function (G function)

The number following the address G determines the meaning of the program paragraph instruction. The G codes fall into the following two categories:

Table 2-8 Implications of modal and non-modal codes

type	meaning
Non-modal G code	The G code is valid in the current program segment only.
Modal G code	This G code is valid until the instruction sets the other G codes.

example:

Mode G code in group 01 at G01 and G00.	
G01 X100 Y100 F200；	G01 valid
Z5	G01 valid
G00 Z50；	G00 valid

Table 2-9 Preparatory function (G function)

G code	group	function
◤G00	01	Positioning (fast)
◤G01		Line insert (cutting feed)
G02		Cclockwise clockwise or clockwise CW
G03		Counterclockwise or counterclockwise screw-in CCW
G04	00	suspend
G 15	17	Polar coordinate are cancelled
G 16		Polar coordinate instruction is on
◤G17	02	The XY plane selection
◤G18		The ZX plane selection
◤G19		The YZ plane selection
G20	06	Inch input
G21		Millimeter input
G28	00	Returns the reference location
G30		Return to reference position 2,3, and 4
G31		Jump function

◤G40	07	Tool radius compensation is cancelled
G41		Left-side tool radius compensation
G42		Right-side tool radius compensation
G43	08	Forward tool length compensation
G44		Negative tool length compensation
◤G49		Tool length compensation is cancelled
◤G50.1	22	Mirrored programming cancellation
G51.1		Mirrored programming is effective
G52	00	Local coordinate system setting
G53		Select the machine tool coordinate system
◤G54	14	Work-piece coordinate system selection 1
G55		Work-piece coordinate system selection 2
G56		Work-piece coordinate system selection 3
G57		Work-piece coordinate system selection 4
G58		Work piece coordinate system selection 5
G59		Work piece coordinate system selection 6
G65	00	Macro program call
G66	12	Modal macro calls
◤G67		The modal macro call is out
G73	09	High-speed flogging cycle
G74		Reverse (left rotation) tapping cycle
G76		Fine boring cycle
◤G80		Fixed cycle cancellation
G81		Drilling ring, point drilling ring
G82		Drilling ring and boring circulation
G83		Discharge drilling cycle
G84		Attack wire cycle
G85		Boring circulation
G86		Boring circulation
G87		The boring cycle
G88		Boring circulation
G89		Boring circulation
◤G90	03	Absolute value programming

◤G91		Incremental value programming
G92	02	Set the work piece coordinate system
◤G94	05	Feed every minute
G 95		Each transfer to give
G98	10	The fixed loop returns to the initial point
G99		The fixed loop returns to the R point

Description: The modal G code in Table 2-9 is represented by ◤.

There can be several G codes in the same program segment, but the same group G code cannot appear in the same program segment, otherwise the machine tool alarm. Under the fixed cycle mode, the G code of any 01 group will automatically cancel the fixed cycle mode, and become the G80 mode.

4.1 Rapid positioning (G00)

G00 instruction, the tool moves to a position in the artifact coordinate system specified by the absolute or relative value instruction at a fast moving speed.

The F0,20%, 50%, 100% can be adjusted through the panel.

G00 is fast moving for positioning, but only for empty travel, and cannot be processed in the middle.

instruction format:

G00 X_ Y_ Z_;

X_ Y_ Z_: the absolute value programming is the coordinate value of the end point;

When programming the relative values, it is the distance from which the tool moves.

explain:

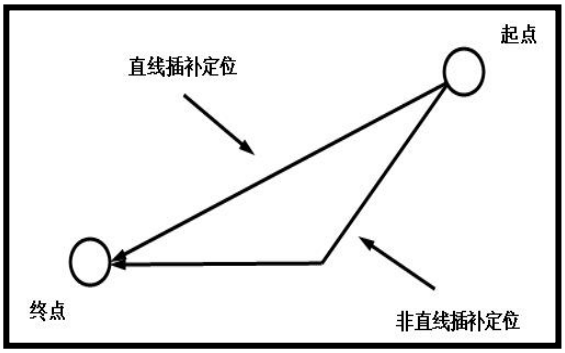
According to the parameter NO.3846 To select any of the following tool paths.

1． Non-line interpolation positioning

The tool can be positioned to each axis at a fast moving speed, and the tool path is generally not straight.

2． Straight line insertion and complement positioning

The tool moves along a straight line to the specified point, the tool positions in the shortest positioning time, and the positioning speed does not exceed the fast movement speed of each axis. However, the path is not the same as the line interpolation (G01).



graph 2-17

Through the user parameter NO.3801 Select the post mode to be G00 or G01.

Fast movement speed in the G00 instruction, given by the parameter NO.0502 Setting.

The setting of linear, nonlinear F0 speed speed and 100% fast speed in each axis G00 mode are detailed in user parameter —— speed F.

4.2 Linear insertion (G01)

Under the G01 instruction, the tool moves along a straight line and is a modal instruction.

instruction format:

G01 X_ Y_ Z_ F_;

X_ Y_ Z_: the absolute value programming is the coordinate value of the end point;

When programming the relative values, it is the distance from which the tool moves.

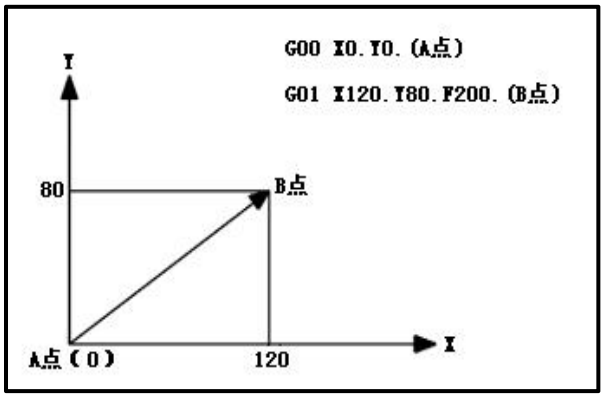
F: Feed speed of the tool.

explain:

The tool moves along a line to the specified position at the feed speed specified by F. The feed speed specified by F is valid until the new F value is specified, so there is no need to specify an F value for each segment.

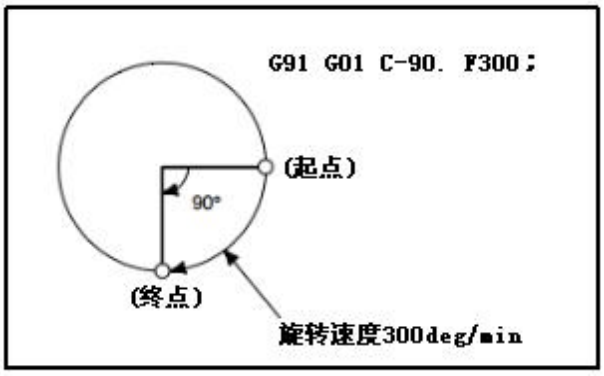
After the system restart, when the first G01 instruction appears in the program, the F value must be specified, otherwise it will be executed at F500mm / min feed speed.

give an example:



graph 2-18

The feed speed of the rotation axis:



graph 2-19

4.3 Arc insertion (G02, G03)

instruction format:

Arc of the XY plane:

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

Arc of the ZX plane:

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Z\_\ X\_\left\{\begin{matrix}I\_\ K\_\\R\_\end{matrix}\right\}F\_\ ;$$

Arc of the YZ plane:

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

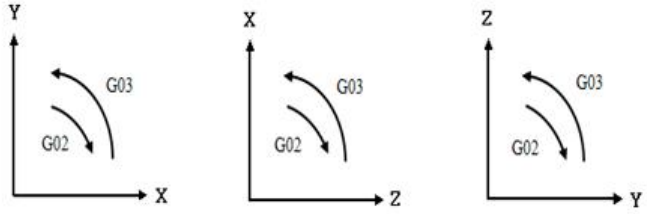
Description of instructions:

instruct	description
G17	The XY plane selection
G18	The ZX plane selection
G19	The YZ plane selection
G02	Arc ate, clockwise
G03	Arc ate, counterclockwise
X	The movement of the X-axis or its parallel axis
Y	Movement of the Y-axis or its parallel axis
Z	Movement of the Z-axis or its parallel axis
I	Distance from the beginning of the X-axis to the center of the arc (with symbols)
J	Distance from the beginning of the Y-axis to the center of the arc (with symbols)
K	Distance from the beginning of the Z axis to the center of the arc (with symbols)
R	Arc radius (with symbol)
F	The feed velocity along the circular arc

explain:

1．The direction of the circular arc interpolation

In the rectangular coordinate plane XY, when viewing the XY plane from the positive direction of the Z axis, the clockwise (G02) and counterclockwise (G03) directions of the XY plane are as follows, as shown in Figure 2-20:



graph 2-20

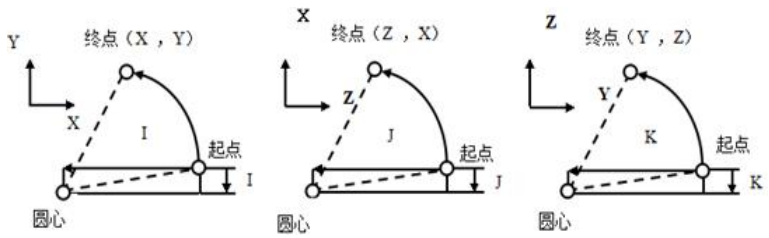
2． The amount of movement in the circular arc

The end point of the arc is specified by the addresses X, Y, or Z, expressed as absolute or incremental values according to G90 or G91. The increment value symbolically specifies the distance from the start of the arc to the end.

3． Distance from the starting point to the center of the arc

Use the addresses I, J, and K to specify the arc center positions of the X, Y, and Z axes, respectively. The values after I, J, and K are vector values from the beginning to the arc, and regardless of G90 or G91, the value is always an incremental value, as shown in Figure 2-21.

I, J, and K must specify their symbols according to the direction.



graph 2-21

I0, J0 and K0 can be omitted.

If the difference between the radius values of the start and end points exceeds the parameter NO. For the value set in 0510, there will be an alarm issued.

4． The whole circle instructions

When X, Y, and Z are omitted (the end point is the same as the starting point), and the center is designated by I, J, K is 360° arc (whole circle).

5． Arc radius

The distance between the arc and the center of the circle can be specified by the arc radius R to replace I, J, and K. When specified over the 180° arc, the radius must be specified with a negative value. If X, Y, Z are omitted, the end and the beginning are the same, and R is specified, the compiled arc is 0°.

Arcs less than 180°: G02 X 70.Y30.R60.F400.

Arcs greater than 180°: G02 X 70.Y30.R-60.F400.

There are two tracks as shown in Figure 2-22:

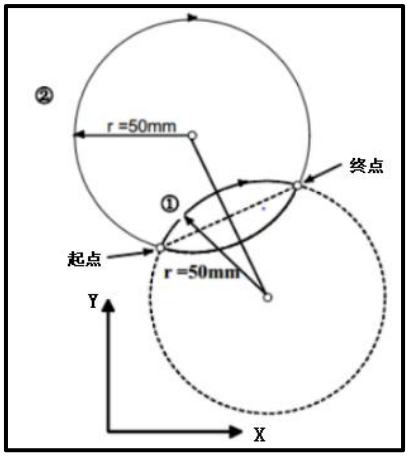
give an example:

①Arc (180°)

G91 G02 X60.0 Y55.0 R50.0 F300；

②Arc (180°)

G91 G02 X60.0 Y55.0 R-50.0 F300；



graph 2-22

6．feed rate

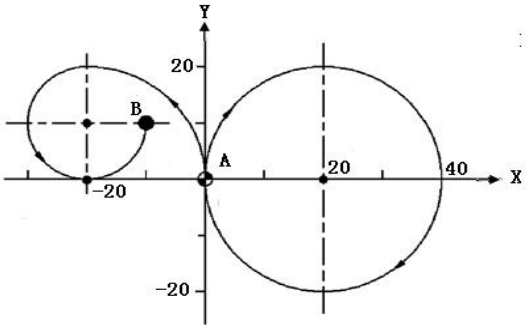
The feed speed of the arc insertion becomes the cutting feed speed specified by the F code. The feed velocity along the arc (the tangent direction velocity of the arc) is controlled as the error between the specified feed speed and the actual tool feed speed. However, this specified speed is the speed determined along the arc after the tool radius compensation.

pay attention to:

1． If I, J, K, and R are specified simultaneously, the arc specified by R is preferred, and I, J, and K are ignored.

2．If the arc of the center angle is close to 180 degrees is specified by R, the calculation of the center position will produce error. In this case, the arc center is designated with I, J, and K.

give an example:

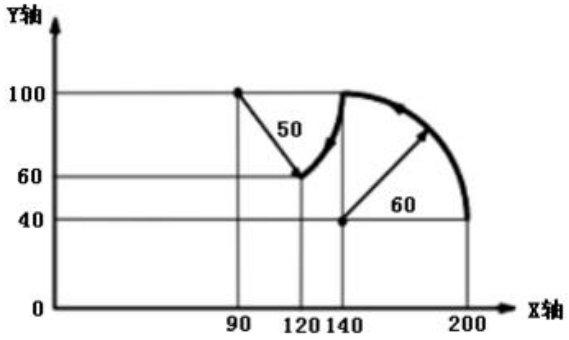


graph 2-23

Example procedure 1:

G90G54G17G00X0Y0S1000M03；	Absolute coordinate programming

G02I20.0F100；	Round processing, can not use the R
G03X-20.0Y20.0I-20.0；	R20 circular arc, available in
X-10.0Y10.0J-10.0	R10 circular arc, available in X-10.0Y10.0
M05；	
M30；	



graph 2-24

Example procedure 2:

Programming in both absolute and incremental values:

(1) Programming in an absolute value manner:

G92 X200.0 Y40.0 Z0；

G90 G03 X140.0 Y100.0 R60.0 F300；

G02 X120.0 Y60.0 R50.0；

perhaps

G92 X200.0 Y40.0 Z0；

G90 G03 X140.0 Y100.0 I-60.0 F300；

G02X120.0Y60.0I-50.0；

（2）Programming with incremental values:

G91G03 X-60.0 Y60.0 R60.0 F300；

G02 X-20.0 Y-40.0 R50.0；

perhaps

G91 G03 X-60.0 Y60.0 I-60.0 F300；

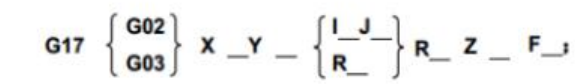
G02 X-20.0 Y-40.0 I-50.0；

4.4 screw insertion (G02, G03)

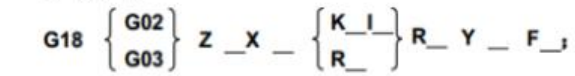
The tool moves from the current point at the feed speed specified by the parameter F.

instruction format:

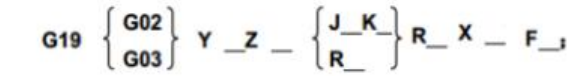
XY平面的圆弧



ZX平面的圆弧



YZ平面的圆弧



The first two digits of the instruction parameters are the positioning parameters.

The parameter word is the axis number (X, Y or Z) of the two axes in the current plane.

These two positioning parameters specify the location where the tool should be moved in the current plane.

The parameter word in the third place of the code parameter is the straight axis except for the circular arc interpolation axis.

Its parameter values are the height of the helix. The specific meaning and limitation of other code parameters are the same as for circular arc interpolation.

The feed speed can be specified in two ways:

1． Feed per minute (G94)

Before F, specify the tool feed per minute.

2． Each transfer (G95)

Before F, specify the tool feed per turn of the spindle.

instruction format:

G94 F_; feed per minute, F: feed speed instruction (mm / min or inch / min)

G95 F_; per feed, F: feed speed instruction (mm / rev or inch / rev)

explain:

1． Tangent speed is constant control

In machining, the system controls the cutting speed so that the tangent speed is maintained at the specified feed speed.

2． Feed per minute (G94)

After specifying G94 (feed per minute), the tool feed per minute is specified directly by the value immediately after F. G94 is a modal code, and once G94 is specified, it remains valid until G95 (per forward) is specified. When the power is on, the default setting is to feed per minute. Adjust the feed rate per minute by using the switch on the machine operating panel at 0% to 150% (10% interval).

3． Each transfer (G95)

After specifying G95 (per turn), the tool feed per turn of the spindle is specified directly specified by the value after F. G95 is a modal code, and once G95 is specified, it remains valid until G94 (feed per minute) is specified. Use the switch on the machine operating panel to adjust each feed multiplier from 0% to 150% (10% interval).

Warning: You cannot use multiplier adjustments for some instructions, such as tapping cycles.

4.6 Pause (G04)

Generates a time pause between the two program segments.

G04 instruction can make the tool for no feed short light finishing processing, generally used for boring plane, hole and other occasions.

instruction format:

G04X__;

G04P__;

P__; (in milliseconds)

X__; (in: seconds)

The pause instruction can delay the next program segment for a period of time.(Pause per second)

The delay time is specified by the instruction, and the instruction range is from 0.001 to 99999.999. In addition, in the cutting mode, in order to conduct an accurate stop check, the stop knife can be specified.

4.7 Polar coordinate instruction (G15, G16)

You can enter the radius and angle values to calculate the end point coordinate values.

From the positive direction of the first axis of the plane specifying the polar instruction, the angle along the counterclockwise direction is positive and the angle along the clockwise direction is negative.

In addition, the radius and angle can be specified under absolute / increment instructions (G90, G91).

instruction format:

G16: Polar coordinate instruction begins

G15: The polar coordinate instruction ends

explain:

1. Use G90 / G91:

When assigning polar instructions with G90, the origin of the coordinate system is the center of the polar coordinates.

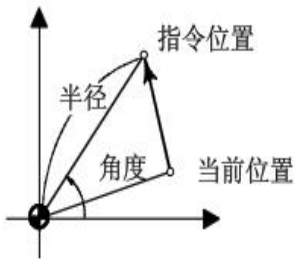
When using G91, the current position is the center of the polar coordinates.

2. When the origin of the work piece coordinate system is set as the center of the polar coordinate:

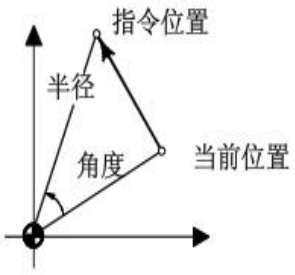
Specifies the radius value at absolute values.

The origin of the work piece coordinate system becomes the center of the polar coordinate.

However, when using the local coordinate system (G52), the origin of the local coordinate system becomes the center of the polar coordinate.



角度为绝对指令的情形

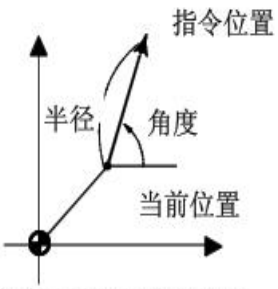


角度为增量指令的情形

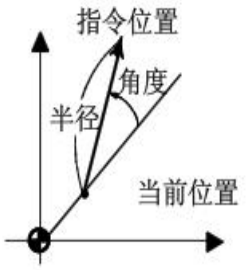
3. When setting the current position as the center of the polar coordinate:

Specifies the radius value in incremental values.

The current position is set as the center of the polar coordinates.



角度为绝对指令的情形



角度为增量指令的情形

place restrictions on:

Axinstructions with the following instructions are not considered as polar instructions.

●Pause of time (G04)

●Local coordinate system setting (G52)

●Change of work piece coordinate system (G92)

●Selection of the mechanical coordinate system (G53)

4.8 Programmable mirroring (G50.1, G51.1)

Specify the symmetry axis in the program, and the program can be mirrored following the symmetry axis.

instruction format:

G51.1 IP_; Programmable image settings

IP_: The axis of symmetry instructed

G50.1 IP_; Programmable image is cancelled

IP_: the axis of the instruction cancellation

explain:

When a mirror is applied to only one axis on the specified plane:

● Ararc instructions G02 and G03 are exchanged.

●The tool radius compensation G41 and G42 are exchanged.

place restrictions on:

In the programmable mirror mode, the G code (G28, G29, G30, etc.) related to the return reference point and the instructions for changing the coordinate system (G52~G59, G92, etc. can not be instructed). When instructing these G codes, please first cancel the programmable mirror mode. An alarm is given when the command is not cancelled.

5 In feed function F code

5.1 Fast movement

instruction format:

G00 X_ Y_ Z_;

G00: Quick point positioning (fast movement)

X_Y_Z_: Location of the destination

explain:

The fast moving speed of each axis is set by the parameter (see the user parameter —— speed F for details), so the fast moving speed does not require programming.

Use the switch on the operation panel of the machine tool to adjust the fast movement speed, and the multiplier value is: F0,25%, 50%, 100%.

F0: by the parameter NO.0503 Setting-up.

5.2 Cutting feed

The feed speed of linear (G01), circular arc (G02, G03) and other codes is specified by the value after the F code. F is the modal value, that is, the original programmed F value is always valid until a new F value is given. In cutting feed, the program segment performs continuously, so the change in feed speed can be minimized. The maximum value of F is controlled by the user parameter, via the parameter NO.0533-0535 Set the maximum allowable feed speed of each axis. The speed of the cutting feed can also be controlled by the feed rate switch on the operation panel. The actual cutting feed speed should be the product of the feed rate switch of F.

The feed speed can be set in two ways:

1． Feed per minute (G94): After F, specify the tool feed per minute.

2．Each turn feed (G95): After F, specify the tool feed per turn of the spindle.

5.3 Automatic acceleration and deceleration control

Automatic acceleration and deceleration control acts in the starting and stopping process of each axis movement to reduce the impact and make the starting and stopping process stable. In order to achieve the same purpose, automatic acceleration and deceleration control also acts in the process of feed speed transformation. For different feed methods, the NC system uses different acceleration and deceleration control modes:

Rapid positioning feed: using linear acceleration and deceleration control, the acceleration and deceleration time constant of each axis is controlled by the machine parameters.

Cutting feed: use exponential acceleration and deceleration control, the acceleration and deceleration time constant is controlled by the machine parameters.

Manual feeding: using exponential acceleration and deceleration control, the manual wheel acceleration and deceleration time constant of each axis is also determined by the user parameter NO.1423 to 1437 controls.

6 Reference point

6.1, and the reference point

The machining center automatic tool changer often uses the Z axis back to the reference point of the machine tool. The machine reference point is a fixed point on the machine and is independent to the machining procedure. Generally, the machine reference point is specified by the manufacturer of the X axis forward, Y axis forward and Z axis forward as the reference point. The machine tool origin is also called the machine tool zero point, which is determined indirectly by the machine tool reference point. After the machine tool is started, the position of the X axis, Y axis and Z axis of the machine tool should be "back to zero", that is, return to the reference point, so that the numerical control device can confirm the position of the original point of the machine tool through the reference point, so as to establish a machine tool coordinate system inside the numerical control system with the zero point of the machine tool as the origin of the coordinates.

6.2 Return to the Reference Point (G28)

The reference point is a fixed position on the machine tool, moving the tool to the return reference point function.

For example, the reference point may be used as a position for the automatic knife changer.

33 By in the parameter (No.970-987) specifies up to four reference points of the machine coordinate system.

Each axis is quickly moved to the middle point to return to the reference point. Therefore, for safety, the tool radius compensation and the tool length compensation should be cleared before executing this instruction.

The coordinate values of the middle point are stored in the CNC, and only the coordinate values of the instruction axis in the G28 segment are stored each time. For other axes, use the coordinate values used for previous instructions.

instruction format:

G28 X_Y_Z_; (return to Point 1 of Reference)

G30 P2 X_Y_Z_; (Return to Reference Point 2)

G30 P3 X_Y_Z_; (Return to Reference Point 3)

G30 P4 X_Y_Z_; (Return to Reference Point 4)

G28 X0 Y0 Z0; It means that the tool first passes through the origin of the working coordinate system and then returns to the reference point, which is easy to appear the tool and clamp workpiece interference.

G91 G28 X0 Y0 Z0; The tool returns the reference point from the current point.

Commonly used format:

G91 G28 Z0 ；

G28 X0 Y0 ；

X _ Y_ Z_ indicates the middle point, whose coordinate value can be used in absolute value or incremental value. As an incremental value, it is the incremental value of the middle point relative to the current point of the tool. The middle point is set to prevent interference with the workpiece or fixture when the tool returns to the reference point. When using this instruction, pay attention to the following issues:

1． Usually, the G28 instruction is used when changing the knife, and the tool length compensation and radius compensation should be cancelled before executing the instruction.

2．In G28, not only the moving instruction value, but also the middle point coordinate value. That is, for the axes that are not instructed in the program segment using G28, the previous coordinate value in G28 is used as the middle point coordinate value of that axis.

for instance:

N01 G90 G00 X100 Y100 Z100；

N02 G28 X200 Y300; (Middle point is 200,300)

N03 G28 Z150; (Intermediate point is 200,300,150)

6.3 Return to the speed setting of the reference point

After power on, before the first reference point, the speed of manual and automatic return to the reference point is the same as the automatic fast movement, and the low speed is determined by the user parameter NO.1207-1212 Settings.

Even after returning to the reference point is completed or the machine coordinate system is established, the manual return to the reference point is the same speed as the set value of the parameter.

pay attention to:

1． The back-reference point speed uses a fast moving rate (F0,25%, 50%, 100%), and the set value is the speed value at a rate of 100%.

2． The reference point return is complete, after the machine coordinate system has been established, the automatic return reference point feed speed will be consistent with the usual fast movement speed.

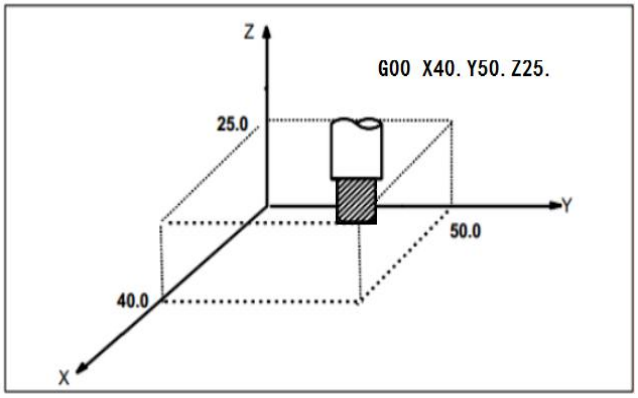
3． Before the reference point is not returned and the machine tool coordinate system is established, the available parameter NO.1072-1077 Set 100% -rate manual fast movement speed through the parameter NO.1081-1086 Set the F0 rate manual fast shift speed.

4．The operation of executing G28 instruction back to the origin is generally not carried out in the absolute mode of G90. If the "G90 G28 X 0 Y 0 Z0;" statement is in danger of serious accident, it is generally required that the operation of the machine back to the origin in G91 mode, namely G91 G28 X0 Y0 Z0; execute the action with the current point as the middle point.

7 The coordinate system

Specify the CNC a position to reach the tool and the tool can move to that position. The position of such a tool is given by the coordinate value in a certain coordinate system.

In the case of 3 program axes (e. g., X, Y, Z axis), the coordinate value is specified below.



graph 2-25

You can specify the coordinate values with one of the following three coordinate systems:

(1) Mechanical coordinate system

(2) Workpiece coordinate system

(3) Local coordinate system

7.1 Machine tool coordinate system

A specific point on the machine tool as the processing reference is called the machine tool zero, which is set by the machine tool manufacturer for each machine tool.

The coordinate system set by using the machine tool zero point as the origin is called the machine tool coordinate system.

After the system is powered on, the machine tool coordinate system is established by performing a manual return to the reference point. Once the machine coordinate system is set, it remains unchanged until the power is turned off.

instruction format:

G90（G53）X_Y_Z_;

X_Y_Z_ Actual coordinate point

Select the Machine Tool Coordinate System (G53)

When the position on the machine frame is specified, the tool moves to that position. The G53 used to select the machine coordinate system is a non-modal G code, meaning that

it is only valid in the program segment of the instruction machine coordinate system. Absolute value (G90) shall be specified for G53. When the incremental value instruction is specified (G91), the G53 instruction is ignored. When the command tool is moved to a special position of the machine tool, such as the tool switch position, the program should be moved in the machine tool coordinate system based on the G53 writing program.

explain:

1． When the G53 instruction is specified, the tool radius compensation, tool length compensation, and tool bias are cleared.

2．Immediately after the power supply is on, the G53 must be set before the G53 instruction, so the power on must manually return to the reference point or automatically return to the reference point by the G28 instruction. When an absolute position encoder is used, the operation is not required.

7.2 Workpiece coordinate system

The coordinate system used in the process of a workpiece is called the artifact coordinate system.

The artifact coordinate system is preset by CNC (set the artifact coordinate system).

Write the program and process the artifact in the set artifact coordinate system (select an artifact coordinate system).

You can change the set artifact coordinate system (change the artifact coordinate system) by moving the origin.

7.2.1 Set up the workpiece coordinate system

You can set the artifact coordinate system in two ways:

1． Using G92

Specify a value after G92 in the program to set the artifact coordinate system.

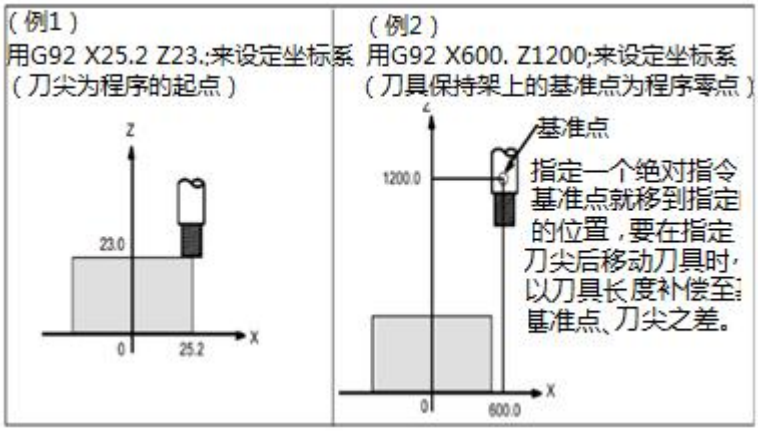
instruction format:

（G90）G92 X_Y_Z_；

explain:

Set the artifact coordinate system so that the points on the tool (for example, the tip) are at the specified coordinate position. If the coordinate system is set with G92 during the tool length offset, G92 sets the coordinate system with unbiased coordinate values. The tool radius compensation was temporarily removed by G92.

give an example:



graph 2-26

2． Enter in the [Coordinate System] interface:

Through the [cutting bias / setting] [coordinate system] interface, 6 artifact coordinate systems can be set, and which artifact coordinate system to use is selected through the code G54~G59.

When using the absolute value instruction, the artifact coordinate system must be established with the above method.

7.2.2 Select the artifact coordinate system

The user can arbitrarily choose the set artifact coordinate system, as described below.

1. After the workpiece coordinate system is set by G92 or the automatic setting of the workpiece coordinate system method, the workpiece coordinate system works with absolute instructions.

2. Through the interface of [knife bias / setting] [Coordinate system], 6 artifact coordinate systems can be set G54~G59. Specify one of the G codes to select one of the 6.

G54 artifact coordinate system 1

G55 artifact coordinate system 2

G56 artifact coordinate system 3

G57 artifact coordinate system 4

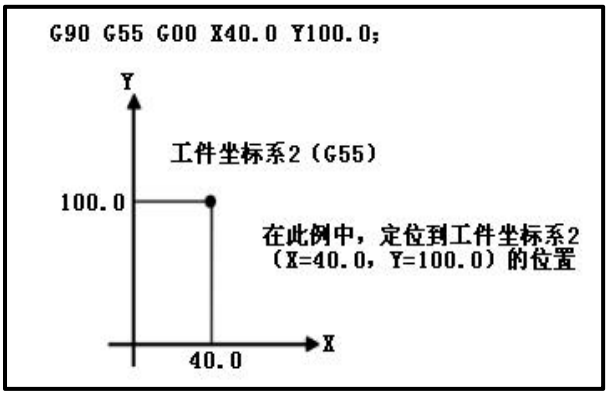
G58 artifact coordinate system 5

G59 artifact coordinate system 6

After the power comes on and returns to the reference point, establish the artifact coordinate systems 1 to 6.

When the power supply is turned on, the G54 coordinate system is automatically selected.

give an example:



graph 2-27

7.2.3 Change the workpiece coordinate system

You can change the position of the six artifact coordinate systems specified from G 54 to G 59.

There are two ways to change the external artifact zero offset value or the artifact zero offset value:

1． Modify through the [bias / setting] [Coordinate System] interface.

2． The coordinate system was set with G92.

instruction format:

G92 X_ Y_ Z_;

7.3 Local coordinate system

When programmed in the artifact coordinate system, the subcoordinate system of the artifact coordinate system can be set to facilitate programming. The subcoordinate system is called the local coordinate system.

instruction format:

G52 X_ Y_ Z_; sets the local coordinate system

G52 X0 Y0 Z0; Cancel the local coordinate system

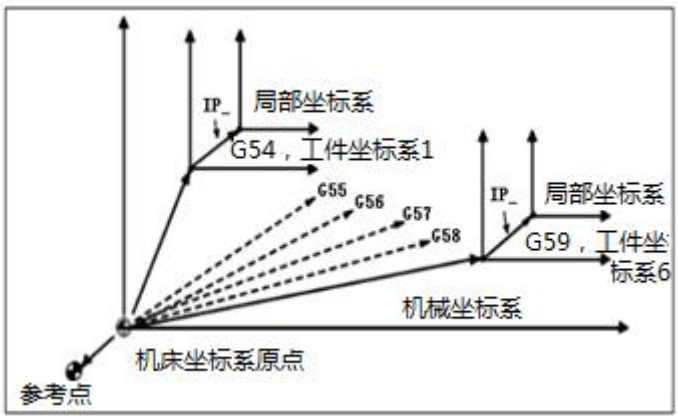
explain:

With the instruction G52 X_ Y_ Z_; specify that the local coordinate system can be set in the artifact coordinate system (G54~G59). The origin of the local coordinates is set to the position specified as X_ Y_ Z_ in the artifact coordinate system.

When the local coordinate system is set, the following move in absolute value mode (G90) instruction is the coordinate value in the local coordinate system. Specifying a new

zero point of the local coordinate system in G52 in the artifact coordinate system can change the local coordinate system.

In order to cancel the local coordinate system and specify the coordinate value in the artifact coordinate system, the zero point of the local coordinate system should be consistent with the zero point of the artifact coordinate system.



graph 2-28

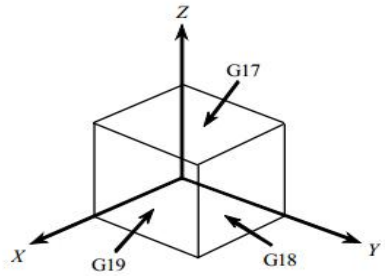
7.4 Plane selection

For arc insertion, tool radius compensation and drilling, using G code.

G17: the XY plane

G18: ZX plane

G19: YZ plane



graph 2-29

explain:

The plane remains unchanged in no specified G17, G18, or G19 program segments.

When the power is on or reset, pass the parameter (No.3802) Select one of them: G17, G18, and G19.

8 Spindle speed function (S function)

8.1 Specification of the spindle speed

When a value is specified after address S, the code signal and the selected communication number are sent to the machine to control the spindle rotation speed. A program segment can contain only one S code. The spindle speed can be directly specified by a maximum value of 5 bits (rpm) after the address S. The unit of the spindle speed instruction depends on the regulations of the machine tool manufacturer. When the move instruction and the spindle S function instruction are in the same program segment, the move instruction and the S function instruction are executed simultaneously, and the default spindle speed can be set by the parameter 0513.

The unit is "r/min", such as S800 means that the machine spindle speed is 800r / min.

8.2 Constant surface cutting speed constant control (G96, G97)

After S specifies the surface speed (relative speed between the tool and the workpiece). Spindle rotation maintains a constant surface cutting speed regardless of the tool position.

instruction format:

●**Constant surface cutting speed constant control:**

G96 S○○○○;

Surface speed (m / min or met et / min)

●**Constant surface cutting speed constant control cancellation:**

G97 S○○○○;

Spindle velocity (rpm)

●**Maximum spindle speed clamp:**

G92 S__ ;

Through the parameter NO.0014 Specify the maximum spindle speed (rpm) after setting S

9 Auxiliary function M code

When the value is specified after the address M, the code signal and the selected communication signal are sent to the machine tool. The machine tool uses these signals to interconnect or disconnect its various functions.

Generally, only one M code can be specified in a program segment. In some cases, up to three M codes can also be specified for some machine tools. Which code corresponds to which machine tool function, which is determined by the machine tool manufacturer.

Except for M98, M99, M code for calling a subprogram, and M code for calling a user macro program, all M codes are processed in the machine tool.

Common M code:

Tables 2-10

M code	functional description
M00	program halt
M01	The condition procedure is
M02	end of program
M03	The spindle is turning
M04	Spindle reversal
M05	The spindle stop
M06	Tool exchange
M07	Cool off with a blow
M08	Cool open
M09	Cooling off
M19	Spindle orientation
M29	Spindle position mode (snap attack)
M30	The program ends and returns to the
M98	call subroutine
M99	Subroutine end return / repeat

In machine tools, M code is divided into two categories: one is used directly out by NC to control the execution of the program; the other is executed by PLC, which controls the spindle, automatic knife changing device and cooling system.

1．M code

The M codes used for program control are M00, M01, M02, M30, M98, M99. Their functions are explained as follows:

M00- - - - - - - - - -Procedure is suspended. When the CNC system performs to M00, the execution of the program is interrupted. When the program stops, all the existing modal information is stored, and press the cycle start button to continue the program. When the CNC, executed to the M00 instruction, the current program will be suspended to facilitate the operator of the tool and workpiece size measurement, workpiece around, manual speed change and other operations.

When pause, the machine is stopped, while all the existing modal information remains unchanged. In order to continue the subsequent procedures, press the "cycle start" key on the operation panel.

M00 is for the nonmodal action M function.

M01- - - - - - - - -The condition procedure is suspended. Similar to M00, when the CNC system performs M01, if the stop switch is in the open state, the M01 and the M00 instruction have the same effect; if the stop switch is in the closed state, the M01 instruction does not have any effect.

If the user presses the Select Stop button on the bright action panel. When the CNC, executed to the M01 instruction, the current program will be suspended to facilitate the operator of the tool and workpiece size measurement, workpiece turn, manual speed change and other operations. In the time of suspension, the feed of the machine tool stops, while all the existing modal information remains unchanged. In order to continue the subsequent procedures, press the "cycle start" key on the operation panel.

If the user does not activate the Select Stop button on the Action panel. When CNC, execute to the M01 instruction, the program will not pause and continue to execute down.

M01 is for the nonmodal action M function.

M02- - -End of the procedure. When encountering the M02 instruction, the numerical control system thinks that the program has ended, stops the operation of the program and sends a reset signal.

M02 is programmed in the last program segment of the main program.

When the CNC performs the M02 instruction, the spindle, feed and coolant of the machine tool all stop and the processing ends.

After the program of M02, you have to redo the program, and then press the Circular Start key on the action panel.

M02 is for the nonmodal action M function.

M30- - -The program ends and returns to the program header. In the program, M30 returns the program to the program header in addition to playing the same role as M02.

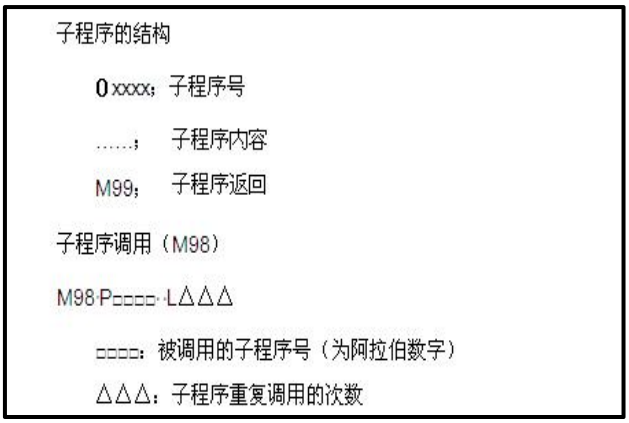
After the program using M30, to restart the program, simply press the Circular Start key on the action panel again.

M98- - - -Call the subprogram. No code signal and selected pulse signal are sent.

M99- - -Subprogram ends, and returns to the main program. No code signal and selected pulse signal are sent. If M30 is not specified in this system, the subroutine will be repeated.

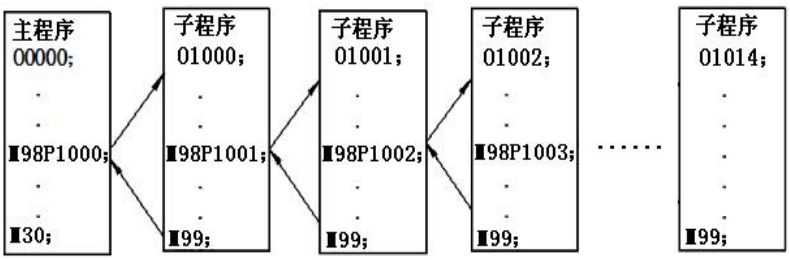
Subroutine call function

If a program contains a fixed sequence or a frequently repeated pattern, such a sequence or pattern can be stored in the memory as a subroutine to simplify the program. A subprogram can be called from the main program. In addition, a subprogram that is being called can also call another subprogram again.



Subprogram nested call

When the main program calls a subprogram, it is called as a level 1 one. Subprogram calls can be nested up to 15 levels as follows:



If M99 is executed in the main program, the control returns to the beginning of the main program and the main program is executed from scratch.

2． Auxiliary function M code

M03- - -spindle forward rotation. Use this instruction to rotate the spindle clockwise at the currently specified spindle speed (negatively from the Z axis to the Z axis). When executing M03 code in the program, add "S" code to specify the spindle speed.

M04- - -spindle reversal. Use this command to rotate the spindle counterclockwise at the currently specified spindle speed (negatively from the Z axis to the Z axis). When executing M04 code in the program, add "S" code to specify the spindle speed.

M05- - -spindle stopped. The M05 code spindle stop is specified in the program.

M06- -Tool exchange. For M06, the machine tool shall have the ATC switching function.

M07- - -blow the air and cool it on. For M07, the machine shall have the blowing and cooling function.

M08- -cool on. To perform M08, the machine tool shall have the cooling function.

M09- -cool and blow off. With M09, the M07 (blow to cool on) and M08 (cool on) functions will be closed simultaneously.

M19- - -spindle orientation. M19 instruction spindle orientation.

M 29- -spindle position mode (rigid tapping). Performing M 29 instruction spindle into position mode for rigid tapping operation. The machine tool shall have the position mode control function.

Machine tool manufacturers often set the self-developed machine tool function to M code (such as machine tool open / closing), these M codes please refer to the operating manual of the machine tool.

10 Tool compensation function

10.1 Tool Length Compensation (G43, G44, G49)

When the machining center is running, it is necessary to exchange tools frequently. After replacing tools of different lengths, if the working coordinate system remains unchanged, serious accidents will occur by overcutting or crash. Therefore, the CNC system introduces the algorithm of tool length compensation, which produces the instruction of tool length compensation.

When setting the zero point of the Z axis workpiece, let the reference surface of the main shaft cone hole theoretically coincide with the zero point of the Z axis workpiece set. When using each tool, the machine can raise a distance according to the zero length of the workpiece, so that the tip of each tool is just on the surface of the workpiece, and this height is the tool length compensation value. As shown in Figure Figures 2-30, H1, H2 are the tool length compensation value. This compensation value is in, the system function knife bias / setting —— tool repair interface setting.

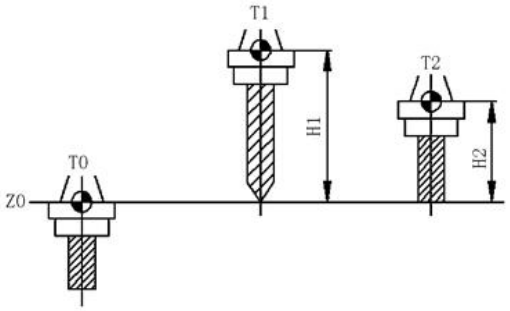


Figure 2-30 Length compensation principle

instruction format:

G43H_; tool length positive compensation;

G44H_; negative compensation for tool length;

G49 or H00; the tool length compensation is cancelled;

The G43 instruction is a positive compensation for the tool length, that is, the actual position reached by the Z axis is the addition of the instruction value and the compensation value; the G44 instruction is a negative compensation for the tool length, that is, the actual position reached by the Z axis is the position of the instruction value minus the compensation value.

G43 or G44 is the modal instruction, and the compensation number specified by H_ is also the use of the modal instruction. The programmer can consider the length of the tool without considering the position of the tip of the tool. Tool wear or damage after replacing a

new tool does not need to change the processing procedure, directly modify the tool compensation value.

Values of H range from 000 to 099. H00 means to cancel the tool length compensation value. Another way to cancel tool length compensation is to use the instruction G49. When NC performs the G49 instruction or H00, immediately cancel the tool length compensation and move the Z axis to the command position without compensation value.

Whether the absolute instruction or the incremental instruction, when G43 is specified, the tool length compensation amount specified by the H code (set in the offset memory) is added to the coordinate value of the end position specified by the programmed move instruction; and when G44 is specified, the same value is subtracted from the coordinate value of the end position, and the resulting coordinate value becomes the end position. When eliminating the axis instruction, you only move the value of the tool length offset.

G43 and G44 are modal G codes valid before another G code within the same group.

In the instruction offset number, only the new tool length compensation changes, not by adding the new tool length compensation to the original tool length compensation.

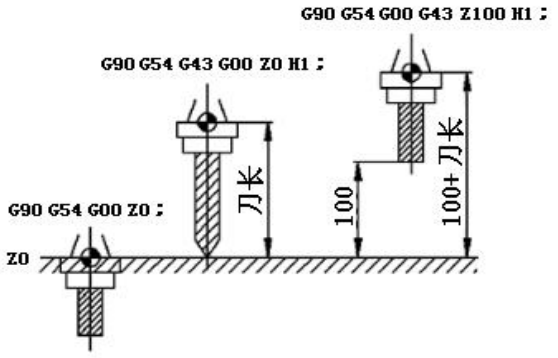
for instance:

H1 ： tool length compensation is 20.0; H2 ： tool length compensation is 30.0.

G90 G43 Z100.0 H1; Z axis will move to 120.0

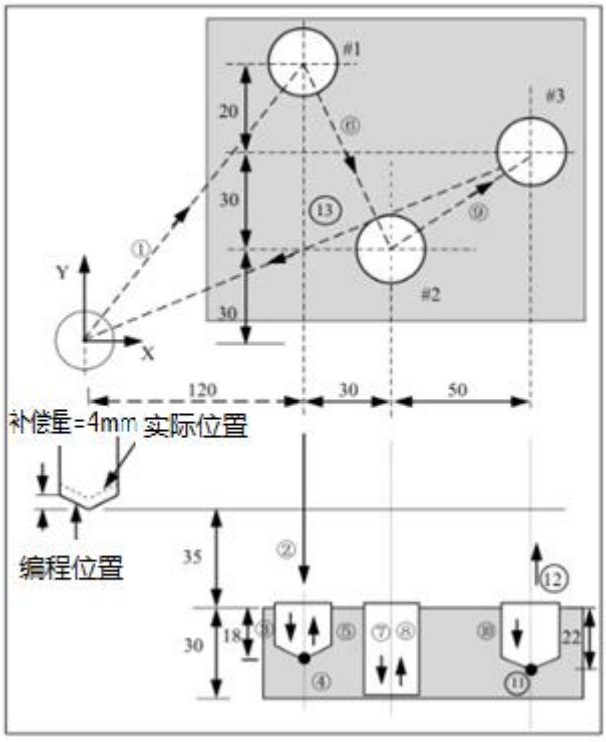
G90 G43 Z100.0 H2; Z axis will move to 130.0

Since the value of compensation value is-999999.999~999999.999mm or-99999.9999 to 99999.9999 inches. Generally, G49 or H00 is relatively safe to use after the Z axis goes back to the origin. Through the change of positive and negative compensation values, the G43 instruction can be used to complete the whole work, so in practice, the G43 instruction is used in most cases. The schematic diagram of G43 instruction is shown in Figure 2-31:



graph 2-31

Example of the drilling processing:



graph 2-32

order:

H1= -4.0 (tool length compensation amount)

G91 G00 X120.0 Y80.0;

G43 Z-32.0 H1 ;

G01 Z-21.0 F1000 ;

G04 P2000 ;

G00 Z21.0 ;

X30.0 Y-50.0 ;

G01 Z-41.0 ;

G00 Z41.0 ;

X50.0 Y30.0 ;

G01 Z-25.0 ;

G04 P2000 ;

G00 Z57.0 H0 ;

X-200.0 Y-60.0 ；

M30 ；

10.2 Tool Radius Compensation (G41, G42, G40)

The function of the tool radius compensation function is to change the programming trajectory with the tool center as the program to the workpiece outline as the programming trajectory, that is, the CNC system is required to automatically calculate the tool center trajectory according to the workpiece outline and the tool radius value in the program. The principle is shown in Figure 2-32:

instruction format:

G41D__; tool radius left compensation

G42 D__; right compensation of tool radius

I P__; the instruction value for the axis movement

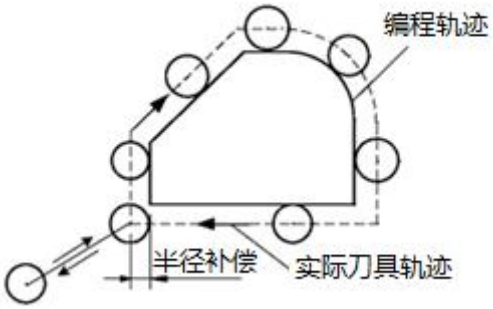
G40__; the tool radius compensation is cancelled

explain:

1． The tool radius compensation is by by G17, G18 and G19 in the selected working plane. For example, when the G17 command is executed, the tool radius compensation only affects the movement of the X and Y axes, and has no effect on the Z axis.

2． D_ is the tool compensation number, its specific value has been set in the compensation memory before machining or commissioning, and D_ is the continuation code.

3．The tool radius compensation must be cancelled before the end of the program, otherwise the tool center will not return to the program origin; the tool radius compensation must be cancelled in G00 and G01 mode, and the machine will alarm in G02 and G03 mode. The tool radius compensation can also be cancelled with D00.



graph 2-33

Positive and negative tool radius compensation amount and tool center path:

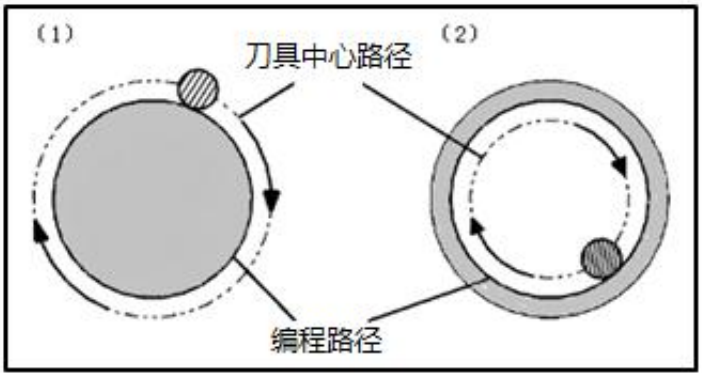
If the tool radius compensation is set to a negative value, it becomes the tool equivalent to the G41 and G42 of the program. Therefore, bypassing the tool on the outside of the workpiece will bypass the inside of the workpiece and vice versa. Figures Figure 2 – 33 represent an example.

In general, the tool radius compensation should be programmed to a positive value (+).

When the tool center path shown in ① is specified in the program, if the tool radius compensation amount is set to negative, the tool center is moved as shown in ②.

If a program as shown in ② begins and the tool radius compensation is set to negative (-), the tool center is moved as shown in ①.

Therefore, the cutting of the Yang and Yin shapes is allowed in the same program, and the spacing between the two can be arbitrarily adjusted by appropriately selecting the tool radius compensation amount.



graph 2-34

Set range of the compensation amount (range of the value set by the compensation amount): ± 999999.999.

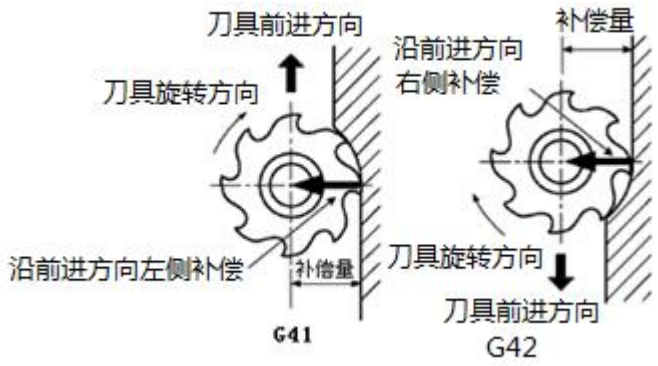
Left and right tool radius compensation judgment:

G41: tool left compensation refers to the direction along the tool, the tool is biased to the left of the workpiece outline.

G42: tool right compensation refers to the direction along the tool, the tool is biased to the right of the workpiece outline.

G40: Cancel the tool radius compensation.

As shown in Figure 2-35:



graph 2-35

Program examples:

O0001；

G90 G54 G17 G00 X0 Y0 S1000 M03；

G41 X20.0 Y10.0 D01;, the compensation begins

G01 Y50.0 F100；

X50.0；

Y20.0；

X10.0；

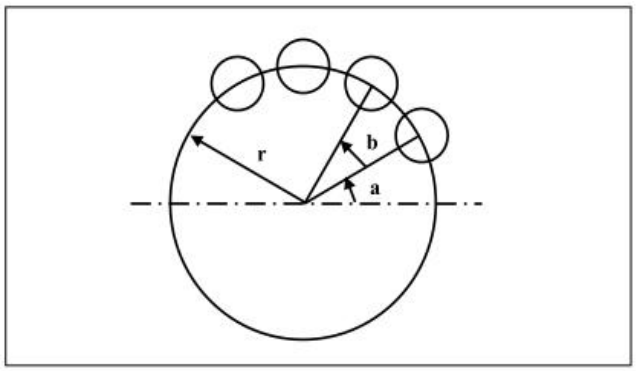
G40 G00 X0 Y0; Compensation cancellation

M05；

M30；

11 Macro instructions

The method of CNC programming with variable mode is called macroprogram programming. The user macro code can realize variable assignment, arithmetic operation, logical judgment and condition transfer, which is conducive to compiling the processing program of special parts, reduce the tedious numerical calculation during manual programming, and simplify the user program. Process the bolt hole circle as shown in Figure 2-36 below.



graph 2-36

In the above figure, a bolt hole circle is programmed and stored in the system. After that, the user can call this program to process the bolt hole circle at any time. Just fill the bolt hole properties such as the number of holes and the deviation Angle, just like the user adds the bolt hole circle function to the system.

11.1 The macro variable

11.1.1 variable representation

In the user macro program, the symbol "#" and the variable name are used to represent the variable, such as G00X # 201, which represents the value of the variable 201 as the programming instruction coordinates of the X axis. If you want to change the symbol of the reference variable, put the negative sign ahead of "#".

For example, the G00X- - # 1.

instruction format:

＃I （I=1，2，3，…) Or # [<formula sub>]

Examples: # 5, # 109, # 501, # [# 1 + # 2-12]

The expression can be used to specify the variable number. Therefore, the expression must be given in parentheses.

Note: The user macro program does not allow the direct use of the variable names. The variable is specified with the variable symbol (#) and the subsequent variable number.

11.1.2 Type of variables

Table 2-11 Type of the macro variable

Variable number	type of variable	function
#0	dummy variable	This variable is always empty, and no value can be assigned to this variable.
#1-#33	local variable	Local variables can only be used to store data in macro programs, for example, operation results. When the power cuts off, the local variables are initialized as empty. When calling the macro program, the independent variable assigns a value to the local variable.
#100-#199 #500-#999	Public variables	The common variables have the same meaning in the different macro programs. When the power is lost, the variable # 100- # 199 is initialized as empty, and the value of the variable # 500- # 999 is saved, even if the power loss is not lost.
#1000	system variables	System variables are often used to read and write CNC data, such as the tool current position and compensation values.

11.1.3 Use of the variables

1. Specify the variable number or formula after the address word

instruction format:

<Address Word> # I

<Address Word> - # I

<Address word> [<child>]

Example: F # 103, set # 103=15 is F15

Z- - # 110, let # 110=250 be Z-250

X 【＃24＋＃18×COS 【＃1】】

2. Variable number can be replaced by variables

Example: # [# 30], let # 30=3 is # 3

3, # 0 are empty variables, and the variables that do not define the variable values are also empty variables

4. Variable value definition

The decimal point can be omitted during the program definition, example: # 123=149

11.2 Call of the macro program

11.2.1 Nonmodal call (G65)

The macro program can be called with the G65 instruction, which is not a modal instruction.

The macro program call is also a subprogram call, which is different from M98 in that:

1． When called with G65, the value of the specified independent variable is passed to the macro program that is called. The M98 does not have that feature.

2．Change the nesting levels of local variables when calling with G65, but M98 does not.

instruction format:

G65 P_ _ L <independent variable> _;

The macro program specified by the address P is called, and the independent variable (data) is passed to the user macro program body.

Macro program number by which P —— was called

The number of times L —— is called (default to 1 to specify the number of repetitions from 1 to 9999)

<Independent variable> —— is transferred to the data in the macro program whose value is assigned to the corresponding local variable.

Programming examples:

Example procedure 1:

O0032；

G54 G0 X100 Z100；

G65 P100L5 X50 Z-30 F1000 U2; without the program number of 100, look up in the user program area

G00 X50 Z10；

M30；

Example procedure 2:

O100；

G01 X[#23] Z[#25] F[#5]；

G01 U[#20]；

G81 X[#23] Z[#25]；

U[#20]；

U[#20]；

G0 X100 Z50；

M30；

11.2.2 Modal call (G66)

The macro program can be called with the G66 instruction, which is a modal instruction.

The macro program call is also a subprogram call, which is different from M98 in that:

1．G66, and the value of the specified independent variable is passed to the macro program that is called. The M98 does not have that feature.

2．Change the nesting levels of local variables when calling with G66, but M98 does not.

instruction format:

G66 P_ _ L <independent variable> _;

Macro program number by which P —— was called

The number of times L —— is called (default to 1 to specify the number of repetitions from 1 to 9999)

<Independent variable> —— is transferred to the data in the macro program whose value is assigned to the corresponding local variable.

Nested calls: G66 calls can have five levels of nesting.

11.2.3 Cancel the modal call (G67)

instruction format:

G67; Cancel the modal macro program call;

Programming examples:

O2005；

G00 X100 Z50；

G66 P020100L2 A2 B20 C20 I30 J20 K20; The P020100 number program is called twice when executing this segment

G01 X80 Z50; After executing this paragraph, call P0100 program number twice again (and update the local variable by the independent variable)

G67; G66 modal call is cancelled

G01 X20 Z50; After the of is, no no after after is this called

11.2.4 Arithmetic and logical operations

The operations that can be performed in the variables are shown in Table 2-12. The expression on the right side of the operator can contain either constants or functions. Variables in the expression are replaced with # i, # j and # k.

Table 2-12 Arithmetic and logical operations

function	form	remarks
assignment	#i = #j	
addition	#i=#j + #k	
subtraction	#i=#j - #k	
multiplication	#i=#j * #k	
division	#i = #j / #k	
sine	#i = SIN 【#i 】	The angle is specified in degrees.
arcsine	#i = ASIN【#j 】	(1) The Angle is specified in degree. (2) Range-90 degrees to 90 degrees
cosine	#i = COS 【#i 】	The angle is specified in degrees.
anticosine	#i = ACOS 【#i 】	(1) The Angle is specified in degree.(2) Range from 0 degrees to 180 degrees
tangent	#i = TAN 【#j 】	The angle is specified in degrees.
Conversion from BIN to BCD	#i = BCD 【#j 】	

pay attention to:

1． In the functions ASIN (# j) and ACOS (# j), the value range of # j should be-1 # j 1, otherwise the program alarms;

2．The angular parameters of all trigonomefunctions are in degrees.

11.3, the control instruction

11.3.1 Unconditional transfer of GOTO statements

instruction format:

GOTO n ；

N: Serial number (1~99999)

Transfer to a statement with the serial number n.

11.3.2 Condition Transfer IF statement

Instruction format 1:

IF [<conditional expression>] GOTON

Specify a conditional expression after the IF. If the specified conditional expression is satisfied, transfer to the statement with serial number n; if the conditional expression is not satisfied, the program performs the next block.

Programming for example

IF 【#1 GT 10】GOTO 2；

…

N2 G00 G91 X10.0；

Instruction format 2:

IF [<conditional expression>] THEN

If the specified conditional expression is satisfied, the predetermined macro statement is executed. But execute only one macro statement.

IF 【#1 EQ #2】THEN #3=0；

Table 2-13 Macro program operators

operator	meaning
EQ	equal to (=)
NE	not equal to (=)
GT	greater than (>)
GE	greater than or equal to
LT	less-than (<)
LE	LE (≦)

11.3.3 Loop (WHILE statement)

instruction format:

WHILE [Conditional Expression] DOm; (m=1,2,3)

…

END m;

Specify a conditional expression after the WHILE.

If the program execufrom DO to the END if the specified conditional expression is satisfied; if the specified conditional expression is not satisfied, the program executes the program block after the END.

1． The label can be reused multiple times.

2． The DO range cannot be crossed.

3．The DO cycles can be nested at level 3.

11.3.4 User macro program

Although the subroutines are very useful for compiling the same processing operation, the user macro program makes it easier and easier to compile the same processing operation by allowing the use of variables, arithmetic and logical operations and condition transfer. The same processing operations can be compiled into general programs, such as chamber machining macro and fixed machining loop macro. When using, the processing program can use a simple instruction to call up the user macro program, and call exactly the same subprogram.

11.4 variables

Ordinary processing procedures directly specify the G code and the movement distance with numerical values.

The representation of the variables:

The computer allows variable names, but the user macro program does not. The variable is specified with the variable symbol (#) and the subsequent variable number. For example: # 1

The expression can be used to specify the variable number. At this point, the expression must be enclosed in parenthesis.

E. g.: # [# 1 + # 2-12]

type of variables:

When the variable values are defined in the program, the decimal points can be omitted.

Example: When definition # 1=123; the actual value of variable # 1 is 123.000.

Reference of the variable:

To use the variable value in the program, specify the address following the variable number. When assigning a variable with an expression, place the expression in parentheses.

For example: G01X [# 1 + # 2] F # 3;

The values of the cited variable are automatically rounded according to the minimum set unit of the address.

for instance:

When G00X # 1; is executed in 1 / 1000mm, CNC assigns 12.3456 to variable # 1 and the actual instruction value is G00X12.346.

Change the sign of the referenced variable value and place the minus sign (-) in front of #.

For example: G00X- # 1;

When referring to undefined variables, both the variable and the address word are ignored.

For example, when the value of the variable # 1 is 0 and the variable # 2 is empty, the execution result of G00X # 1Y # 2 is G00X0.

Such a variable becomes an "empty" variable when the variable value is not defined. Variable # 0 is always an empty variable. It cannot be written, but can only be read.

(a) quote

When referring to an undefined variable, the address itself is also ignored.

(b) operation

Except for assigning a value with <null>, in the other cases <null> is the same as 0.

(C) The conditional expression

The <empty> in EQ and NE differs from 0.

Limit: the program number, sequence number and optional program segment skipping number cannot use the variable.

Variables are not used in the following ways.for instance:

O#1

/#2 G00 X100.0

N#3 Z200.0

11.4.1 System Variables

System variables are used to read and write the NC internal data. For example: tool offset value and current position data. However, some system variables can only be read through. System variables are the basis for automatic control and universal machining programs development.

11.4.2 Arithmetic and logical operations

The operation listed below can be performed in the variables. The expression on the right of the operator may contain constants and / or variables consisting of a function or operator. Varivariables # j and # k in an expression can be assigned constant. The variables on the left can also be assigned with an expression.

function	form	remarks
assignment	#i = #j	
addition	#i=#j + #k	
subtraction	#i=#j - #k	
multiplication	#i=#j * #k	
division	#i = #j / #k	
sine	#i = SIN 【#i 】	The angle is specified in degrees.
arcsine	#i = ASIN【#j 】	(1) The Angle is specified in degree. (2) Range-90 degrees to 90 degrees
cosine	#i = COS 【#i 】	The angle is specified in degrees.
anticosine	#i = ACOS 【#i 】	(1) The Angle is specified in degree.(2) Range from 0 degrees to 180 degrees
tangent	#i = TAN 【#j 】	The angle is specified in degrees.
Conversion from BIN to BCD	#i = BCD 【#j 】	

When CNC handles numerical operation, if the absolute value of the integer is greater than the absolute value of the original number, if the absolute value of the original number is lower. Care should be taken of negative numbers.

11.4.3, macro program statements and NC statements

The following program segment is the macro program statement:

1. A program segment that contains arithmetic or logical operations (=).

2. A program segment that contains control statements (for example, GOTO, DO, END).

3. A program segment containing the macro program calling instructions (e. g., calling a macro program with G65, G66, G67, or other G code M code).

4. Any program segments other than the macro program statement is a NC statement.

11.4.4 Transfer and circulation

In the program, using the GOTO statements and IF statements can change the flow of control. Three transfer and cycle operations are available.

Transfer and cycle: the GOTO statement (unconditional transfer)

IF statement (conditional transfer IF THEN)

WHILE Statement (then loop)

1. Unconditional Transfer (GOTO statement)

Transfer to a program segment marked with the serial number n. A P / S alarm occurs when assigning a sequence number beyond 1 to 99999. An expression can specify a serial number.

instruction format:

GOTOn

N: Order Number (1-99999)

for instance:

GOTO1

GOTO#1O

2. Conditional transfer (IF statement)

F After that, it specifies a conditional expression.

IF [<Conditional Expression>] GOTO n

If the specified conditional expression is satisfied, transfer to a program segment marked with the sequence number n. If the specified conditional expression is not satisfied, perform the next segment.

for instance:

If the value of the variable # 1 is greater than 10, transfer to the program segment of the sequence number N2.

IF [# 1GT10] GOTO 2 N2 G00 G91 X10.0 if the conditions are met

If the conditions are not met, continue the downlink procedure.

IF [<conditional expression>] THEN

If the conditional expression is satisfied, execute a predetermined macroprogram statement. Only one macroprogram statement is executed.

for instance:

If # 1 and # 2 have the same value, 0 is assigned to # 3.

IF #1EQ#2 THEN#3=0；

12 The fixed cycle function

Fixed cycles make the programmer programming easy. With the fixed loop, the fixed mode processing operation can be instructed in one program segment with G code; without the fixed loop, multiple program segments must be compiled. In addition, fixed cycles can save memory space.

Table 2-14 Fixed cycle

G code	Drill cut (-Z-direction)	The hole bottom action	Backback (+ Z direction)	explain
G73	Cut into each other	not have	quick travel	High speed discharge drilling
G74	Cut into	The spindle stops the	Cut into	Left spin attack
G76	Cut into	oriented spindle stop	Cut into	Fine boring cycle
G80	not have	not have	not have	Cancel the fixed
G81	Cut into	not have	quick travel	Drilling cycle
G82	Cut into	Time out	quick travel	Drilling cycle
G83	Cut into each	not have	quick travel	Discharge drilling
G84	Cut into	Spindle stops spindle	Cut into	Attack wire cycle
G85	Cut into	not have	Cut into	Boring circulation
G86	Cut into	The spindle stop	quick travel	Boring circulation
G87	Cut into	The spindle is turning	quick travel	Back boring
G89	Cut into	Time out	Cut into	Boring circulation

12.1 1-hole processing and fixation cycle

Generally, hole processing, fixed cycle usually consists of 6 actions, as shown in Figure 2-36:

Action 1: X, Y axis positioning, so that the tool quickly positioning to the hole processing position;

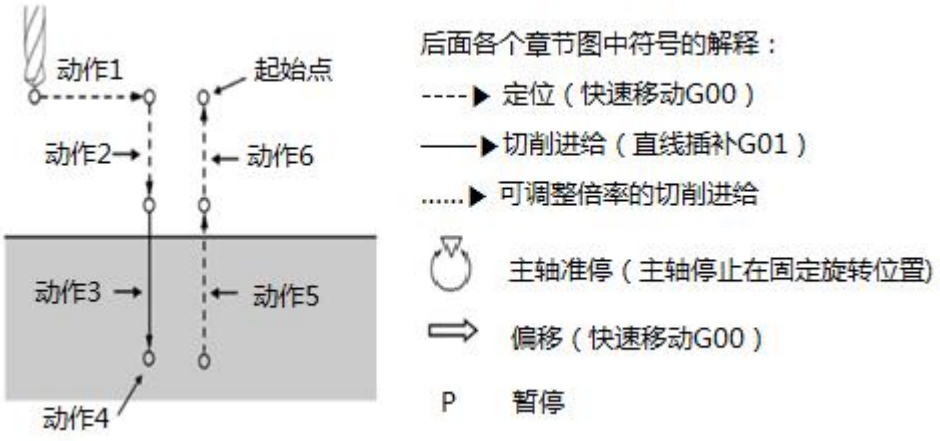
Action 2: quickly move to R point, the tool from the initial point to R point;

Action 3: hole processing, in the way of cutting into the feed;

Action 4: the action at the bottom of the hole, including pause, spindle quasi-stop, tool shift and other actions;

Action 5: return to the R point and continue the processing of the hole;

Action 6: quick return to the initial point, usually return to the initial point.



graph 2-37

1． The initial plane

The initial plane is a plane specified for safety cutting. The distance from the initial plane to the part surface can be arbitrarily set at a safe height, in the plane where the initial point is located as shown in FIG. 2-37. When processing several holes with the same tool, the G98 function is used to bring the tool back to the initial point in the initial plane, otherwise the G99 is used to return to the R point.

2． R drop

The plane where R point is located is also called R point reference surface, this plane is the knife from moving forward to the height of the plane, the distance from the workpiece mainly considers the change of the surface size of the workpiece, generally desirable 2~5mm, using G99, the tool will return to the reference surface.

3． Hole bottom plane

When processing the blind hole, the bottom plane of the hole is the height of the Z axis of the bottom of the hole, and the general tool should extend the bottom plane of the workpiece for a distance, mainly to ensure that all the hole depth is processed to the size, drilling processing should also consider the impact of the drill tip on the hole depth.

The hole machining cycle is unrelated to the plane selection instruction (G17, G18 or G19), that is, no matter which plane is selected, the hole machining is positioned in the XY plane and drilled in the Z axis direction.

Three problems should be considered for the action order designation of the fixed cycle:

(1) Whether the coordinate data is used in the absolute value or in the incremental value mode;

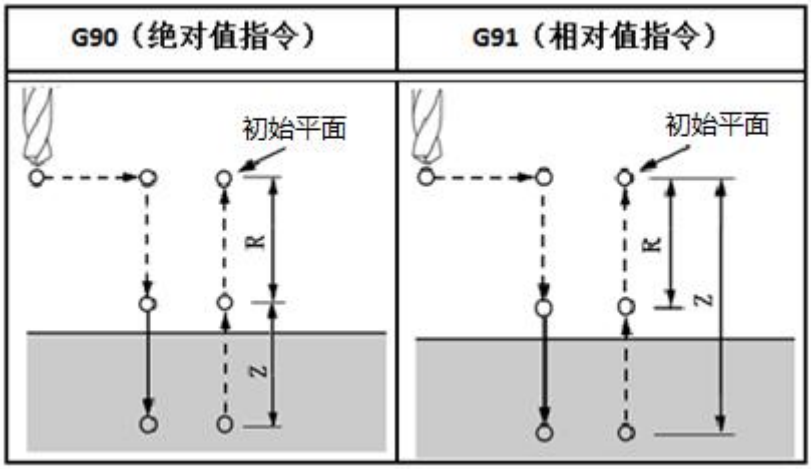
(2) Whether the return point plane is selected in the plane of the initial point or the R point;

(3) Consider what kind of hole processing cycle method, such as G73~G89 to be introduced below.

12.2 Main instructions affecting the hole processing cycle

12.2.1 Coordinate Programming Instructions (G90 / G91)

The data designation of R and Z in the fixed cycle instruction is related to the mode of G90 or G91. Figures 2-38 shows the coordinate calculation method when G90 or G91. In G90, R and Z take their end coordinate value; in G91, R is the distance from the initial point to point R, and Z is the distance from R to point Z in the bottom plane of the hole.



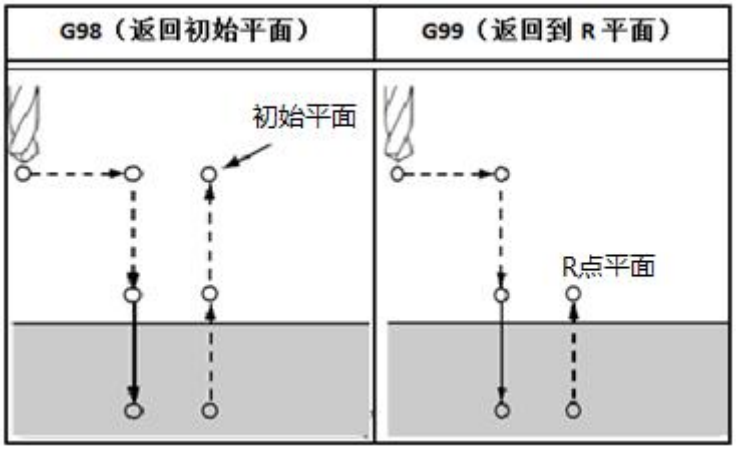
graph 2-38

12.2.2 Select the return plane (G98, G99)

G98 or G99 determines the plane the tool reaches on return. If G98 is ordered, starting with the program segment, the tool will return to the initial plane, and if G99 is ordered to the R point plane, as shown in Figure 2-39.

Usually when a set of identical holes is processed, the first hole is returned to R with G99, and the last hole is returned to the initial plane with G98.

Generally, if the processed hole is in a flat plane, the G99 instruction can be used, because the R point returns to the G99 mode for the next hole, and the R point is very close to the surface of the workpiece, which can shorten the processing time of the part. However, if the workpiece surface has a convex or bar higher than the processed hole, it is very likely to collide the tool and the workpiece when using G99. At this point, G98 should be used to make the Z axis return to the initial point and then locate the next hole, so that it is safer.



graph 2-39

12.3 3-hole processing and circulation mode

The general format of the hole machining cycle mode instruction is as follows:

G73～G89 X __Y __Z __R __Q __P __F __K __；

X _Y _ plane location coordinate value, either absolute or incremental;

Z Specifies the position of the bottom plane of the hole, either in absolute values or in incremental values;

R specifies the position of the plane where the R point is located, either in absolute or incremental values;

Q is used to specify each processing depth in G73 or G83, and the displacement amount is specified in G76 or G87. The Q-value is all incremental values, regardless of the choice of G91 and G90;

P is used to specify the pause time of the tool at the bottom of the hole, as the time unit of P in G04, that is, in ms, no decimal point;

F Specifies the hole machining and cutting feed speed. This instruction is a modal instruction, even if the fixed cycle is cancelled in the subsequent processing;

K For the number of repeats of the instruction hole, L1 is ignored. If the G90 mode is selected in the program, the tool repeats the machining in the position of the original hole; if G91 is selected, several equal distance holes distributed in a straight line can be processed, and L is only valid in the instructed program segment.

Cancel hole processing, use G80, and if any 01 group G code appears in the middle, the hole processing will be automatically cancelled, so G01, G00, G 00, G03 can cancel the fixed cycle, its effect is the same as G80.

12.3.1 (G73)

This cycle performs a high-speed deep-hole machining operation. Cut it in intermittently to reach the bottom of the hole, remove the metal debris from the hole, and process it simultaneously.

instruction format:

G73 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

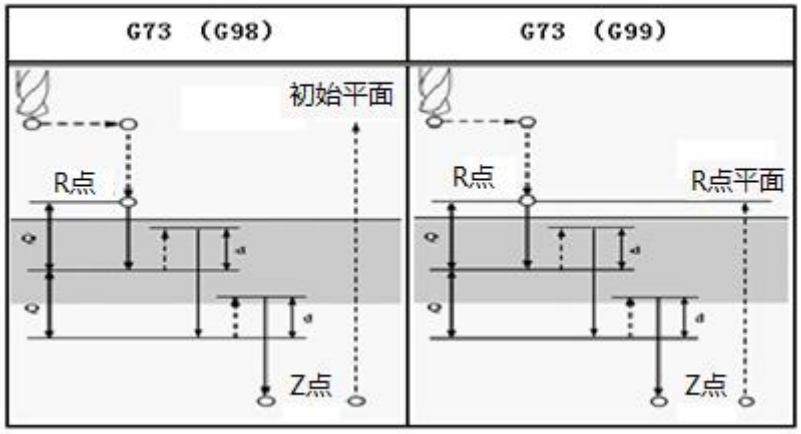
R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

Q_: Cutting depth for each cutting feed

F_: Cutting-in-feed speed

K_: number of replication



graph 2-40

explain:

movement:

The high-speed chip discharge drilling cycle performs intermittent feeding along the Z axis. When using this cycle, the blade is processed, and the blade discharge amount d can pass the parameter NO.3808 is set up to achieve high-speed and efficient drilling.

Q represents the cutting depth for each cut feed and must be specified as a positive value, and if Q is specified as a negative value, the symbol will be ignored.

additional function:

When the G73 code and M code are specified in the same program segment, the M code is executed with the positioning action and then the next drilling action.

When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Rotation of the main shaft:

Before using the G73 drilling cycle, the spindle rotation must be specified by the M code.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed cycle: G73 and 01 group G codes cannot be used in the same program segment, otherwise G73 is cancelled.

3． Cancel the fixation cycle before switching the drilling shaft.

4．Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, Q), the new cycle parameters will not take effect; when the subsequent program specifies both hole data and new cycle parameters (Z, R, Q), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

M3 S2000；	The spindle is turning
G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120；	Position, drill hole 1, and return to point R
Y-550；	Position, drill hole 2, and return to point R
Y-750；	Position, drill hole 3, and return to point R
X1000；	Position, drill hole 4, and return to point R
Y-550；	Position, drill hole 5, and return to point R
G98 Y-750；	Position, drill 6 holes, and then return to the initial position plane
G80 G28 G91 X0 Y0 Z0；	Return to reference point
M5；	The spindle stops spinning
M30；	end of program

12.3.2 Fine boring cycle (G76)

This cycle is used for performing a high-precision boring process.

The spindle stops when it reaches the hole bottom, and the tool is withdrawn after leaving the surface of the workpiece.

instruction format:

G76 X_ Y_ Z_ R_ Q_ P_ F_ K _ ;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

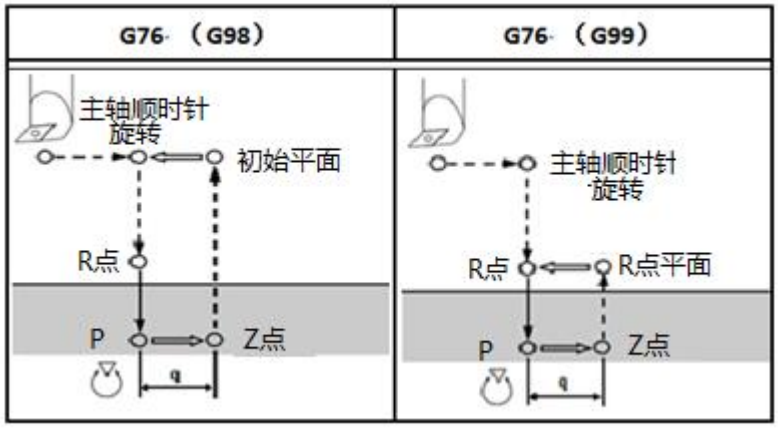
Distance from the initial position to point R (incremental value mode)

Q_: hole bottom offset

P_: Time out

F_: Cutting-in-feed speed

K_: number of replication



graph 2-41

explain:

movement:

The precision boring cycle is used for boring precision holes. When it reaches the bottom of the hole, the main shaft stops and quasi-stop, moves the offset, and exits the machining hole.

The spindle rotates for a fine boring cycle. When the bottom reaches the hole, the spindle performs a quasi stop and the tool moves in the opposite direction of the tip. This

ensures that the processing surface is not destroyed, and realizes the precision and effective boring processing.

additional function:

When the G76 code and M code are specified in the same program segment, the M code is executed while positioning the positioning action and then the next boring action is performed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Before boring cycles, the spindle rotation must be specified by the M code.

P is the bottom pause time in milliseconds and can only be a multiple of 100.

warn:

Q (hole bottom offset) is the modal value and must be specified with care since it also acts on G73 and G83.

pay attention to:

1. Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2. Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

3. Cancel fixed cycle: G76 and 01 group G codes cannot be used in the same program segment, otherwise G76 is cancelled.

4. Cycle parameters: After the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, Q, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, Q, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segment.

Program examples:

G0 G90 G54 X50. Y160. Z30. S1000 M3	Quickly positioning to hole 1, the spindle begins to rotate.
G98 G76 X50. Y160. Z-10. F30. R3. P1000 Q5.	Do the boring cycle, boring the first hole, hole bottom orientation, move 5mm, and then return to the initial
X-50.	Bore the second hole, and then return

Y-160.	Bore the third hole, and then return to
X50.	Bore the fourth hole, and then return to
G80	Fixed cycle is cancelled.
M30	end of program

12.3.3, Common Drill cycle (G81)

This cycle is used for the usual drilling process.

The feed is cut to the bottom of the hole and the tool exits in a fast moving manner.

instruction format:

G81 X_ Y_ Z_ R_ F_ K_ ；

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

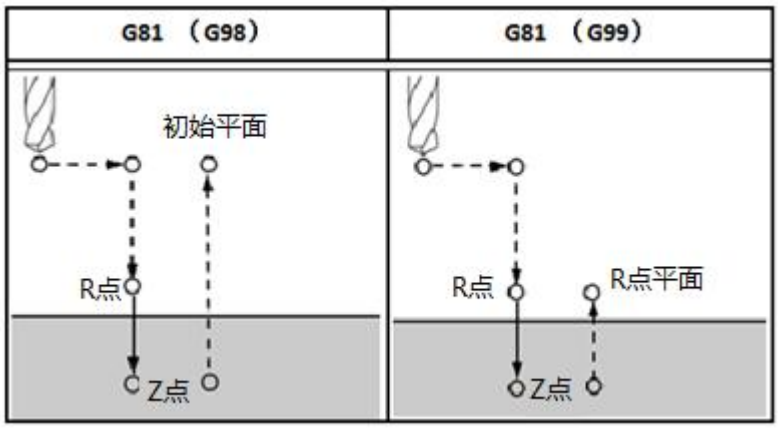
Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

F_: Cutting-in-feed speed

K_: number of replication



graph 2-42

explain:

movement:

After positioning along the X axis and Y axis, move quickly to R point, drill from R point to Z point, and then move quickly and back.

additional function:

When the G81 code and the M code are specified in the same program segment, the M code is executed along with the positioning action, and the next drilling action is followed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Before using the G81 drilling cycle, the spindle rotation must be specified through the M code.

pay attention to:

1. Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2. Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

3. Cancel the fixed cycle: G81 and 01 group G codes cannot be used in the same program segment, otherwise G81 is cancelled.

4. Cycle parameters: after the complete drilling cycle instruction specifies the new cycle parameters separately (Z, R), the new cycle parameters cannot take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G00 G90 G54 X50. Y160. Z30. S1000 M03	Quickly positioning to hole 1, the spindle begins to rotate.
G98 G81 X50. Y160. Z-10. F30. R3.	Cydrilling cycle, drill the first hole and return to the initial plane.
X-50.	drill the second hole and return to the initial
Y-160.	drill the third hole and return to the initial
X50.	drill the fourth hole and return to the initial
G80	Fixed cycle is cancelled.
M30	end of program

12.3.4, Common Drill cycle (G82)

This cycle is used for the usual drilling process.

The cutting is cut to the bottom of the hole and then retracted from the bottom in a fast moving manner.

This cycle can improve the accuracy of the hole depth.

instruction format:

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

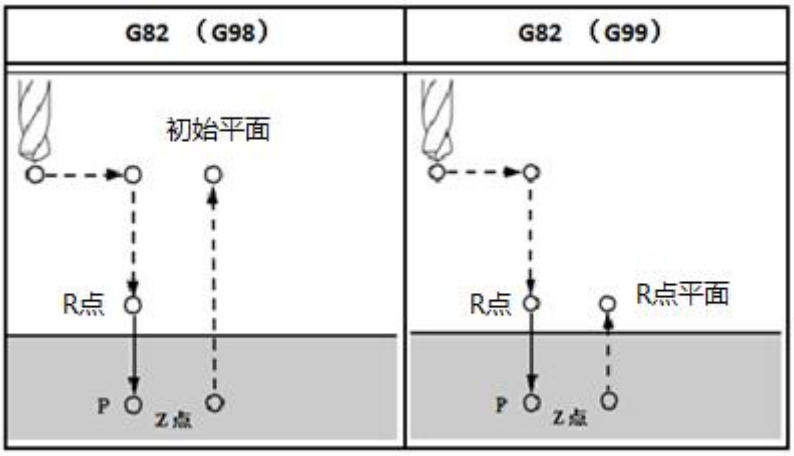
R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

P_: Time out

F_: Cutting-in-feed speed

K_: number of replication



graph 2-43

explain:

movement:

After positioning along the X and Y axes, move quickly to R and perform drilling from R to Z. When you reach the bottom of the hole, perform a pause and then move back.

additional function:

When the G82 code and M code are specified in the same segment, the M code is executed along with the positioning action and the next drilling action is followed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Spindle rotation:

Before using the G82 drilling cycle, the spindle rotation must be specified by the M code.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

P is the hole bottom pause time in milliseconds, and the specified value is a multiple of 100.

G82 is similar to G81, the only difference is that G82 increases pausing at the bottom of the hole, so it is suitable for the processing of blind hole, hole or boring step hole to improve the surface of the bottom, machining accuracy, while G81 is only suitable for the processing of general holes.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel the fixed cycle: G82 and 01 group G codes cannot be used in the same program segment, otherwise G82 is cancelled.

3． Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

4．Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G0 G90 G54 X50. Y160. Z30. S1000 M3	Quickly positioning to hole 1, the spindle begins to rotate.
G98 G82 X50. Y160. Z-10. F30. R3. P2000	Cyring, drill the first hole, pause the bottom for 2 seconds, and then return to the initial plane.
X-50.	drill the second hole, pause the bottom for 2
Y-160.	drill the 3rd hole, pause the bottom for 2 seconds,
X50.	drill the fourth hole, pause the bottom for 2
G80	Fixed cycle is cancelled.
M30	end of program

12.3.5, Drilling Cycle (G83)

This cycle processes the deep hole.

The cycle is intermittently cut to reach the bottom of the hole and removing the metal debris from the hole.

instruction format:

G83 X_ Y_ Z_ R_ Q_ F_ K_;

X_Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

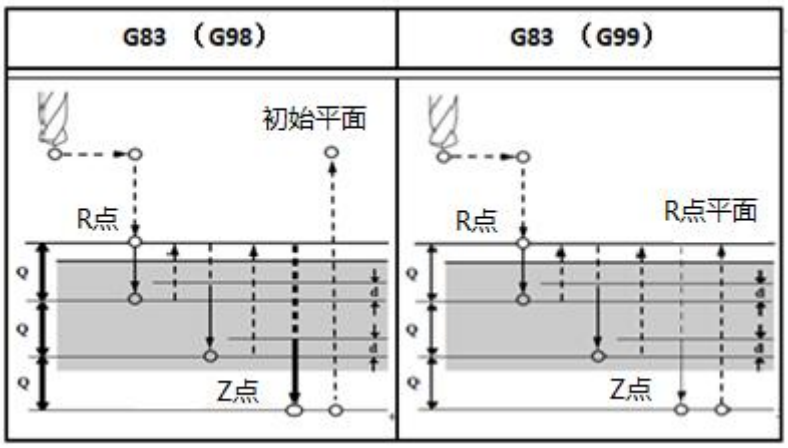
R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

Q_: Cutting depth for each cutting feed

F_: Cutting-in-feed speed

K_: number of replication



graph 2-44

explain:

movement:

The G83 instruction is also used for deep hole processing, performing intermittent cutting to the bottom of the hole and removing debris from the hole during drilling. Hole processing action as shown in figure 2-43, along the X axis and Y axis positioning, quickly move to R point, from R to Z point to perform debris drilling processing, when reaching the bottom of the hole, quickly move back, the knife amount d (tool intermittent drop from fast

forward to work into the point to the previous cutting into the distance between drop) can be through the parameter NO.3963 Setting-up.

additional function:

Slightly different from the G73 is each tool intermittent feed back to the R point plane. Q represents the cutting depth for each cut feed and must be specified as a positive value, and if Q is specified as a negative value, the symbol will be ignored. When the G83 code and the M code are specified in the same program segment, the M code is executed along with the positioning action, and the next drilling action is followed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Before using the G83 drilling cycle, the spindle rotation must be specified by the M code.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed cycle: G83 and 01 group G code cannot be used in the same program segment, otherwise G83 is cancelled.

3． Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

4．Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, Q), the new cycle parameters will not take effect; when the subsequent program specifies both hole data and new cycle parameters (Z, R, Q), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

M3 S2000 ；	The spindle is turning
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120.；	Position, drill hole 1, and return to point R
Y-550.；	Position, drill hole 2, and return to
Y-750.；	Position, drill hole 3, and return to
X1000.；	Position, drill hole 4, and return to

Y-550.;	Position, drill hole 5, and return to
G98 Y-750.;	Position, drill hole 6, and return to the initial position plane
G80 G28 G91 X0 Y0 Z0;	Return to reference point
M5;	The spindle stops spinning
M30;	end of program

12.3.6 Boring Cycle (G85)

This cycle is used for boring and the tool returns at the rate of cutting feed.

instruction format:

G85 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

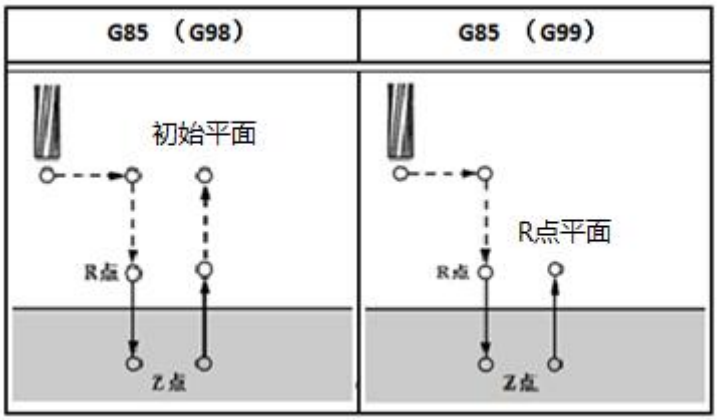
Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

F_: Cutting-in-feed speed

K_: number of replication



graph 2-45

explain:

movement:

After positioning along the X axis and Y axis, move quickly to the R point, perform the boring process from R point to Z point, and perform the cutting drop back to the R point at the bottom.

additional function:

When the G85 code and the M code are specified in the same program segment, the M code is executed along with the positioning action, and the next drilling action is followed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Turning of the spindle must be specified before using the G85 boring cycle through the M code.

pay attention to:

1. Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2. Cancel fixed cycle: G85 and 01 group G code cannot be used in the same program segment, otherwise G85 is cancelled.

3. Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

4. Cycle parameters: after the complete drilling cycle instruction specifies the new cycle parameters separately (Z, R), the new cycle parameters cannot take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G0 G90 G54 X50. Y160. Z30. S200 M3	Quickly positioning to hole 1, the spindle begins to rotate.
G98 G85 X50. Y160. Z-10. F100. R3.	Do the boring cycle, bore the first hole, and then return to the initial plane
X-50.	Bore the second hole and return to the
Y-160.	Bore the third hole and return to the
X50.	Bore the 4th hole and return to the initial
G80	Fixed cycle is cancelled.
M30	end of program

12.3.7 Boring Cycle (G86)

This cycle is used for boring, and when reaching the bottom, the spindle stops and the tool moves quickly back.

instruction format:

G86 X_ Y_ Z_ R_ F_ K_;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

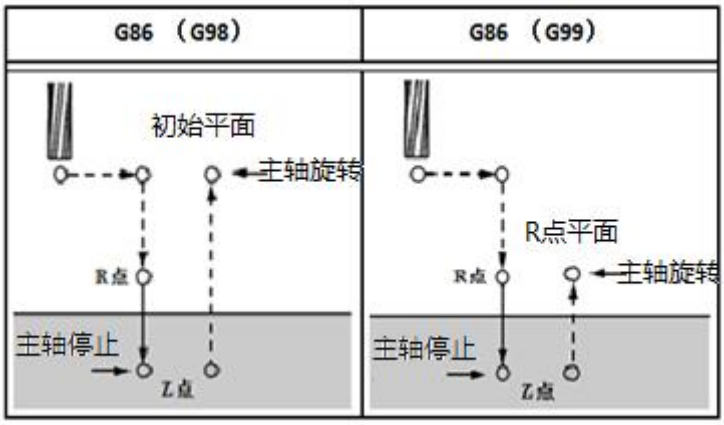
Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

F_: Cutting-in-feed speed

K_: number of replication



graph 2-46

explain:

movement:

After positioning along the X axis and Y axis, you quickly move to the R point, from the R point to the Z point, when reaching the bottom, the spindle stops, and then quickly move back.

additional function:

When the G86 code and the M code are specified in the same program segment, the M code is executed along with the positioning action, and the next drilling action is followed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Before using the G86 boring cycle, the spindle rotation must be specified by the M code.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed cycle: G86 and 01 group G codes cannot be used in the same program segment, otherwise G86 is cancelled.

3． Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

4．Cycle parameters: after the complete drilling cycle instruction specifies the new cycle parameters separately (Z, R), the new cycle parameters cannot take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G0 G90 G54 X50. Y160. Z30. S200 M3	Quickly positioning to hole 1, the spindle begins to rotate
G98 G86 X50. Y160. Z-10. F100. R3.	Do the boring cycle, bore the first hole, and then return to the initial
X-50.	Bore the second hole and return to
Y-160.	Bore the third hole and return to
X50.	Bore the 4th hole and return to the
G80	Fixed cycle is cancelled.
M30	end of program

12.3.8 Back boring cycle (G87)

This cycle is used for performing a high-precision boring process.

instruction format:

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

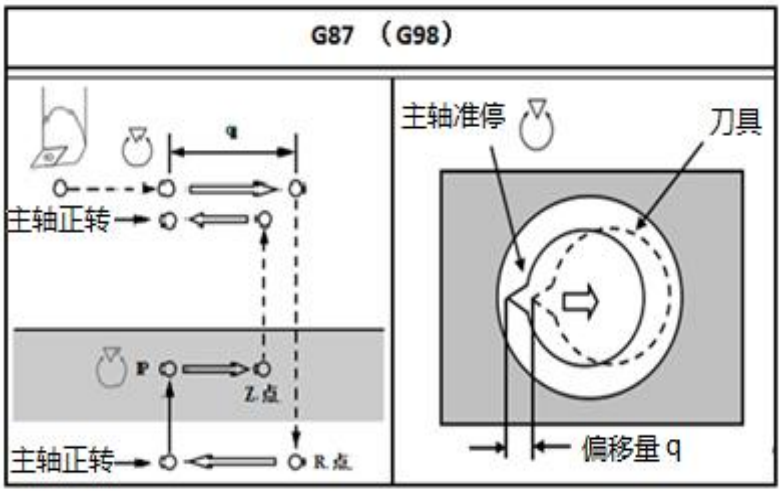
R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

Q_: hole bottom offset

P_: Time out

F_: Cutting-in-feed speed



graph 2-47

explain:

movement:

After positioning along the X axis and Y axis, the spindle quasi-stops, the tool moves to the opposite direction of the knife tip, and quickly moves to the R point at the bottom of the hole. Then the tool moves toward the tip and the spindle is bored along the Z axis to the Z point. After reaching the Z point, the spindle stops and the quasi stops, the tool moves to the tip in the opposite direction, and then the tool returns to the initial position. In the initial position, the tool moves the blade towards the tip, and the machining of the next program segment continues.

additional function:

When the G87 code and the M code are specified in the same program segment, the M code is executed while performing the positioning action, and then the next boring action is performed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Before using the G87 boring cycle, the spindle rotation must be specified through the M code.

P is the bottom pause time in milliseconds and can only be a multiple of 100.

warn:

Q (hole bottom offset) is the modal value and must be specified with care since it also acts on G73 and G83.

pay attention to:

1. Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2. Cancel the fixed cycle: G86 and 01 group G codes cannot be used in the same program segment, otherwise G87 is cancelled.

3. Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

4. Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G0 G90 G54 X50. Y160. Z30. S1000 M3	Quickly positioning to hole 1, the spindle begins to rotate.
G98 G87 X50. Y160. Z1. F30. R-10. P1000 Q3.	Performing the boring cycle, boring the first hole, orientation at the initial position, offset 3mm, stop at Z for 1 second, and then return to the initial plane.
X-50.	Bore the second hole, and then return to the initial plane
Y-160.	Bore the third hole, and then return to the initial plane
X50.	Bore the fourth hole, and then return to the initial plane

G80	Fixed cycle is cancelled.
M30	end of program

12.3.9 Boring Cycle (G89)

This cycle is used for boring and machining.

instruction format:

G89 X_ Y_ Z_ R_ F_ K_;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

P_: Time out

F_: Cutting-in-feed speed

K_: number of replication

explain:

movement:

This loop is almost identical to the G85. The difference is that the cycle can perform a pause at the very bottom of the hole.

additional function:

When the G89 code and the M code are specified in the same program segment, the M code is executed along with the positioning action, and the next drilling action is followed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

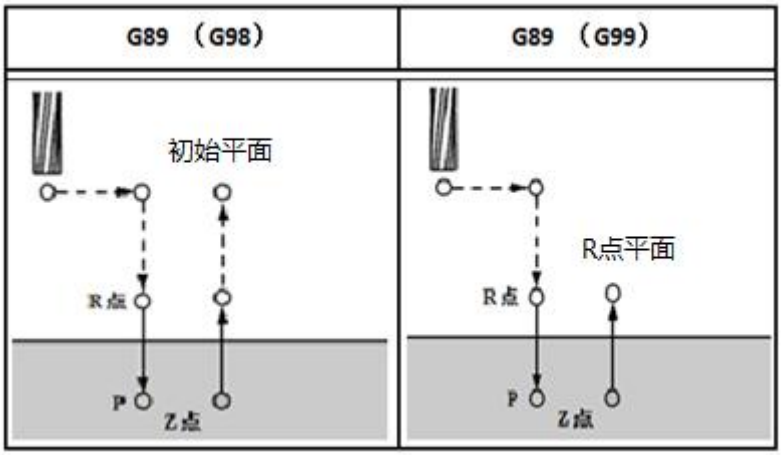
Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

Spindle rotation:

Spindle rotation must be specified before using the G89 boring cycle through the M code.

P is the bottom pause time in milliseconds and can only be a multiple of 100.



graph 2-48

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed loop: G89 and 01 group G codes cannot be used in the same program segment, otherwise G89 is cancelled.

3． Switching of the shaft: Please temporarily cancel the fixation cycle before switching the drilling shaft.

4．Cycle parameters: after the complete drilling cycle instruction specifies the new cycle parameters separately (Z, R), the new cycle parameters cannot take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G0 G90 G54 X50. Y160. Z30. S200 M3	Quickly positioning to hole 1, the spindle begins to rotate.
G98 G89 X50. Y160. Z-10. F100. R3. P1000	Perform the boring cycle, bore the first hole, pause the bottom for 1
X-50.	Bore the second hole, and then
Y-160.	Bore the third hole, and then
X50.	Bore the fourth hole, and then
G80	Fixed cycle cancellation
M30	end of program

12.4, and the tapping cycle

12.4.1 Left-Turn Tap Cycle (G74)

This cycle performs left-handed tapping, and the spindle rotates clockwise when reaching the bottom of the hole.

instruction format:

G74 X_ Y_ Z_ R_ P_ F_ K_;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

P_: Time out

F_: Cutting-in-feed speed

K_: number of replication

explain:

movement:

The spindle rotates counterclockwise to perform the tap, and to return at the hole bottom, the cycle is used to process the reverse thread.

additional function:

When G74 code and M code are specified in the same program segment, M code is executed while positioning the positioning action, and the next tapping action is performed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

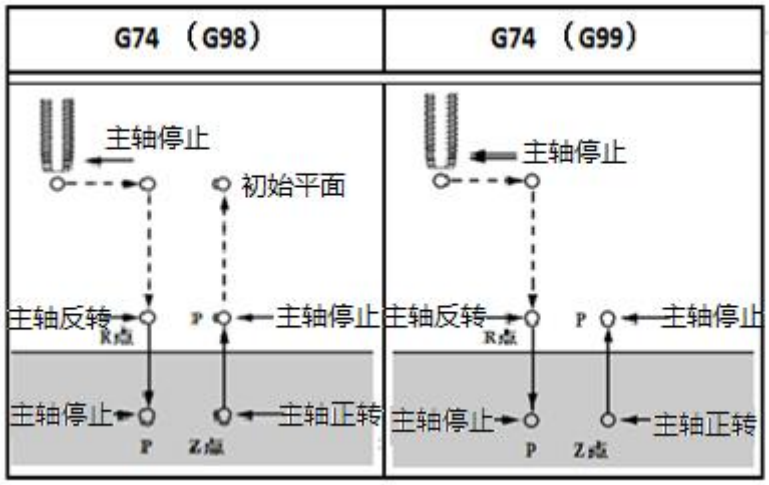
Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

P is the bottom pause time in milliseconds and can only be a multiple of 100.

Spindle rotation:

Before using the G74 left-handed tapping cycle, the spindle must be rotated counterclockwise by the M code.



graph 2-49

warn:

During the tapping cycle, adjust the feed rate and spindle rate, and use the feed holding button.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed cycle: G74 and 01 group G codes cannot be used in the same program segment, otherwise G74 is cancelled.

3． Axft switching: temporarily cancel the fixation cycle before switching the tapping shaft.

4．Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G00 G90 G54 X50. Y160. Z30. S100 M3 ；	Quickly locate to hole 1, and the spindle begins to rotate clockwise.
G98 G74 X50. Y160. Z-10. F100. R3. P1000 ；	Conduct the tapping cycle, tap the first hole, and then return to the initial plane, and
X-50. ；	Tap the second hole and return to the initial
Y-160. ；	Tap the third hole and return to the initial
X50. ；	Tap the 4th hole and return to the initial
G80 ；	Fixed cycle is cancelled.
M30 ；	end of program

12.4.2 Tap Cycle (G84)

This cycle performs the tapping, and the spindle rotates in the opposite direction when arriving at the pore bottom.

instruction format:

G84 X_ Y_ Z_ R_ P_ F_ K_;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of the bottom (absolute mode) from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value (absolute value mode) distance from initial position to R point (incremental value mode)

P_: Time out

F_: Cutting-in-feed speed

K_: number of replication

explain:

movement:

The spindle rotates the tapping, and to return, the spindle rotates in the opposite direction, in order to process the thread.

additional function:

When G84 code and M code are specified in the same program segment, M code is executed while positioning the positioning action, and the next tapping action is performed. When assigning the number of repeats K, execute M code only in the first hole and not M code for later holes.

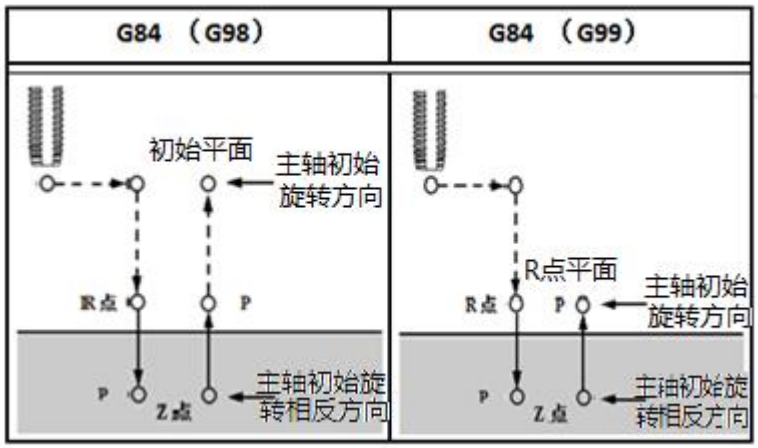
Length compensation:

When you specify the tool length bias (G43, G44, G49) in the fixed cycle program segment, add the bias while locating to the R point.

P is the bottom pause time in milliseconds and can only be a multiple of 100.

Spindle rotation:

Spindle rotation prior to using the G84 tapping cycle must be specified by M code S.



graph 2-50

warn:

During the tapping cycle, adjust the feed rate and spindle rate, and use the feed holding button.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed cycle: G84 and 01 group G codes cannot be used in the same program segment, otherwise G84 is cancelled.

3． Axft switching: temporarily cancel the fixation cycle before switching the tapping shaft.

4．Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Program examples:

G00 G90 G54 X50. Y160. Z30. S100 M3 ；	Quickly locate to hole 1, and the spindle begins to rotate
G98 G84 X50. Y160. Z-10. F100. R3. P1000 ；	Conduct the tapping cycle, tap the first hole, and then return to the initial plane, and pause at the bottom
X-50. ；	Tap the second hole and return to the
Y-160. ；	Tap the third hole and return to the
X50. ；	Tap the 4th hole and return to the
G80 ；	Fixed cycle is cancelled.

M30 ； end of program

Left-handed rigid tapping cycle (G74)

In the rigid tapping mode, the spindle motor works like a servo motor, which can realize high-speed and high-precision tapping.

instruction format:

G74 X_ Y_ Z_ R_ P_ F_ K_

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

P_: Time out

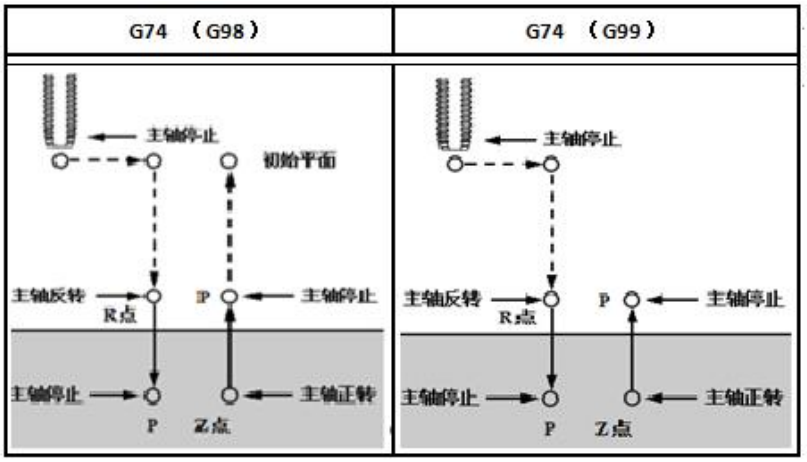
F_: Cutting-in-feed speed

K_: number of replication

Specify the rigid mode:

Specify M29 S_ _ _ _ prior to the tapping command section;

Specify M29 S_ _ _ _ in the program section of the tapping command.



graph 2-51

explain:

movement:

After positioning along the X and Y axes, move quickly to R and tap from R to Z. When the tapping is finished, the spindle stops and stops, then the spindle rotates in the

positive direction and returns to point R, the spindle stops and then moves quickly to the initial position. When the tapping cycle is executing, the default feed rate and spindle speed rate are 100%, but the fallback speed rate can be passed by the parameter NO.38,31 Adjustment.

feed:

In each minute feed (G94), thread lead = feed speed spindle speed; in each feed (G95), feed speed.

Warning: Adjust feed ratio and spindle ratio during tapping cycle, and use feed hold button.

pay attention to:

1. Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2. Cancel fixed cycle: G74 and 01 group G codes cannot be used in the same program segment, otherwise G74 is cancelled.

3. Axft switching: temporarily cancel the fixation cycle before switching the tapping shaft.

4. Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment specifies the new cycle parameters separately (Z, R, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Every-minute feed method:

G0 G90 G54 X50. Y160. Z30.	Quickly positioning to well 1
G94	Specifies each minute feed mode
M29 S200	Specifies the rigid tapping mode
G98 G74 X50. Y160. Z-10. F200. R3.	Perform the tapping cycle, tap the first hole, and then return to the initial plane
X-50.	Tap the second hole and return to the initial
Y-160.	Tap the third hole and return to the initial
X50.	Tap the 4th hole and return to the initial plane
G80	Fixed cycle is cancelled.
M30	end of program

For each transfer method:

G0 G90 G54 X50. Y160. Z30.	Quickly positioning to well 1
G95	Specifies the mode of each transfer
M29 S200	Specifies the rigid tapping mode

G98 G74 X50. Y160. Z-10. F2. R3.	Perform the tapping cycle, tap the first hole, and then return to the initial plane
X-50.	Tap the second hole and return to the initial
Y-160.	Tap the third hole and return to the initial
X50.	Tap the 4th hole and return to the initial plane
G80	Fixed cycle is cancelled.
M30	end of program

Rigid tapping cycle (G84)

In the rigid tapping mode, the spindle motor works like a servo motor, which can realize high-speed and high-precision tapping.

instruction format:

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

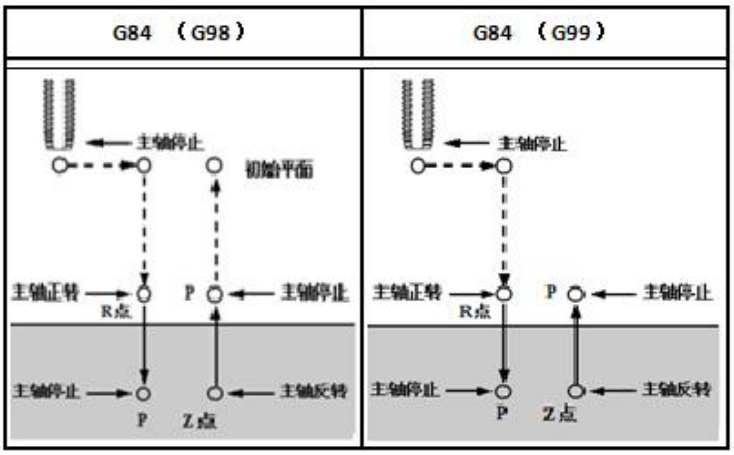
P_: Time out

F_: Cutting-in-feed speed

Specify the rigid mode:

Specify the M29 S_ _ _ _ prior to the tapping command section

Specify the M29 S_ _ _ _ in the program section of the tapping command



graph 2-52

explain:

movement:

After positioning along the X and Y axes, move quickly to R and tap from R to Z. When the tapping is finished, the spindle stops and stops, then the spindle rotates in the positive direction and returns to point R, the spindle stops and then moves quickly to the initial position. When the tapping cycle is executing, the default feed rate and spindle speed rate are 100%, but the fallback speed rate can be passed by the parameter NO.38,31 Adjustment.

feed:

In each minute feed (G94), thread lead = feed speed spindle speed; in each feed (G95), feed speed.

Warning: Adjust feed ratio and spindle ratio during tapping cycle, and use feed hold button.

pay attention to:

1． Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2． Cancel fixed cycle: G84 and 01 group G codes cannot be used in the same program segment, otherwise G84 is cancelled.

3． Axft switching: temporarily cancel the fixation cycle before switching the tapping shaft.

4．Cycle parameters: after the complete drilling cycle instruction, the subsequent program segment specifies the new cycle parameters separately (Z, R, P), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segments.

Every-minute feed method:

G0 G90 G54 X50. Y160. Z30.	Quickly positioning to well 1
G94	Specifies each minute feed mode
M29 S200	Specifies the rigid tapping mode
G98 G84 X50. Y160. Z-10. F200. R3.	Perform the tapping cycle, tap the first hole, and then return to the initial plane
X-50.	Tap the second hole and return to the initial
Y-160.	Tap the third hole and return to the initial
X50.	Tap the 4th hole and return to the initial plane
G80	Fixed cycle is cancelled.

M30	end of program

For each transfer method:

G0 G90 G54 X50. Y160. Z30.	Quickly positioning to well 1
G95	Specifies the mode of each transfer
M29 S200	Specifies the rigid tapping mode
G98 G84 X50. Y160. Z-10. F2. R3.	Perform the tapping cycle, tap the first hole, and then return to the initial plane
X-50.	Tap the second hole and return to the initial
Y-160.	Tap the third hole and return to the initial
X50.	Tap the 4th hole and return to the initial plane
G80	Fixed cycle is cancelled.
M30	end of program

Rtapping cycle (G74, G84)

Deep hole tapping is difficult in rigid tapping because the tool prevents movement or increases resistance. In such cases, the dust removal rigid tapping is very suitable for use.

In this cycle, the tool enters several times to the bottom of the hole. There are two modes of deep hole tapping: high-speed deep hole tapping cycle and standard deep hole tapping cycle, which can be passed through the parameter NO.3811 for selection.

instruction format:

G74（G84） X_ Y_ Z_ R_ P_ Q_ F_ K_

X_ Y_: The hole-position data

Z_: Absolute coordinate value of hole bottom (absolute value mode)

Distance from R to the bottom of the hole (incremental mode)

R_: Absolute coordinate value of the R point (absolute value mode)

Distance from the initial position to point R (incremental value mode)

P_: Time out

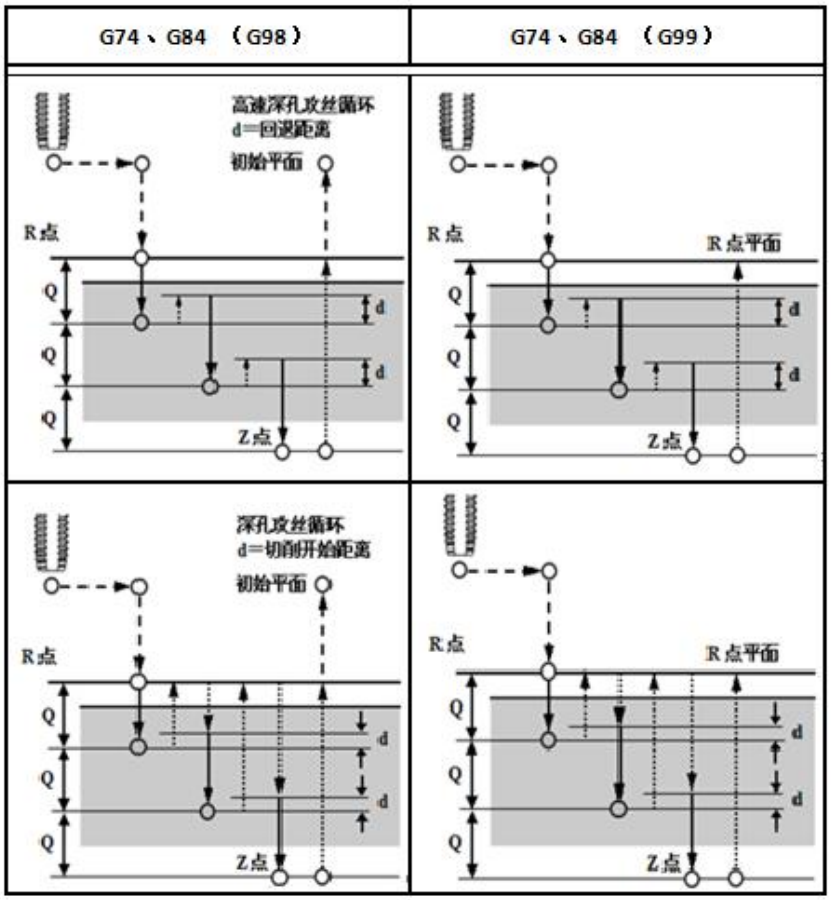
Q_: Cutting depth for each cutting feed

F_: Cutting-in-feed speed

Specify the rigid mode:

Specify M29 S_ _ _ _ prior to the tapping command section;

Specify M29 S_ _ _ _ in the program section of the tapping command.



graph 2-53

explain:

movement:

High speed deep hole tapping cycle: after positioning along the X axis and Y axis, quickly move to the R point, cut from the R point at each blade depth Q, and then the tool backback distance d (parameter NO.3833), the backback velocity multiplier by the parameter (NO.3831) The setting of the device. Continue cutting feed from d, when reaching the Z point, the spindle stops, then rotates in the opposite direction and retreats.

feed:

Standard deep hole tapping cycle: after positioning along the X axis and Y axis, quickly move to R point, cut from R point at each blade depth Q, and then step back to R point, step back, speed multiplier by parameter (NO.3831) The setting of the device. Move from the R point (this speed is the same as the fallback speed) to the distance of d from the last cutting end point (parameter NO.3833) is the position where cutting starts. When reaching Z, the spindle stops and then rotates in the opposite direction.

Warning: Adjust feed ratio and spindle ratio during tapping cycle, and use feed hold button.

pay attention to:

1. Tool radius compensation: the tool radius compensation is ignored when using the fixed cycle instructions.

2. Cancel fixed cycle: G74, G84 and G 01 cannot be used in the same segment, otherwise G84 is cancelled.

3. Axft switching: temporarily cancel the fixation cycle before switching the tapping shaft.

4. Cycle parameters: After the complete drilling cycle instruction, the subsequent program segment separately specifies the new cycle parameters (Z, R, P, Q), the new cycle parameters will not take effect; when the subsequent program specifies both hole position data and new cycle parameters (Z, R, P, Q), the new cycle parameters take effect immediately, and the mode takes effect in the subsequent program segment.

12.5 Fixed cycle cancellation (G80)

Used to cancel the fixation cycle.

instruction format:

G80;

Note: Cancel all fixed cycles, perform the normal operation, and the R and Z points are also cancelled. This means that, in an incremental manner, R =0 and Z =0. Other drilling data were also cancelled.

Program examples:

M3 S100; Main spindle, S100

G90 G98 G81 X10. Y10. Z-30. R-10. F100; after positioning, cycle drilling 1

X20; hole 2

Y20; hole 3

X30; hole 4

Y30; hole 5

G80 G91 G28 Z0; Cancel the fixed cycle and return to the reference point

M05;, and the spindle stops

M30;, the program ends and returns to the beginning of the program

The third part, the operation part

1 Operation panel

1.1 Panel division

The CNC system adopts the integrated panel design, including USB interface, LCD display area, MDI panel area, soft button operation area and operation panel area.

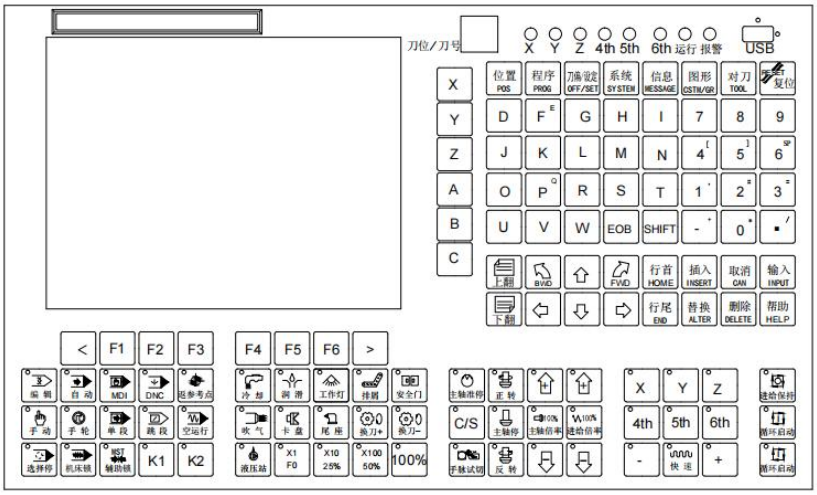


Figure 3-1 System panel

1.2 Division function division

1.2.1 LCD, display area

The milling machine system uses an 8-inch high-bright color LCD display with a resolution of 800600 p i x el.

1.2.2 MDI, panel area

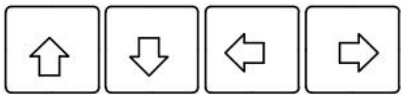
The MDI panel keys are as shown in Figure 3-2:

Figure 3-2 The MDI panel

The MDI panel function page key is used to select various display screens. There are eight kinds of display screens: POS position, PROG program, OFF / SET knife bias / setting, SYSTEM system, MESSAGE information, CSTM / GR graphics, and HELP help. The lathe version control system supports T OOL knife (quick knife) popover screen. The page is explained as follows:

Table 3-1

key	name	Functional description and use
	Reset key (RESET)	Remove the alarm, CNC reset, and feed output stop.
	Address / number key	Use to write programs with letters, numbers, and other characters.
	(EOB) key	Use to end with ";" for writing the program.
	(SHIFT) upper key	Press SHIFT when the program enters the edit state, and after the prompt, '_' becomes '^', indicating that the upper file character can be typed.

	Input key (INPUT)	For input parameters, compensation amount and other data. During program editing, move the cursor selects a line of program segment. Press this key to select a single character in the program segment. The program machining percentage can also be displayed in the automatic monitoring interface.
	Cancel the (CAN) key	Eliminate the characters or symbols entered into the key input buffer register.
	Help key	Press the help key at different interfaces, and there will be a text comment prompt.
	The cursor moves the key	 In fields, move a field down, up, left, and right.
		 When the program edit, the cursor jumps to the beginning and end of the current editing line.
	Turn the page key	 Change the page of the display screen in the reverse direction (turn the page up and down).

		 Change the left and right page direction of the display screen (left and right page turning).
	ins	Edit operation for inserting, deleting, and replacing changes of the program.
	delete key	
	Replace the key	
	Location interface key	Press the position interface key to display multiple mode coordinates, and processing information.
	Program interface key	Press the program interface key to display and write the processing program.
	Kade bias / setting key	Press the knife bias / setting key to manage the tool-related compensation, set the coordinate system, and set some system functions, system time and system password.
	System key	It can be used for system parameter editing settings, PLC and other related parameter information settings.
	Information key	Press the information key for alarm information display, alarm resume display, waveform diagnosis, etc.
	Graphics key	Press the graphic key, can be used to process the track graphics display and figure parameter setting.

对刀 TOOL	To the knife key	The milling machine version control system is invalid.

1.2.3, Software operation area

There are 8 selected soft keys in this system, corresponding to 6 function keys and left and right page turning keys on the function interface.

1.2.4 Operation panel

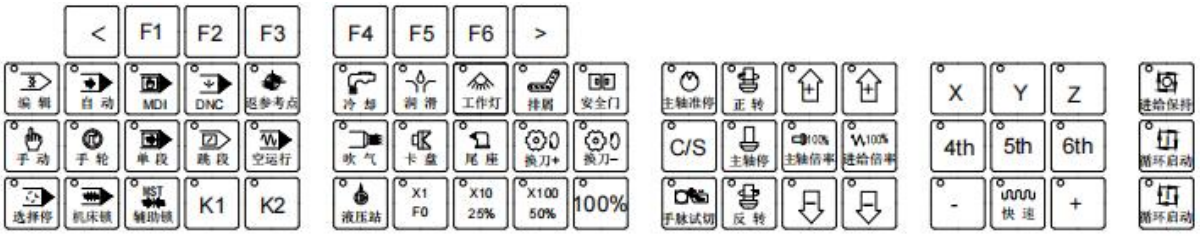


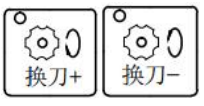
Figure 3-3 Operation panel

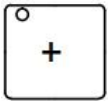
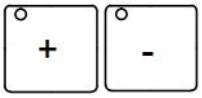
The key functions of the operation panel of this system are described as follows:

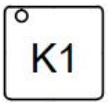
Table 3-2

key	name	Functional description and use
编辑	edit mode	Enter the editing operation mode, the automatic mode, MDI, DNC mode to switch to the editing mode, the system immediately stop running in the current program segment;
自动	Automatic way	Enter the automatic operation mode, the system runs the internal memory program;
MDI	MDI way	When entering the MDI (input) operation mode and switching to the MDI mode under the automatic operation mode, the system immediately stops running in the current program segment;
DNC	DNC way	When entering the DNC operation mode and switching to the MDI mode under the automatic operation mode, the system immediately stops the operation in the current program segment;

	Single paragraph	Single-segment / continuous running status switch under automatic, MDI and DNC mode;
	dry running	Empty program operation under automatic, MDI and DNC mode;
	Jump section	Program segment skipping switch, the first "/" symbol of the program skipping segment, when the hop segment function is valid, the system skipped the program segment to continue to run;
	Choose to stop	Automatic, MDI, DNC mode program, add M01 code, when the selection of stop valid, will be executed to stop;
	Machine tool lock	Automatic, MDI, DNC, manual, hand wheel mode of the machine tool feed shaft action output is invalid;
	Auxiliary lock	The output of M, S and T auxiliary code functions in automatic, MDI and DNC mode is invalid;
	Handwheel mode	Enter the hand wheel mode, move the machine tool by shaking the hand wheel, the optional multiplier X1, X10, X100;
	manual mode	Enter the manual mode, and move the machine tool through the panel button;
	Return to reference point	Return to the reference point mode;
	feed hold	Automatic, MDI, DNC mode, press this key program to stop running;
	Cycle start	Automatic, MDI, DNC mode, press this key to start running;

	cooling	Open / close the coolant device;
	work light	On / off working lighting lighting;
	lubricating	Open / close the machine tool lubrication function, which can be manually controlled;
	chip removal	Open / close the dust exhaust machine device in the machine;
	exit	Open / close the machine tool safety door function;
	scavenging	Open / close the air blowing device;
	cartridge	The key function of the milling machine version control system is invalid;
	tailstock	The key function of the milling machine version control system is invalid;
	hydraulic pressure station	Open / close the hydraulic station device in the machine;
	tool changing	 Control the knife library clockwise / counter clockwise rotation, the machine tool to support this function;

	Spindle function	 Manually control the spindle clockwise turn, spindle stop, spindle counterclockwise reversal;
	Spindle ratio	 In the automatic operation, the spindle rate is adjusted, the system default is 16 levels, 0% ~150% adjustable;
	Feed rate	 In automatic operation, the feed rate is adjusted, the system default total 16 levels, 0% ~150% adjustable;
	The main shaft quasi stop	Execute the spindle quasi-stop, the spindle can be defined to stop to an angle. Used for lathe equipment automatic mechanical arm material and for milling machine equipment knife library knife orientation;
	C/S pattern	Implement M29 into the position C / S control mode;
	Hand pulse test cut	In the automatic operation mode, this function can be opened through the hand wheel shaking pulse drive processing program operation;
	Each axis is selected	For shaft selection of X / Y / Z / 4 / 5 / 6 axis in manual wheel and manual mode;
	direction of motion	 For manual mode, X / Y / Z / 4 / 5 / 6 axis motion direction selection and execution;

	quick movement	Used for manual fast movement, after opening this function. The shaft movement speed will operate at a fast moving speed;
	Hand wheel ratio Fast rate	 Incremental / handwheel mode movements X1, X10, X100 Quick rate selection: default F0,25%, 50%, 100% four levels
	Custom key	K 1-K 7 for the system custom function expansion button;

1.2.5 Handheld unit

The handheld unit is composed of hand shaking pulse generator and coordinate axis selection switch, which is used for hand shaking mode and incremental feed coordinate axis.

The appearance and specific parameters are subject to the actual order of the machine tool manufacturer.

2 System power-on, shutdown and safe operation

2.1 Power system

The following must be confirmed before the system:

1. The machine tool condition is normal.

2. The power supply voltage for the system and external signals meets the requirements.

3. The system power line and signal line are correct and firm.

4. To ensure safety, take a photo of the "emergency stop" button first.

5. After the machine is powered on, wait for the system to turn on.

After the system self-test is normal and initialized, the actual location interface is displayed as shown in Figure 3-4:

graph 3-4

System startup and operation steps:

●When the system starts up and enters the system operation interface, the initial working mode is displayed as an "emergency stop" alarm;

● In order to control the system operation, it is necessary to rotate right and pull up the red "emergency stop" button in the lower right corner of the operating table to eliminate the alarm reset, and press the reset operation;

- Select the system "return reference point" mode to perform the return reference operation.

2.2 Shutdown

Before the system shutdown, confirm the following items:

1．Each axis of the CNC machine tool is in a stop state;

2． Auxiliary functions (such as spindle, coolant water pump, etc.) are closed;

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

When cutting off the power supply, you must confirm the following items:

1．Check the system status display should be "stop" state;

2． Check that all the movable parts of the CNC machine tool are in a stopped state;

3．Press the emergency stop button and press the shutdown (power) button to power down.

Power-off in emergency:

During the operation of the machine tool, the power supply of the machine tool can be cut off immediately in an emergency case to prevent accidents. However, it must be noted that after cutting off the power supply, the coordinates of the system may deviate from the actual position, and the operations such as returning to zero and correction must be conducted again.

2.3 Safe operation

2.3.1 Reset operation

After pressing [reset], the system is in the reset state:

1．All axis movement stops;

2． M, S, T function stop;

3． Existing alarm information is cleared.

Can be used for system abnormal output, coordinate axis abnormal action.

Available by the parameter NO.3004 The system is in the clear / reset state when the change is reset.

2.3.2 Emergency stop

Press the emergency stop button during the operation of the machine tool, and the system enters the emergency stop state. Then the movement of the machine tool stops immediately, and all the output such as the rotation of the main shaft and the coolant are all closed.

Release the emergency stop button (although it is different from the machine tool manufacturer, but usually left turn this button can automatically jump up) after the emergency stop is lifted, but all the output needs to be restarted.

[pay attention to]

1． Confirm whether the cause of the fault is eliminated before removing the emergency stop button.

2． After the emergency stop button is removed, the reference point operation should be executed again to ensure the correctness of the coordinate position.

Generally, the emergency stop signal is the normally closed contact signal. When the contact is disconnected, the system enters the emergency stop state and makes the machine tool stop urgently. In this system, by the parameter NO.0020 (whether the emergency stop signal is often open) setting.

Normally closed emergency stop signal circuit connection is shown in Figure 3-5:

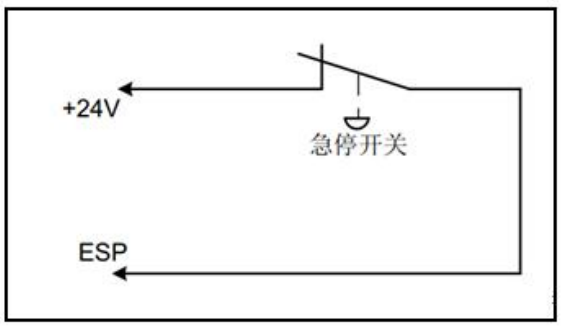


Figure 3-5 Normally closed emergency stop signal circuit connection

2.3.3, and return to the reference point

The premise of controlling the movement of the machine tool is to establish the machine tool coordinate system. Therefore, after connecting the power supply and reset, the system should first operate the machine tool for each axis back to the reference point.

The following are the steps to using the travel switch:

1．Select [return reference point] on the operation panel to enter the return reference mode;

2．According to the running direction of the machine return reference, select the moving axis (X, Y, Z, 4th, 5th, 6th) button, click the direction (+ -) button, and press it once;

3．The machine tool moves along the direction of the selection axis, and the speed of the feed axis is set as the fast shift speed (fast move F0 fast shift rate) before touching the deceleration point;

4．The speed after the deceleration point will slow down and move (the parameter NO.1207-1210 control);

5．The system supports multi-axis simultaneous return parameters, that is, in the process of one axis return parameters, other axes are operated without interference;

6． After returning to the reference point, the return reference point indicator light on the operation panel is on;

7．In the return state, clicking the spindle forward and reverse can realize the spindle point function.

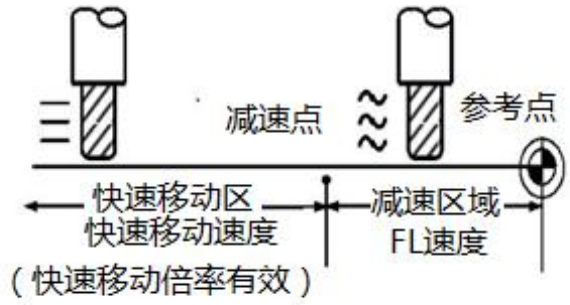


Figure 3-6 Schematic diagram of the process back to the reference point

2.4 Cycle start and feed hold

During the operation of the program, you can press the operation panel "feed hold" button to suspend the operation of the program. After the program is suspended, the "feed hold" button light is lit. However, the program pause does not quit the program processing state, so you can press the "cycle start" key to start the program operation again at any time.

When the program is in a suspended state, the system can be switched to manual or hand wheel mode to move the coordinate axis. When the system switches back to run the program again, the system will automatically default the program break point to continue to run. Therefore, the user should ensure that there is no motion interference in the return process.

When the system feed is maintained, switch to manual / manual wheel mode for other operations, the spindle rotation should be manually stopped. Start the spindle manually before starting the restart.

The "Circular Start" and "Feed Hold" keys in the system panel are used for [automatic] mode, [MDI] mode, and [DNC] mode.

2.5 Override protection

In order to avoid damage to the machine due to the feeding shaft, the machine must take overrange protection measures.

System overrun range, protection is divided into hardware overrun protection and software overrun protection, the user should set two kinds of overrun protection in the actual use.

2.5.1 Hardware override protection

The travel limit switch is installed at the positive and negative maximum stroke of the machine tool and each feed shaft. When the override occurs, the override shaft hits the limit switch and finally stops, and the system prompts the override alarm information.

Override during automatic operation: in automatic mode When the tool touches the limit switch while moving along a certain axis, all axes will slow down and finally stop, and the override alarm is displayed, and the program stops to the extended program segment.

Excessive during manual, hand wheel and automatic operation: in the operation process, as long as any shaft of the machine contacts the limit switch, immediately slow down and stop the movement of the shaft.

2.5.2 Software overshoot protection

Software travel range is determined by NO.1225-1230 parameter sets the maximum stroke of each axis, NO.1234-23 Set the maximum negative stroke of each axis, take the machine coordinate value as the reference value. If the moving shaft exceeds the soft limit position, an override alarm occurs.parameter NO.1243-1248 When the soft limit of each axis is valid.

Manual, manual wheel, automatic operation: in the operation process, as long as any shaft of the machine exceeds the limit of the software, immediately slow down and stop the movement of the shaft.

2.5.3 Release of the override alarm

The method to remove the hard limit and soft limit alarm is: press the [reset] key on the panel, and then in the manual or hand wheel mode, move the override axis in the reverse direction (such as forward out, the negative out, such as negative out), and move out of the overrange.

3. Interface display and data modification and setting

3.1 Boot-on interface display

After the system, enter the interface as shown in Figure 3-7:

Figure 3-7 status is shown

3.1.1 Status display

As shown in Figure 3-7, the previous behavior state of the most downstream soft key display row, showing the following information:

Operation mode: display the current working mode, including automatic mode, editing mode, DNC mode, MDI mode, manual mode, hand wheel mode, and return mode.

Program execution status: including program stop, program operation and program keep, etc.

Status display: including U disk, connection display, alarm prompt, reset indication.

Time: displays the current time of the system, 24 hours system, the format is "time: minutes: seconds".

3.1.2 Modal code display

Display the current modal code of the system, including G instruction, tool length compensation H, radius compensation D, currently called tool number T and M code.

3.1.3 Type in the data to display

The top row of the status display line shows the prompt and the key value being entered.

Tip: There is only indicated for the screen where '>' is typed. The unable screen has no prompt.

Upper prompt: press the SHIFT key in the input state, and after the prompt, '_' changes to '^', indicating that the upper character can be typed.

The typed key value is displayed after the prompt, which disappears when pressing the soft key [input] or any other valid input key.

3.1.4 Display of the program number and serial number

The program number and serial number are displayed on the upper right side of the system interface.

In non-programming mode, display the program number and sequence number of the last execution program. After using the program number [search] program and pressing the [Run] soft key, the retrieved program number and serial number are displayed.

3.1.5 Display of the doubling rate, actual speed, number of processed pieces, and program execution time

For you, the display interface and program monitoring interface display information about the current system operation:

1. Show the hand wheel / increment multiplier, fast multiplier, spindle multiplier, and feed multiplier, and show the actual feed speed F and spindle speed S.

2. Displays the execution time and number of pieces. When the program performs to M30 / M50, the counter adds 1.

3．Program cycle time.

[pay attention to]:

1． Whether the count is cleared after the system is powered on by NO. Parameter setting of the # 3002;

2．Cycle time: When the automatic run starts, start starts. After starting up, clear the zero, and stop the time.

3.2 Position interface is displayed

1. Press the [position] function key;

2. Switch the function software and display the following three interfaces:

(1) Show the absolute coordinates of the parts

Figure 3-8 Coordinate interface

Press the [Operation] soft key on the absolute coordinate interface to perform the following functions:

Coordinate reset: perform [coordinate reset] to clear the impact of the machine lock. This operation is required in edit mode before unlocking the machine lock.

Step: Press the "Edit Mode", switch to the "Absolute Coordinate" interface under the position interface display, and press the [Operation] soft key to perform the coordinate reset function.

Parts rate clearance: to clear the number of processed parts, the user level permission is required.

[pay attention to]:

① The display format of spindle speed S is "actual speed (programming speed) (spindle ratio)", and the display of actual spindle speed requires a position encoder of 1024 / 2500 lines on the spindle.

② Actual feed rate F = programmed F rate multiplier.

③ When the speed is per turn, because the unit is 0.0001 mm / min, the display unit is 0.01 mm / rpm and the programming rate display unit is 0.01 mm / r.

(2) Show the relative position of the relative coordinate system

After boot, as long as the machine tool moves, its movement position can be displayed by the relative position, and can be cleared at any time.

Relative coordinate single axis zero: in the relative coordinate interface, press [operation], press X or Y, Z, then the axis of the key is selected and highlighted. Press [Origin] in the downlink soft key, and the corresponding axis coordinate is reset.

Relative coordinates All axes zero: press the soft key [All axes], then the relative position coordinates of all axes are reset to 0.

Relative coordinate setting: Set the relative coordinates of each axis. For example, after entering X20 and pressing [Settings], the relative coordinate of the X-axis is shown as 20.

Part In: After performing the [Score In] operation, the system displays half of the relative coordinate value of the axis.

Parts rate clearance: to clear the number of processed parts, the user level permission is required.

Figure 3-9 The relative coordinate interface

Figure 3-10 Aaxis reset

(3) Display the comprehensive position

Figure 3-11 Integrated coordinate interface

Figure 3-12, the custom display interface of the integrated coordinate PLC signal

Also displays the current position in the following coordinate system:

Position in the relative coordinate system (the relative coordinate system);

Position in the part coordinate system (absolute coordinates);

Position in the mechanical coordinate system (machine tool coordinates);

Total movement (automatic and MDI mode).

Press the soft key [operation] to set the relative coordinates.

As shown in Figure Figure 3-13:

Relative coordinate single axis zero: in the relative coordinate interface, press [operation], press X or Y, Z, then the axis of the key is selected and highlighted. Press [Origin] in the downlink soft key, and the corresponding axis coordinate is reset.

Relative coordinates All axes zero: press the soft key [All axes], then the relative position coordinates of all axes are reset to 0.

Relative coordinate setting: Set the relative coordinates of each axis. For example, after entering X20 and pressing [Settings], the relative coordinate of the X-axis is shown as 20.

Part In: After performing the [Score In] operation, the system displays half of the relative coordinate value of the axis.

Parts rate clearance: to clear the number of processed parts, the user level permission is required.

Figure 3-13- - -Coordinate System Settings

3.3 Program list is displayed

The following steps can display a list of programs and details:

1. Select the [Edit] method;

2. Press the [program] function key;

3． Press the soft key [list] to display the total number of current stored programs in the system, and the program name, size, comment, and modification time of each program;

4．Within the program list, the current use program appears in blue and the rest in black.

The following steps display system storage capacity and details:

1．Select the [Edit] method;

2． Press the [program] function key;

3．Press the soft key [List] to display the total number of system programs and the program directory.

Figure 3-14, a list of programs

3.4 Instruction value is displayed

1．Press the [program] function key;

2．Select the automatic mode or the MDI mode;

3．Press the corresponding soft keys to display the following two images respectively:

(1) Press the soft key [monitor] to enter the program monitoring interface

Displays the instruction value of the current program segment, the absolute coordinate and movement amount of each axis, the current mode value, coordinate change and F and S changes, etc.

Figure 3-15 Program monitoring interface

(2) Press the soft key [program] to display a page of the program segment in the memory. Can display multiple pre-processing programs, which can be used to check the content of the pre-processing procedures.

Figure 3-16 Display interface for program execution

Figure 3-17, the program monitor PLC signal customization interface

3.5 System alarm display

When the alarm occurs, jump directly to the alarm interface and display the latest alarm information.

In the [information] [alarm log] interface, the alarm details can be found in the system according to the alarm number and alarm information. As shown in Figure Figure 3-18:

Figure 3-18 shows the alarm interface

3.6 The Help interface is displayed

1. Move the cursor in the system parameter interface, press the [Help] function key to help:

(1) Parameter number prompt;

(2) Permissions required to modify this parameter;

(3) The value range value can be edited;

(4) Detailed description of the parameters.

2. Move the cursor of the coordinate system interface to select the artifact coordinate system to be used, and press the [Help] function key to prompt help:

(1) Set the display content of the window;

(2) Detailed help description of the search, measurement, division and input of the workpiece coordinate system.

Other interfaces press the [Help] function key to enter the help main interface, mainly displaying the operation help information, including the display contents and operation functions of each window as shown in Figure 3-19:

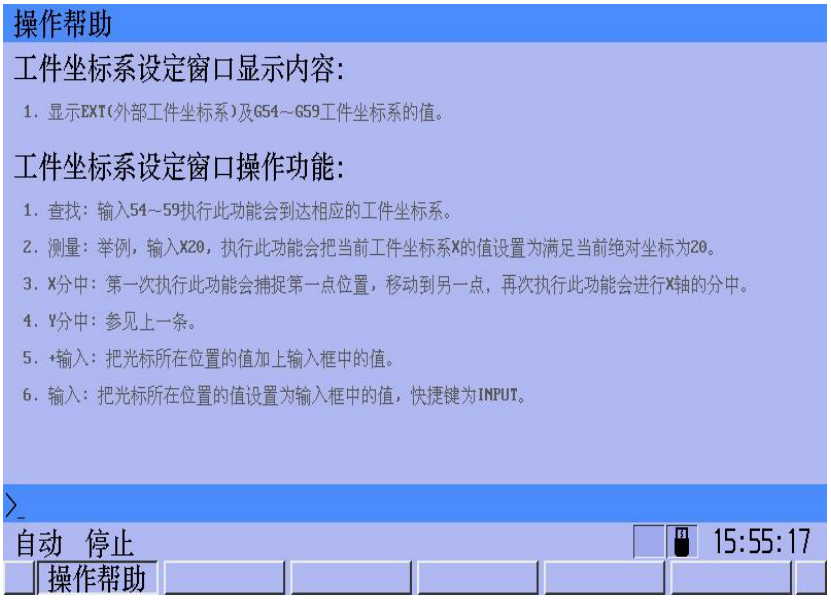


Figure 3-19 The help interface shows it

3.7 Display of the tool compensation amount

3.7.1 Display of the tool compensation amount

Press the function key [knife bias / setting] (OFF / SET) [shape] to enter the offset / shape interface, the content of this interface is shown in Figure 3-20:

Display 1-399 group tool length compensation, tool length wear, tool radius compensation and tool radius wear value.

The Offset / shape interface performs the following functions:

1: Retrieval: retrieve the serial number in the input box

2: C input: input the relative coordinates into the length compensation H where the cursor is located, and the cursor is invalid in the other column

3: Empty: empty the knife fill and wear value of all serial numbers, set as 0

4: Clear: Set the value of the cursor position to 0

5: + Input: Add the value of the cursor location plus the value in the input box

6: Input: Set the value of the cursor position to the value in the input box, and the shortcut key is INPUT

Figure 3-20 Bias / Shape interface

3.7.2 Modification and setting of the tool compensation amount

1．substantive input

(1) Select the shape / wear and correction;

(2) Because the display is divided into multiple pages, you can choose the required page by pressing the page turning button;

(3) Move the cursor to the position of the compensation number to be input;

Scanning method: press the upper and lower cursor keys to move the cursor in line;

Search method: move the cursor directly to the typed position with the following key order: compensation number soft key [No retrieval].

(4) Input the compensation amount (the decimal point can be entered);

(5) After pressing the soft key [input], input the compensation amount, and display it on the display screen.

2．increment load

(1) Move the cursor to the position of the compensation number to be changed;

(2) Type the increment value with the data keys;

(3) Press the soft key [+ input], add the current compensation amount and the typed increment value, and the result is shown as the new compensation amount.

(Example) set compensation 5.678

1.5 for keyboard input

Newly set compensation amount 7.178(=5.678+1.5)

3．C import

(1) Select No.1 knife as the base knife and face the knife on the workpiece surface;

(2) Enter Z0 in the standard system, and determine the Z value of the coordinate system according to [measurement];

(3) No.1 knife does not move, return to the position coordinate interface, Z relative coordinates clear zero;

(4) Change the no. 2 knife again, and input C at the no. 2 knife repair;

(5) The value entered at this time is the compensation value of knife 2 relative to knife 1.

[pay attention to]

The system can modify the tool compensation value only when the state is stopped. After entering the current tool information, the corresponding wear is emptied.

3.8 Display of the coordinate system

Press the function key [knife bias / setting] (OFF / SET) [Coordinate System] to enter the coordinate system setting interface, the content of the interface is shown in Figure 3-21:

Figure 3-21 Coordinate system interface

The following operations can be performed in the work piece coordinate system setting interface:

1. Find: Enter 54-59 to perform this function and quickly locate to the corresponding artifact coordinate system.

2. Measurement: if input X20, after performing the measurement, the system sets the X value of the current work piece coordinate system to meet the value of the current absolute coordinate of 20.

3. X score: the first time to execute this function will capture the first point position, move to another point, and execute this function again will divide the X axis.

4． Y score: Idento.

5．+ Input: Add the value of the cursor location plus the value in the input box.

6．Input: Set the value of the cursor position to the value in the input box and the shortcut to INPUT.

3.9 Display of the setting interface

1．Press the function key [knife offset / setting] (OFF / SET);

2．Press the soft key [setting] [parameter] to display the parameters that can be set;

3． Press the direction key [] to move the cursor to the project to be changed;

4．Enter the parameter value, press the soft key [input], the parameter is modified and displayed.

Settable parameters include:

NO.3202 Line number is automatically added (0: No 1: Yes)

NO.3203 Line number increment value parameter (integer greater than 0)

NO.3999 Language selection (0: Chinese 1: ENGLISH)

NO.3206 Show prompt at system startup (0: No display 1: display)

NO.3207 Same ential character keys (0: no switch 1: switch)

NO.3208 EOB performs insertion function (0: not executed 1: executed)

NO.3209 Reset is clear MDI procedure (0: not clear 1: clear)

NO.3000 Program Cursor Color Selection (0: Light Blue 1: Yellow)

NO.3008 Whether the PLC signal is read in the program restart

NO.4022 Custom macro variable is open (0: not open 1: open)

Figure 3-22 Setting the parameter interface

3.10 password setting interface

Display the current system permission level:

1. Press the function key [bias / setting];

2. Press the soft key [Settings] direction key [] / [BWD] to enter the password setting interface.

The authority level is divided into low level authority, user permission and system authority.

From the lower level permission to the higher level permission, you need to enter the corresponding permission password.

Low level authority: under low level authority, user coordinate system, tool offset, and few user basic parameters.

User rights: under the user level authority, advanced user parameters (running speed parameters, user programming parameters, mechanical accuracy compensation parameters, feed axis / spindle parameters, reference point can be executed / changed and system connection parameters, etc.

System permission: under the advanced system permission, you can execute the change / input of the advanced system parameters (system full parameters, pitch compensation, PLC signal parameter editing).

[Downte]: Reduce the current operation level permission of the system;

[Modify the password]: Enter the corresponding password to change the current system operation permission.

Figure 3-23 Password setting interface

3.11, Time setting interface

The time interface can display the time information of year, month, day, hour, and minute, and week (week).

1. Press the function key [bias / setting];

2. Press the soft key [Settings] direction key [] / [BWD] to enter the time setting interface;

3. Press the direction keys [], [] to select the parameters to set and move the cursor to the project to be changed;

4. You can modify the value by inputting / + input, and press the soft key [operation] [input] to complete the time setting;

5. Year, month and day are edited on the Gregorian calendar time, and hours and minutes are edited in 24 hours.

Figure 3-24, time setting

3.12 Display of the macro variable

3.12.1, the display of the macro variables

1. Press the function key [knife offset / setting].

2. Press the soft key [macro variable] to enter the macro variable setting interface.

3.12.2 Modification and setting of the macro variable

1. Press the function key [knife offset / setting].

2. Press the soft key [macro variable] to enter the macro variable setting interface.

3. Press the page turn key to display the page to set the variable to move the cursor and locate the variable to be changed.

4. After entering the variable number, press the soft key [Search] to quickly locate to the position to modify.

5. Enter the new numerical value with the number key.

6．Press the software [input] or press the shortcut key INPUT to complete the modification.

Figure 3-25 The macro variables are shown

Before modifying the parameters, you need to enter the password to obtain the corresponding operation permission. See the password modification.

3.13, and the parameter display

3.13.1 Parameter display

1．Press the function key, [System];

2．Press the soft key [parameter] to enter the parameter setting interface, as shown in Figure 3-26;

3． Can directly input the letter after the parameter category to switch or through the arrow [], [] interface switch;

4．It can also pass through the NO. Numerical retrieval: Enter the parameter number to be retrieved in the digital edit key area, and press the soft key [Retrieve].

In Figure Figure 3-26, the parameters are shown

3.13.2 Modification of the parameters

Before modifying the parameters, you need to enter the password to obtain the corresponding operation permission. See the password modification.

1．Bit parameters directly press the soft key [ON: 1] / [OFF: 0] to modify;

2． After entering parameter value of other parameters, press soft key [input] or [input] of MDI panel for modify;

3．After the input is completed, make the restart according to the system prompts.

3.14 System matching for quick debugging interface of the machine tool

In the machine tool debugging interface, all the common debugging items of machine tools are sorted out to system debugging, machine tool debugging and absolute value debugging (the absolute value function system is valid), and the application interface of electronic gear ratio calculation is established.

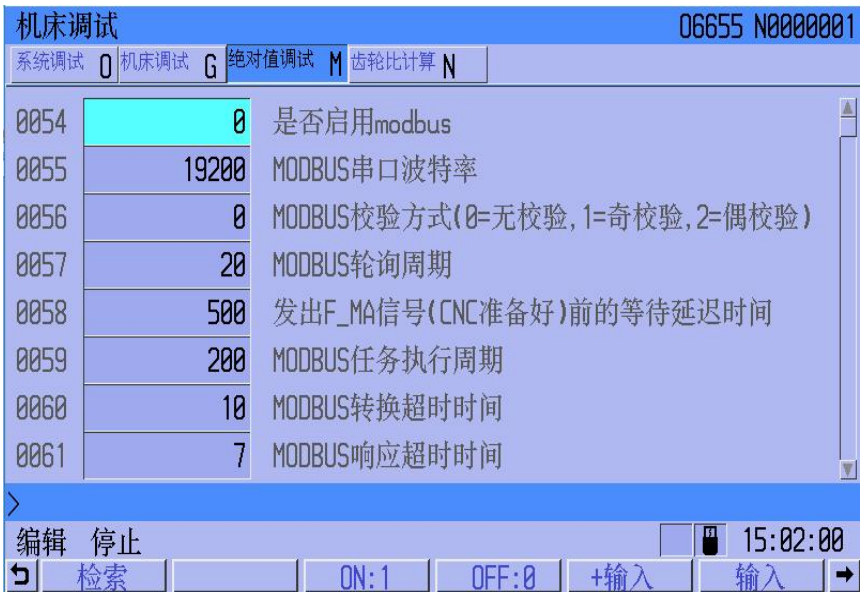
3.14.1 System debugging interface

graph 3-27

3.14.2, machine tool debugging interface

graph 3-28

3.14.3 The debugging interface for absolute value



graph 3-29

3.14.4 Gear ratio calculation interface

graph 3-30

3.15 Display of the system pitch compensation

3.15.1 Display of the pitch compensation

1. In the stop state, press the function key [System] to enter the system interface.

2. Press the soft key [pitch compensation] to enter the pitch compensation interface.

3. Use directional keys [], [], [], [], [] and page turning keys to move the cursor; also enter the compensation number and soft key [operation] [retrieval] for quick positioning.

4. Set the pitch compensation interface of each motor by [BWD] or [FWD].

Figure 3-31 The pitch compensation interface

3.15.2 Setting of pitch compensation

1. Follow the steps to locate the corresponding modification items;

2. Type the set value;

3. Press the soft key [input] to achieve data modification and display;

4. Press the soft key to [+ input] to increase the setting based on the original data.

1． Each feed shaft fill interval by NO.1126-1131 parameter (motor pitch compensation point interval) set.

2． The origin of each feed axis, the compensation number of the farthest end of the positive and negative direction is set by NO. Parameter setting No.1135-1158.

3．Before modifying the parameters, you need to enter the password to obtain the corresponding operation permission. See the password modification.

3.16 Ladder diagram for display

In the ladder diagram display interface, you can press the flip button to view the current PLC signal status.

1．Press the function key, [System];

2． Press the soft key [ladder diagram] to enter the ladder diagram display interface;

3．Enter the PLC, and press the [Find] soft key to retrieve the address. If you enter R7.0, continuously press [Find] to display all R7.0 addresses.

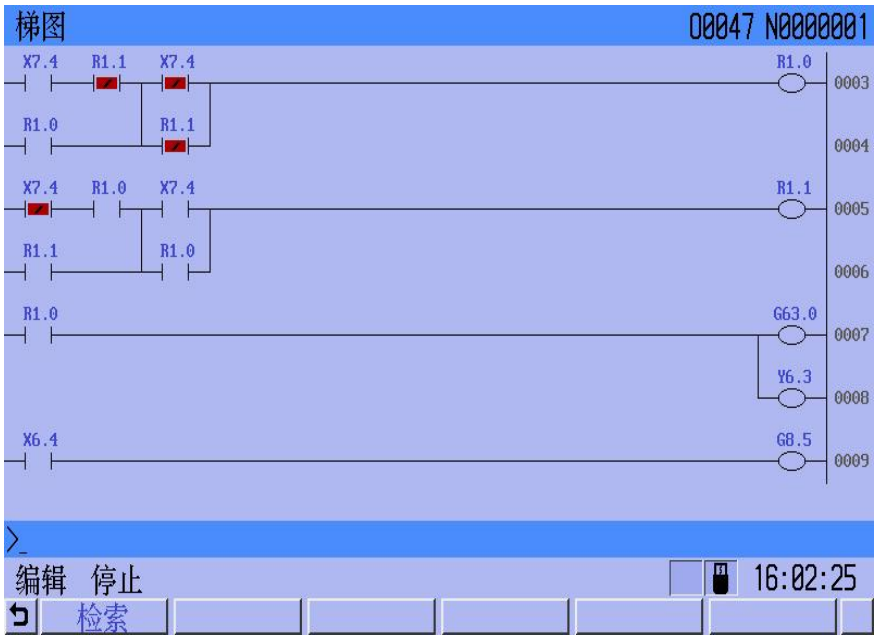


Figure 3-32, the ladder diagram interface display

3.17 The PLC signal is shown

The state of DI / DO signal between CNC and machine tool, the signal state transmitted between CNC and PLC, the internal data of PLC and the internal state of CNC are all displayed in the PLC signal display interface.

1. Press the function key, [System].
2. Press the soft key [PLC signal] to enter the PLC signal display interface.
3. Press [BWD] / [FWD] to enter different signal display interfaces.

There are five display methods in the page: signal [X signal] to PLC, signal from PLC to the machine [Y signal], CNC to the system [F signal], signal from PLC to CNC [G signal], [R signal], [A signal], [K signal], [D signal], [T signal], [C signal].

4. Before modifying the PLC signal parameters, it is necessary to enter the password to obtain the corresponding operation permission. See the modification of the password.

Figure Figure 3-33 The PLC interface is shown

3.18 PLC custom display function of the signal

In the position-integrated interface and the system monitoring interface, press the left and right arrows of the MDI panel to switch the PLC signal parameters in the system ratio.

graph 3-34

graph 3-35

The parameters displayed in the PLC customization interface can be customized by the PLC editing software component annotation function. The number of custom parameters is limited to 10, as shown in the figure below. Before defining the custom annotation text content, add @0~@9 position characters. The system will put the text comments to be added to the corresponding position according to the position characters.

	地址	数量	注释
1	X5.1		主轴报警
2	A0.0		主轴报警
3	X5.6		气压报警
4	A0.1		气压报警
5	X3.5		润滑报警
6	A0.2		润滑报警
7	X5.3		库电机过载
8	A0.3		库电机过载

graph 3-36

[pay attention to]:

(1) The customized parameters in the PLC parameter customization interface are limited to only be bit parameters, and the parameters of non-bit parameters cannot be displayed in the interface;

(2) The number of content characters of custom parameters is limited to 8 Chinese characters or 16 full English characters.

3.19 System information display

1．Press the function key [System] to enter the system interface;

2． Press the soft key [System] to enter the system version information display interface;

3．Displays the system version, system creation time, PLC version creation time, IP address and other information.

As shown in the following figure, the following operations can be performed in the system information display interface:

1．IP address of the system:

(1) [System] function key [System] soft key [operation];

(2) Enter the IP address;

(3) Change according to [modify IP].

2． System restart

After other operations: the system information display interface presses [Operation] [System restart] [Confirm Y] for the system restart.

Figure 3-37 The system display interface

3.20, graphical function display

Graphical display function depicts the tool movement track during automatic or manual operation. The tool path simulated on the display screen can check whether the track and shape of the machining are correct.

3.20.1 Description of the tool path (graphic simulation)

Press the [graphics] function key to enter the graphical interface. As shown in Figure Figure 3-38:

1． You can monitor the machining shape of the running program.

2． Displays the current position of the tool in the artifact coordinate system.

3．F, S, and T values for the depicted program.

Figure 3-38 The graphical display interface

3.20.2, the setting of the drawing parameters

Press the [graphics parameters] soft key to enter the graphical parameter setting interface, and set the tool track description required.

In the drawing parameter interface, the selection of drawing coordinate plane, drawing range, drawing display scale and the drawing center position displayed in the interface are mainly set.

Graphical parameter NO.3118 erase the current drawing when the program starts. As shown in Figure Figure 3-39:

Figure 3-39 Figure graph parameter setting interface

4 Manual operation

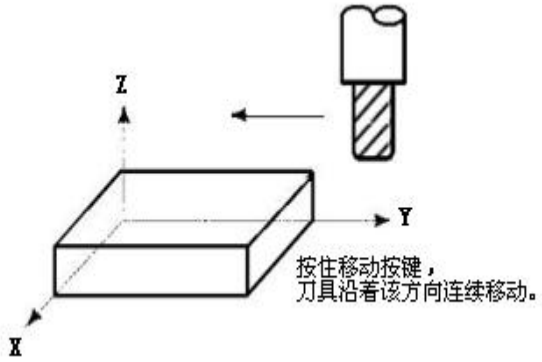
Press the function key [manual] key to enter the manual operation mode, mainly including feeding shaft control, spindle control and machine tool panel control, etc.

4.1 Control

In manual operation mode, the triaxes can run at manual feed speed or manual fast movement speed.

4.1.1 Manual feed

In manual mode, first press the machine operation panel [X], [Y] and [Z] to select the shaft to move, and then press [+] and [-] to control the corresponding feed axis to start moving. The movement speed can be changed by adjusting the feed rate, and the shaft movement stops when the button is released.



graph 3-40

[pay attention to]

Manual feeding speed of each axis by NO.1063-1068 parameter (motor feed speed) set.

4.1.2 Manual and fast movement

Press the fast feed key [fast], the indicator light is on, enter the manual fast movement state, and then press the axis feed direction key, each axis at a fast running speed.

4.1.3 Manual feed and manual fast movement speed multiplier selection

When manual feed, you can press the operation function key [feed increase] or [feed decrease]. The system defaults to 16 levels, 0% -150%.

When moving manually fast, you can press the operation function keys [F0], [25%], [50%] and [100%] to select the rate of manual fast movement speed. The default fast rate is F0,25%, 50% and 100%. The F0 speed is given by the NO.1081-1086 parameter (motor fast shift F0 speed) set.

[pay attention to]

1． Manual feed rate and fast feed rate can be viewed in the location interface.

2．When choosing to move at 100% speed, please pay attention to any foreign body on the work table to prevent impact and falling.

3．Fast multiplier selection is effective for the following movement speeds:

(1) G00 fast feed;

(2) Fast feed in the fixed cycle;

(3) Fast advance at G28;

(4) Manual fast feed.

Operation steps for the feed axis:

1．Select the feed key switch in the switch in the mode;

2．Press the switch intended to move the axis and direction from the feed axial selection switch;

3．During holding the switch, the tool to parameters (NO.1063-1068). When the switch is released, the tool stops moving;

4． Manual input, the speed can be adjusted by manual input rate;

5． When the fast move switch is pressed during the feed axial selection switch, the tool is moved quickly during the fast move switch. During the fast movement, the fast moving rate controlled by the fast moving rate switch is effective.

The steps mentioned above are just one example. For the actual operation, please refer to the instructions provided by the machine tool manufacturer.

4.2 Spindle control

Given the S speed in MDI mode, in manual, hand wheel and MDI mode, press the operation function key [positive turn] and [reverse turn], and the spindle rotates forward and backward.

With manual, hand wheel and MDI, press the operation function key [spindle stop] to stop the spindle rotation.

Spindle quasi-stop: the manual control spindle stop on the fixed rotation position, convenient to return the knife and other operations.

[pay attention to]

1. Maximum spindle speed by NO.0014 (spindle speed clamp) parameter setting.

2. Resreset and emergency stop buttons can stop the main shaft.

3. When the spindle is turning forward, it is forbidden to press the spindle reversal key directly. Stop the spindle first before reversing the spindle.

4.3 Other manual operations

4.3.1 Coolant / blow cooling control

Press the operation function key [Cooling] to switch between the coolant open and close. The boot is turned off by default.

Press the operation function key [Blow] to switch between open and close. The boot is turned off by default.

For practical operation, refer to the instructions provided by the machine tool manufacturer.

4.3.2, working light control

Press the operation function key [working light] to switch between the working light on and off. The boot is turned on by default.

For practical operation, refer to the instructions provided by the machine tool manufacturer.

4.4 Operation knife operation

In the numerical control system, find the position of the axis coordinates of the workpiece on the machine tool workbench by tool trial cutting, edge finder (medium rod), base tool, non-base tool, etc.

Determine the G54 coordinate plane, press the function key OFF / SET knife bias / setting to enter the setting interface.

1．Try to cut the points

If the requirements are not high, or the workpiece is wool blank, and the shape can be milled, in order to facilitate operation, the tool used can be used directly touched, so as to determine the working origin, the steps are as follows:

(1) The milling cutter to be used on the spindle, and rotate the spindle at medium speed;

(2) Manually move the milling cutter along the X direction near the measured side of the workpiece, until the milling cutter just cuts the workpiece material;

(3) Keep X and Y unchanged, raise the Z axis along the + Z direction, press the X score key, the screen shows "the first position coordinate of the X axis has been obtained, please move to the second position";

(4) Move the X-axis to the other side of the workpiece, and cut the workpiece material just right with the tool;

(5) Raise the main axis along the + Z direction, and press the X point middle key to display the middle screen. At this time, the data displayed by the X axis in the coordinate system is the X zero;

(6) Use the same method to complete the setting of the coordinate system.

2．The score is the best

The principle of centrifugal force is adopted in the middle rod. In the middle, the spindle speed can only be set between 350~600r / min, and must not exceed 600 r/min, and generally should be about 500r / min. The steps are as follows:

(1) The middle rod is installed on the main shaft, and the initial test end is at the bottom;

(2) Set the spindle speed at about 350~600r / min;

(3) Manually close the middle rod to the side of the workpiece along the X axis, the swing range of the middle rod from large to small to overlap, adjust the hand wheel ratio to 0.01mm, and continue to close to the workpiece and the middle rod just separate;

(4) Raise the main axis along the + Z direction, move in X to the other side of the workpiece, and touch the other side of the workpiece with the same method;

(5) Raise the main axis along the + Z direction, and divide it according to X;

(6) The Y-axis is measured in the same method.

3．Benchmark knife to knife

(1) Set the base and the knife value to 0;

(2) Use the base knife on the workpiece surface to face the knife;

(3) In the selected coordinate system, enter Z0 and press [Measurement];

(4) Return to the relative coordinate interface, and clear up the relative coordinate of the Z-axis;

(5) Replace the second tool and face the knife on the surface of the workpiece;

(6) Press [C input] at the supplementary value of the second tool, which is the compensation value of the second knife relative to the reference knife.

4． Non-benchmark knife to knife

Non-reference knife knife is to operate each knife, and input the coordinate value of the Z-axis machine into the compensation value of the corresponding tool.

5 Hand wheel (hand pulse) and incremental operation

Handwheel mode / Incremental mode is selected by the parameter NO.0006 (hand wheel / incremental mode selection), 1 is hand wheel mode, 0 is incremental mode.

The minimum moving unit in handwheel / increment mode is set by the parameter (minimum moving unit (mm) in handwheel / increment mode, and the system default is 0.001mm.

5.1 Hand wheel (hand pulse) feed

In the hand wheel mode, the shaft can be moved bit by bit by rotating the hand pulse generator on the handheld / operating panel of the machine tool. Using the handwheel shaft selector switch, select the shaft that will be moved.

5.1.1 Selection of movement quantity

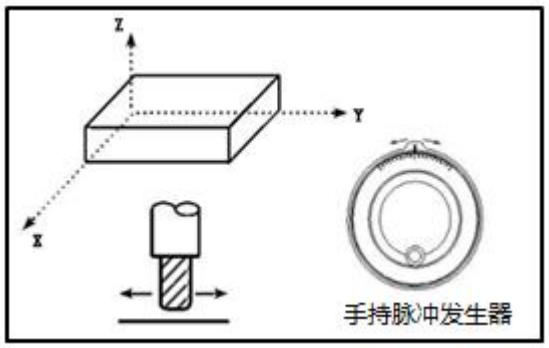
parameter NO.0006 (hand wheel / increment mode selection) is set to 1; press the operation function key [increment / hand wheel] to select the hand wheel mode.

Select [1] [10] [100] on the hand wheel to change the hand wheel / increment feed rate, and the transformation order is 1,10,100. Display the current hand wheel / increment rate in the [position] interface.

5.1.2 Selection of moving axis and direction

In handwheel mode, select the handwheel to move the shaft.

Control by hand pulse, see the machine tool manufacturer or handwheel (hand pulse) manual. Generally speaking, the hand pulse is clockwise positive and the hand pulse is counterclockwise negative. As shown in Figure Figure 3-41:



graph 3-41

Operation steps for hand wheel feeding:

1．First, choose the hand wheel operation mode;

2．Press the handwheel to select the shaft that will be moved;

3．Select the multiplier rate of movement by pressing the starting wheel to feed the distance multiplier switch;

Each moment of the degree of movement is the minimum set unit;

4．Rotate the hand wheel to move the tool along the selected axis. The tool moves 100 scales for each turn.

[pay attention to]

The speed of the rotating hand pulse should not exceed 5 r/s. If it exceeds 5 r/s, the scale and movement may not appear.

5.2 Incremental feed-in

System parameter 0006 (hand wheel / increment mode selection) is set to 0; press the operation function key [increment / hand wheel] to select the increment mode.

Select [1], [10] and [100] on the hand wheel / increment feed rate, and the transformation order is 1,10,100. Display the current handwheel / increment rate in the [Location] interface or the [System] interface.

In increment mode, the positive / negative rotation on the hand wheel, and the positive / negative direction of the X axis [hand wheel / increment minimum movement distance in the hand wheel / increment rate].

[pay attention to]

In the case of large ratio, try to avoid too dense keys to avoid the movement amount inconsistent with the theoretical value.

5.3 Trial cutting of the hand wheel

After the editing, the system can run the correctness of the test program at low speed.

Press the hand wheel test cutting function key set by the system, switch the operation mode to the automatic mode, and rotate the hand wheel for the trial processing of the program. In this case, pressing the cycle start button will not move the machine tool. The machine tool will follow the parameter NO.0576 The set feed rate runs.

6 Automatic operation

6.1 Operation mode

6.1.1 Automatic operation / memory operation

1．First, import the program into the system memory;

2．Select the program to run:

Scanning method: press the top, bottom, left, right, page turn keys to move the cursor to select the program;

Search method: input the program number, press the soft key [No retrieval];

3．Press the soft key [Run] to load the memory program;

4．Operation mode selection [automatic] mode;

5． Press the cycle start button and the indicator light is on to start the procedure;

6．In the automatic operation, if the feed keeping switch on the machine operation panel is pressed, the automatic operation will be temporarily stopped. At this point, if the cycle start switch is pressed again, the automatic operation will start again. In addition, when the reset key on the operation panel is pressed, the automatic operation ends and the system enters the reset state.

[pay attention to]

The above steps 2,3 and 4 can also type the program number (such as O1234) in the [program] interface (PROG), and press the MDI function key [insert] (INSERT) or [] (down direction key) to load the program corresponding to the input program number.

6.1.2 MDI runs

Enter instructions for a segment from the MDI panel and you can execute it.

Example: G00 X10.5 Y200.5;

1．Press the operation function key [MDI], and the operation mode is selected in the MDI mode (MDI mode);

2．Press the MDI function key [PROG] key;

3．Enter "G00 X10.5";

4．Press [Inset] key, enter "G00 X10.5" and be displayed;

5． Enter Y200.5; press [Inset], Y200.5 is entered and displayed;

6．Press the [Cycle Start] key to run the input MDI program.

[pay attention to]

1．If you find the input error before inserting the key, press CAN to cancel typing one by one, and then enter X and the correct value again;

2．If you press the [Insert] key for an error, enter the correct value again;

3．After entering the need to modify, you can select the modified character, [delete] key to delete, and then press the upper step to type, also select the character to be modified, press the [replace] key to directly replace the modified character;

4．After selecting MDI operation mode, it can only input and edit in the program interface, and the monitoring interface is not editable;

5． After the MDI program is executed, the system will automatically empty the program buffer;

6．Also available through the NO.3004 (system status on reset) changes to no clear, then you can cancel one through the CAN key.

Figure 3-42 The MDI program interface

6.1.3 DNC runs

When large programs need to run, DNC operation mode is usually selected. The operation steps are performed as follows:

1. Press the operation panel function key [DNC] to enter the DNC mode;

2. Select the [program] interface;

3. Set the relevant serial port communication parameters;

4. Connect the back end of the system [RS232 interface] with the PC serial port (see the equipment connection part for the connection diagram and line order);

5. Use the transmission software to transfer programs;

6. After the program appears in the system [program] interface, press the [cycle start] button to start executing the program.

pour:

Empty operation cannot be used in DNC mode, but it does not affect the normal program processing.

The system itself performance limitations, the DNC transmission is too fast, will cause the system alarm transmission program error. At this time, open the empty operation, the machine running speed is too fast, the slow system processing will lead to transmission errors.

6.2 Start of automatic operation

1．Select the editing method;

2．Select a procedure, and the operation step is:

System memory: MDI function key [program] soft key [list] soft key [operation] find program soft key [run];

The upper right of the display shows the program number to be run: OXXXX (system memory).

3． Select the [Automatic] operation mode;

4．Press the [cycle start] button on the operation panel.

Fig. 3-43 System memory mode

Figure 3-44 Moving disk (U disk) mode

6.3 Execution of automatic operation

After starting the automatic operation, the program is executed as follows:

1. Read a program segment instruction from the specified program;

2. Decoding the read program segment instructions and becoming the executable data;

3. Send the machine code to start the machine to execute this segment;

4. Read the next program segment instructions;

5. Decode the next segment instruction into executable data (program preread).

[pay attention to]

1. After the execution of the previous program segment, the system can start the execution of the next program segment immediately due to the buffer register, and the cursor moves to the program segment to be executed;

2. Then repeat 4 and 5, and perform the automatic operation until the end of the program.

6.4 Stop of automatic operation

There are two ways to stop the automatic runs. One is to use the program to enter the stop command at the place to stop in advance, and the other is to press the feed to hold button on the operation panel to make it stop.

6.4.1 Procedure suspension (M00)

After the program segment containing M00 is executed, the automatic operation is stopped, and the same as the single program segment stop, and all the modal information is saved. Press the cycle start key to start automatically again.

6.4.2 Select Pause (M01)

As with M00, stop operation after execution of the program segment specified by M01. However, only the Select Stop button on the machine operation panel is on.

6.4.3 End of Procedure (M02 / M30)

1．Represents the end of the master program;

2．Stop the automatic operation, the spindle, feeding, the coolant of the machine tool all stop, into a reset state;

3． The M02 ends and stops at the end of the M02 program;

4． The M30 ends and returns to the starting point of the program.

Note: M02 / M30 is written in the last segment of the main program.

6.4.4 Feed maintenance

In the automatic operation, press the function key [feed] key on the operation panel to temporarily stop the automatic operation. After pressing the feed hold button, the machine has the following status:

1．When the machine tool is moving, the feed feed deceleration stops;

2． In the execution of the pause, the rest to pause;

3． Ststopped after performing the actions of M, S and T.

After pressing the automatic loop start key, the program continues.

6.4.5 Reset position

Use the [reset key] on the MDI to end the automatic operation and become a reset state. If the reset is performed during the motion, it stops after the mechanical deceleration.

In addition, when running the program in the [automatic], [MDI] and [DNC] modes, the machine tool can also be stopped by switching to other modes (stop the machine tool after running the current program segment).

6.5 Coolant / blow cooling control in automatic operation

1．In automatic mode, the operation panel function key [cooling] / [blow gas] key function is valid. This key is pressed to open and pressed again to close;

2． Add M08 code to open the coolant, M07 code to start the blowing cooling;

3．Automatic operation can manually close the coolant or blow cooling, or specify the M09 code to turn off the coolant or blow cooling in the program.

6.6 Machine tool and auxiliary function lock the operation

1． In [automatic], [MDI] and [DNC] operation modes, press the function button [machine tool lock] of the primary operation panel, and the machine will make [feed shaft] lock and [auxiliary lock] switch between lock and unlock. The boot default state is all unlocked.

2． In [Edit], [Manual], [hand wheel], [increment] and [return reference] modes, press the function button of the primary operation panel [machine lock], the machine will make [feed shaft] lock switch between lock and unlock.

3． When the feed axis is locked, the [feed lock] display light of the system operation panel is on. At this time, each axis of the machine does not move, but the display of position coordinates is the same as when the machine moves, and M, S and T can be executed.

4．[Auxiliary lock] When locked, the [auxiliary lock] display light of the system operation panel is on, and the machine tool M, S, T code instructions are not executed.

5．When the feed shaft and the auxiliary lock function are locked simultaneously, the feed lock is used together with the auxiliary lock for program verification.

1． After pressing the [machine tool lock] and running the program, the machine tool position is inconsistent with the coordinate position, and the machine tool needs to return to zero after operation.

2．M00, M01, M30, M98, M99.

3．Users can switch between [feed lock] and [auxiliary lock] according to the two operation conditions of 1 and 2.

6.7 Automatic operation of the medium feed and fast feed speed adjustment

In the automatic operation mode, the feed speed multiplier can be adjusted by rotating the feed multiplier dial switch to adjust the feed speed.

You can continuously press the operation panel function keys [F0], [25%], [50%], [100%] keys to switch the fast feed rate, so as to adjust the fast feed speed.

[pay attention to]

Universal use between the feed speed multiplier switch and the manual continuous feed speed multiplier switch.

6.8 Spindle speed adjustment during automatic operation

In the automatic operation mode, the spindle speed ratio can be adjusted by rotating the spindle ratio shift switch to adjust the spindle speed.

[pay attention to]

Feed spindle speed rate switch is common with manual spindle speed rate switch.

6.9, with an empty run

Before the program processing, you can use [empty run] to test the program, generally with [auxiliary lock], [feed lock] use.

Press the operation panel function key [empty operation] key to switch the empty operation mode on / off state. When the empty operation function is on, the key light is on.

[pay attention to]

Empty running speed is given by NO.0504 parameter (empty running speed) to set.

6.10, with a single-segment run

To detect the single segment of the program, select [single segment].

In [automatic], [MDI] and [DNC], press the operation function key [single segment] to make the machine switch between opening and closing the single section. When the single section function is opened, the key light is on.

When a single segment is running, the system is suspended after each program segment. Press the [cycle start] key to continue the next segment until the program is finished.

[pay attention to]

1． In G28, single-program segment stops are performed even at the middle point.

2．The program segments of M98, M99 and G65 cannot be stopped by single program segments except those of M98, M99 except N, O and P.

6.11 Transformation of various operation modes

In the [automatic], [MDI], [MNC], [DNC], [edit], the system stops the program after running the current program segment.

When the program is directly converted to [manual wheel], [MDI], [manual wheel] under [automatic], [MDI] and [DNC], the program stops running after slowing down.

[pay attention to]

In the [automatic], [MDI] and [DNC] modes, programs are directly converted to [hand wheel], [manual] and [return reference], which is incorrect operation, and may lead to damaged tool / machining artifacts or loss of machining program progress.

7 Back to zero operation

7.1 Mechanical zero point

Machine tool coordinate system is the reference coordinate system for numerical control system, which is the inherent coordinate system of machine tool. The origin of the machine tool coordinate system is called the mechanical zero point or the machine tool zero point, also known as the reference point, and is usually installed at the maximum stroke in the positive direction of the feed axis. After the machine tool is designed, manufactured and adjusted, the mechanical zero point is determined, which is a fixed point. Digital control device does not know the mechanical zero when the power, usually automatic or manual back to the mechanical zero, that is, the return parameter operation.

The system uses the stroke switch to perform the return reference, that is, when the system is operation, the machine tool feed shaft moves in the negative direction A-B-C point (related to the machine tool and system parameters). When the block on the feed shaft is pressed, the C-D-E point during the stroke switch, and the system moves E-F at low speed (parameter setting). When the travel switch is out of the block, the return reference axis immediately stops moving and the return reference G point (this is when the limit switch is set as the return reference, if the index signal is set as the return reference, it does not stop until the index signal is received). As shown in Figure Figure 3-45:

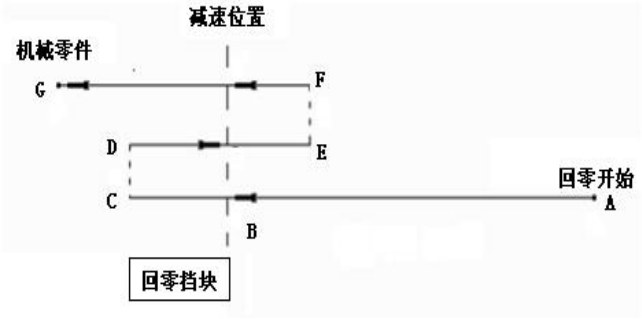


Figure 3-45 Schematic diagram of the reference reference point process

[pay attention to]

1． The effective width of the block should ensure the completion of the deceleration, usually not less than 25 mm.

2． For the absolute value coding motor, you can also use the motor "positioning" return reference mode, which requires NO.1216-1221 parameter (change the actual parameter content) setting.

3．The machine tool coordinate value after returning the reference point is determined by NO. Set 1252-1257 parameter (absolute coordinate value of motor reference point).

7.2 Return participation process

The following are the steps to using the travel switch:

1．Press the operation panel function key [reference point] key to enter the reference mode;

2．According to the running direction of the machine tool, select the button of the moving axis (X, Y, Z, 4th, 5th, 6th), click the direction (+ -) button, and press it once;

3．The machine tool moves along the direction of the selection axis, and the speed of the feed axis is set as the fast shift speed (fast move F0 fast shift rate) before touching the stroke switch;

4．The system supports multi-axis simultaneous return parameters, that is, in the process of one axis return parameters, other axes are operated without interference;

5．After returning to the reference point, the return reference point indicator light on the operation panel is on;

[pay attention to]

During the point movement, it is necessary to adjust the spindle speed in advance and adjust the spindle speed under the MDI state. It is recommended that the speed should not be too high.

7.3 Return the reference with program instructions

The system supports the program instruction G28 to zero, and the travel block is equivalent to the zero return of manual machinery.

Example return program: G91 G28 Z0

[pay attention to]

1． This method needs to use the travel block, if your CNC machine tool is not installed with mechanical zero, please do not use the mechanical back to zero operation.

2．At the end of the return machine zero, the indicator light of the corresponding shaft is on.

8 Program storage and editing

8.1 Store the program in the system memory

8.1.1 Type it in with the MDI keyboard

1. The mode selection is the [edit] mode;

2. Press the [Procedure] key;

3. keyed address O;

4. Type the program number;

5. Type the EOB key (but none).

Through this operation, save the program number, and then insert each word in the program with the insert key.

8.1.2 Use the USB interface for input

Figure 3-46 U disk display interface

1. Mount U disk: in [edit] mode, select [program] soft key [U disk];

2. Select the NC program that you want to enter into the memory;

3. Enter the display name of the program in the system, namely the program number (such as O1234);

4．Press the soft key [Import] key to input the NC program in the U disk into the memory.

8.2 Search of program numbers

When the memory is stored in multiple programs, the required program can be called up by the retrieval method, edited or executed, and this operation is called program retrieval.

8.2.1 Search method

1．Select the Method [Edit];

2．Press the [List] soft key to display the program list screen;

Figure 3-47 List of programs display interface

3．Type the address key O, type the program number to retrieve;

4． Press the soft key [search] or key, key to search;

5．At the end of the retrieval, the retrieved program is selected and highlighted.

The program number in the upper right corner only displays the selected program number after pressing the soft key [Read].

8.2.2 Scanning method

1．Select the Method [Edit];

2．Press the soft key [List] to display the list of programs;

3． Select key or page key;

4．When editing the mode, continuously press the upper and lower direction keys to display the stored programs one by one.

8.3 Delete of the programs

8.3.1 Remove the program in the system memory

1．Select the [Edit] method;

2．Press the soft key [List] to display the list of programs;

3． Press [Operation] to display the subordinate menu;

4．Retrieve or select the program to delete, press the soft key [Delete].

[pay attention to]

This operation is irreversible. Please backup before deleting the program to prevent loss of important data.

8.3.2 Remove the moving disk

1．Select the [Edit] method;

2．Press the soft key [U disk];

3． Press [Operation] to display the subordinate menu;

4．Retrieve or select the deleted program, press the soft key [Delete] to delete the selected program.

[pay attention to]:

This operation can also delete other files in the mobile disk, please operate carefully.

8.4 Output the program

The system supports exporting the programs in the system memory to the U disk by:

1．Select the [Edit] method;

2．Press the soft key [List] to display the list of programs;

3．Press [Operation] to display the subordinate menu;

4． Retrieve or select the program to be output;

5．Press the soft key [Import] to output the program to the U disk.

8.5 Copy and paste of the program segment

Copy and paste the program segments in the system memory to improve the programming efficiency.

1．Select the [Edit] method;

2．Press [Operation] to display the subordinate menu;

3．The cursor is placed at the beginning or end of the segment to copy select the soft key [select];

4． Move the direction key to select the segment to copy, press the soft key [Copy] to copy the selected segment (highlight);

5．Move the cursor to the paste position, and press [Paste] to complete the copy and paste.

Insertion, modification, and deletion of 8.6 words

For the programs in the system storage, the user can edit them:

1． Select [Edit] method.

2． Press the soft key [List] to display the list of programs.

3． Retrieves the program to edit.

4． Press the soft key [Read] to load the selected program.

5．You can directly type the program number to edit, press the upper direction key [] or the lower direction key [] to select the direct loader.

6．There are two ways to retrieve the word to be edited (see Note 2):

(1) Using the scanning method

(2) Use the method of retrieval words

7．Make word modification, insert, delete and other editing operations.

[pay attention to]:

The concept and editing unit of a word: The word is composed of the address and the data following it. For the user macro program, the concept of the word is completely gone, generally known as the "editing unit". In a scan, the cursor appears at the beginning of the Edit Units. When inserting, the insert is after the Edit Unit. Definition of the editing unit:

(1) The content prior to moving from the current address to the next address. For example: G65 H01 P # 103 Q105; there are 4 editing units.

(2) The so-called address is a letter; (EOB) is a single word.

By this definition, the word is also an editorial unit. In the following instructions on editing, the so-called word, correctly should say "editing unit".

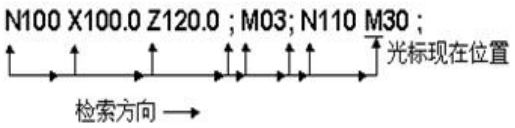
The cursor is always at the bottom end of an editing unit, and the operation of the editing is also carried out on the editing unit indicated by the cursor. In automatic mode, the program is also executed from the editing unit indicated by the cursor. Moving the cursor to the position to edit or the position to execute is called retrieval.

8.6.1 word search

1. Using the method of scanning

Scan word by word.

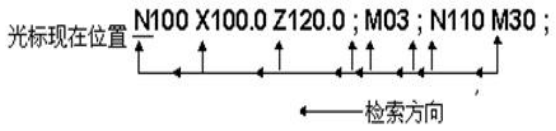
When pressing the cursor key



graph 3-48

At this point, on the screen, the cursor moves in the right direction, word by word. That is, the selected word is selected and highlighted.

(1) When pressing the cursor key



graph 3-49

At this point, on the screen, the cursor moves in the opposite direction, word by word. That is, the selected word is selected and highlighted.

(2) Press the cursor or time, the cursor moves by line, that is, select the word at the beginning of the next line or the last line respectively.

(3) If the cursor is pressed continuously, the cursor will be moved continuously and quickly.

(4) Press the flip page key, turn the page, and move the cursor to the word at the beginning of the next page.

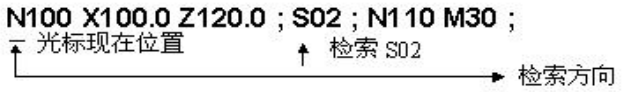
(5) Press the page turning button, the screen to the previous page, the cursor to the word at the beginning.

(6) Continuously press the turn page or turn the page, automatically quickly turn the page.

(7) Press the [head] and [end of the line] to jump directly to the first or last of the current edit line.

2. Method of retrieving words

Starting with the cursor present position, retrieve the specified word in the forward or opposite direction.



graph 3-50

(1) Enter the address S with the key

(2) Enter '0' and '2'

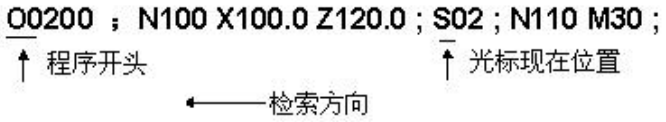
[pay attention to]

When searching S02, if only S2 cannot be retrieved, you must enter S02.

1． Press the soft key [search Up] or [search below] to start the search.

2． If the retrieval is complete, the cursor is displayed at S02.

3．Return to the beginning of the program.



graph 3-51

(1) Method 1: Retrieve the program number, and the cursor can be moved to the beginning of the program.

(2) Method 2: Reload.

① Place it in [Edit] (EDIT) mode;

② Press the [program] key to display the program screen;

③ Press the soft key [operation] [REWIND];

④ The cursor moves to the beginning of the program.

Insertion of the 8.6.2 words

1．In the [Edit] mode;

2．Retrieve or scan to the previous word to insert (see 8.6.1 for the method);

3． MDI keyboard to type the words to be inserted;

4．Press [Insert] to insert the word.

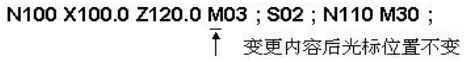
8.6.3 Change of the word

1．In the [Edit] mode;

2．Retrieve or scan to the word to be changed (see 8.6.1);

3． MDI keyboard type the word to be changed;

4．Press the [Replace] key, the original word is deleted, and the modified word is inserted in the original position to complete the change.



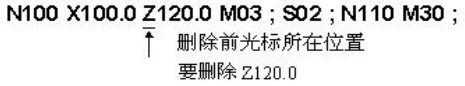
graph 3-52

8.6.4 word deletion

1．In the [Edit] mode;

2． Retrieve or scan to the word to be deleted (see 8.6.1);

3．Press [Delete] key, then the word where the current cursor is deleted, and the word after the cursor is deleted.



graph 3-53

9 U disk operation

Through the USB interface facing the front, the system can read and write the U disk, which can output the program and parameters to the U disk by the system, or can input the program, parameters and upgrade the system from the U disk.

10 System upgrade

After the system power up, long press the [reset] key to enter the system upgrade interface. When the system detects the U disk, the system automatically detects and lists the files in the U disk. As shown in Figures 3 – 54.

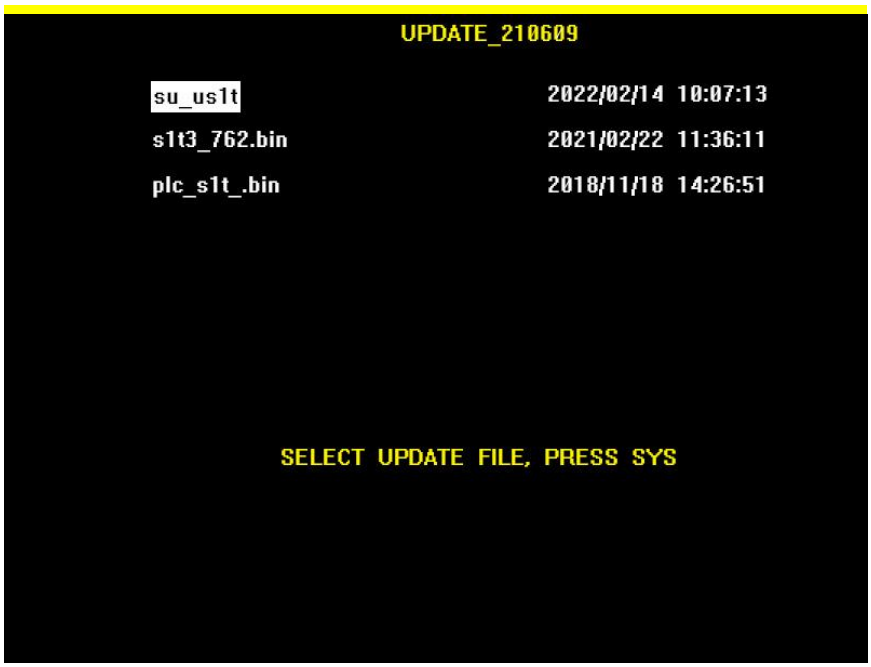
The system files in this interface include system version file, system PLC file and system upgrade package.

System version filename: s 1 * * _ * * *.bin

System PLC file name: plc _ s 1 * _.bin

System file package name: s u_us1t

Select the files to be upgraded or restored through the system, and press the soft key [S YSTEM System] to upgrade the progress bar process. Note that the system will not be automatically restarted after the system PLC file upgrade, and the system file and system file package will automatically restart the system boot after the upgrade.



graph 3-54

[pay attention to]

1. Do not cut the power off or unplug the U disk during the upgrade process.

2. It is strongly recommended that users back up the old version first when upgrading the new version.

11 Parameter backup / recovery

graph 3-55

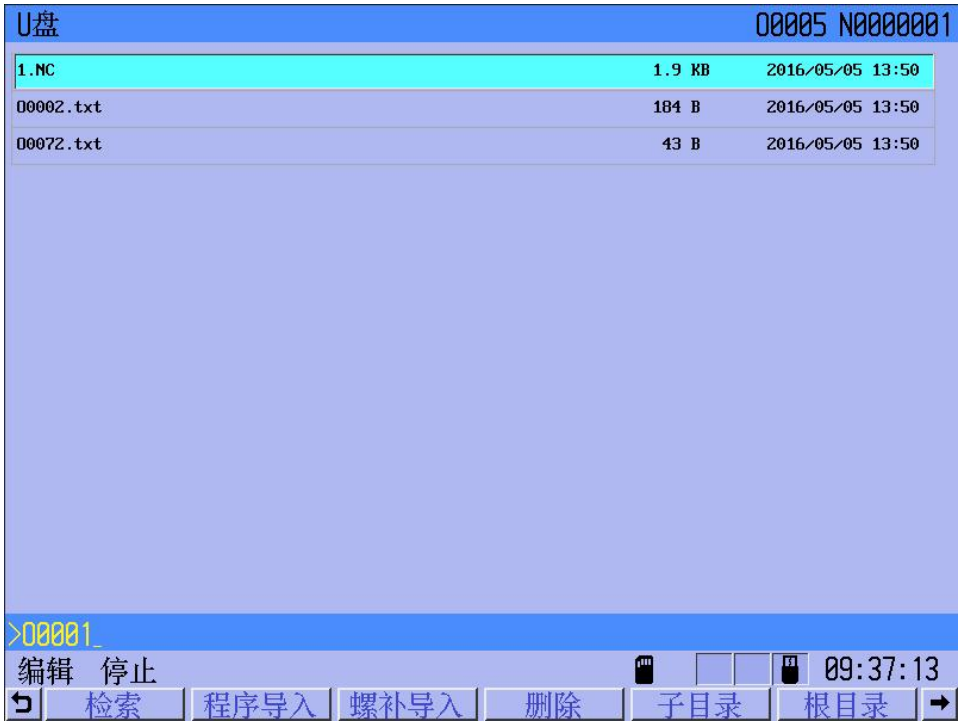
In system editing mode, switch to the [PROG program] interface [Mobile Storage] interface. See the U-disk interface for the specific operation steps;

After inserting the U disk, the lower right corner of the system displays the "U disk icon" and loads the U disk content;

➡ Parameter backup: press the soft key on the screen [Operation] soft button at the bottom of the screen [] soft button at the bottom of the screen [Parameter backup], after completion, the interface displays "backup complete". Backup content supports system parameters and system PLC files;

➡ Parameter recovery: press the soft key on the screen [Operation] soft button at the bottom of the screen [] soft button at the bottom of the screen [parameter recovery], after completion, the interface displays "recovery complete". Recovery content to support the system parameter files;

12 Program input



graph 3-56

In system editing mode, switch to the [PROG program] interface [Mobile Storage] interface. See the U-disk interface for the specific operation steps;

After inserting the U disk, the lower right corner of the system displays the "U disk icon" and loads the U disk content;

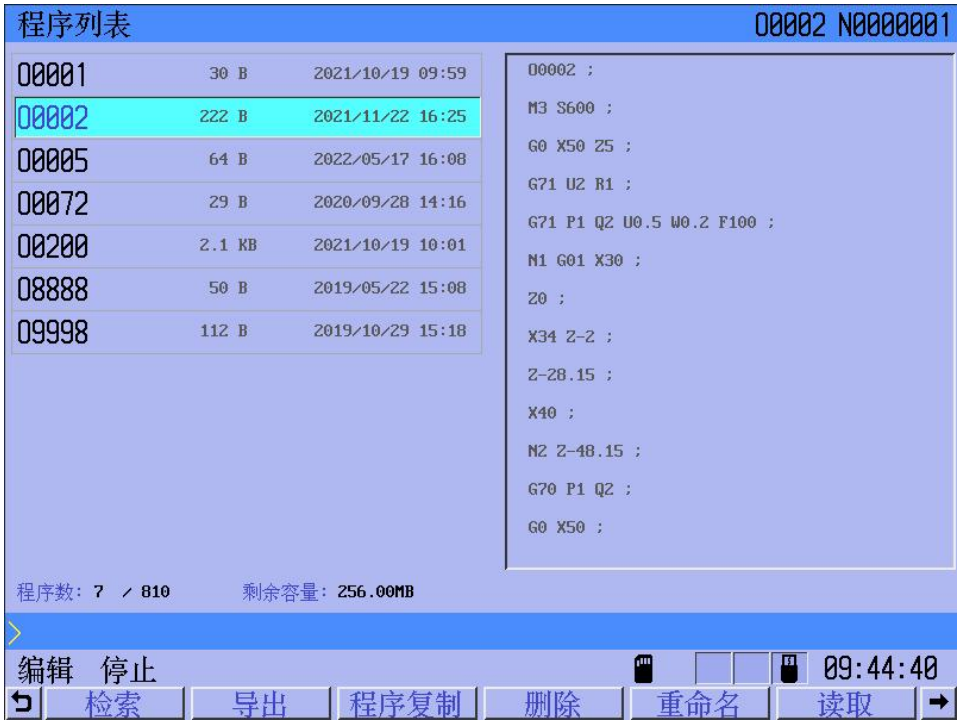
Press the soft key [Move Storage], move the cursor to select the U disk program;

Enter the program name such as: O 0001, press [program import] to complete.

[pay attention to]

The input program name should start with the English letter "O", and then type the custom four Arabic digits to name it;

13 Program export



graph 3-57

In system editing mode, switch to [PROG program] interface [list] interface;

After inserting the U disk, the lower right corner of the system displays the "U disk icon" and loads the U disk content;

Press the soft button on the screen [Operation] [Export] on the soft button below the screen, and the interface displays "Export complete";

[pay attention to]

The program name of the system will start with the English letter "O", followed by four Arabic digits (the name of the exporter) + underscore "_" out. The NC was named. Such as: O 0002_out.NC ；

Article 4 Equipment connection

1. System composition

The CNC system is mainly composed of the following units:

（1）numerical control system;

（2）Additional operation panel and accessories (optional);

（3）Digital AC servo unit / step drive unit;

（4） Servo motor / stepping motor;

（5）isolation transformer.

2. System installation and connection

First of all, check whether the CNC system, driver, motor, photoelectric encoder and other components that need to be installed are complete, intact and matched.

The CNC system must be installed firmly, and some space should be reserved around the system to ensure that the heat of the system can be distributed. The CNC system shall be installed in a place that is easy to operate and avoids processing iron chips and coolant.

Strong and weak current should be separated, and the power supply of the CNC system and the drive unit should be provided by the isolation transformer, separated from the strong current part of the machine tool. The various signal lines should be kept far away from the AC contactor to reduce interference. When designing the electric control cabinet wiring, attention should be paid to separate the signal line and the power line.

Connect the plugs firmly, tighten the fixing screw, and prohibit the live plug of the system.

There should be no strong electric, strong magnetic interference source around the CNC system, such as electric spark, laser cutting machine and other high-frequency equipment. At the same time, try to stay away from flammable, explosive goods and all kinds of dangerous goods.

3. Equipment installation size

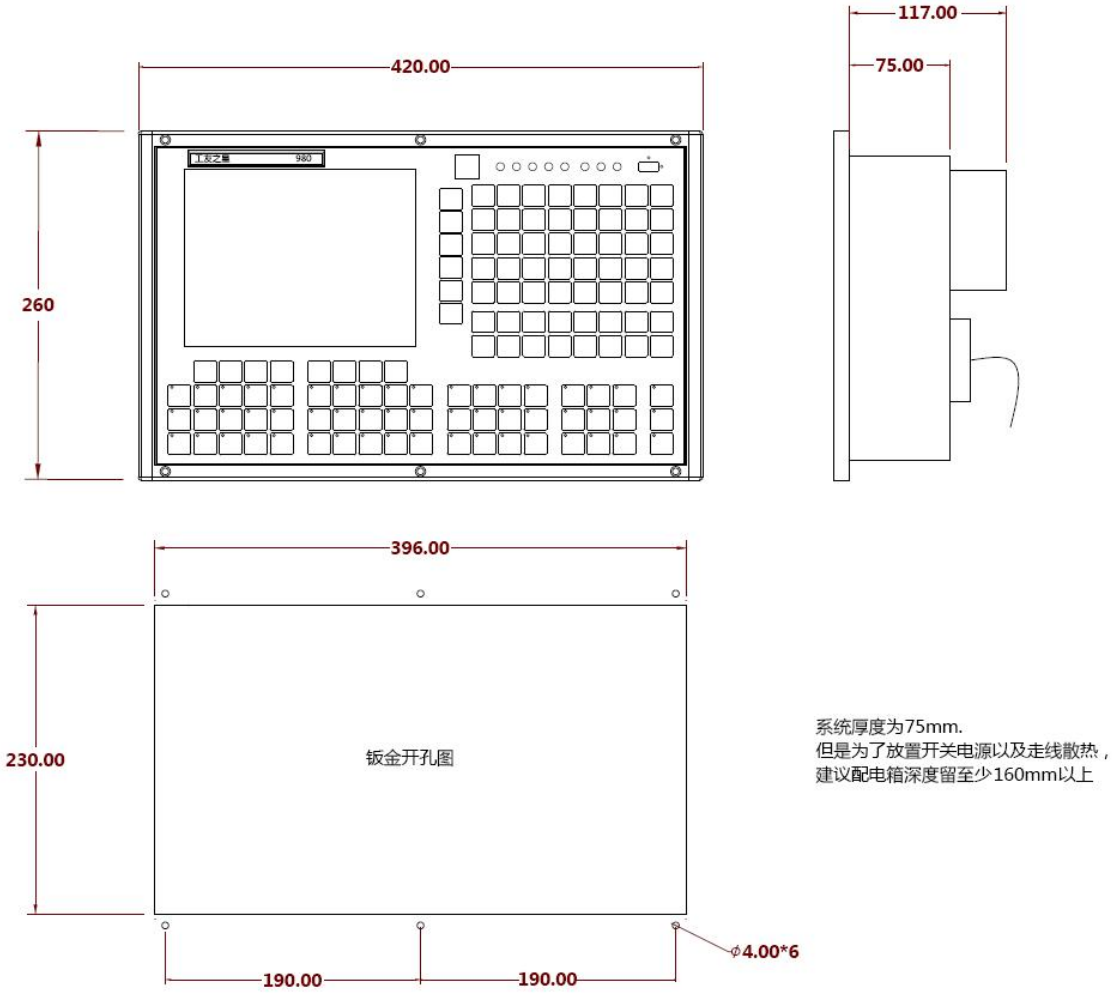


Figure 4-1 Installation dimensions of the system

4. Interface layout

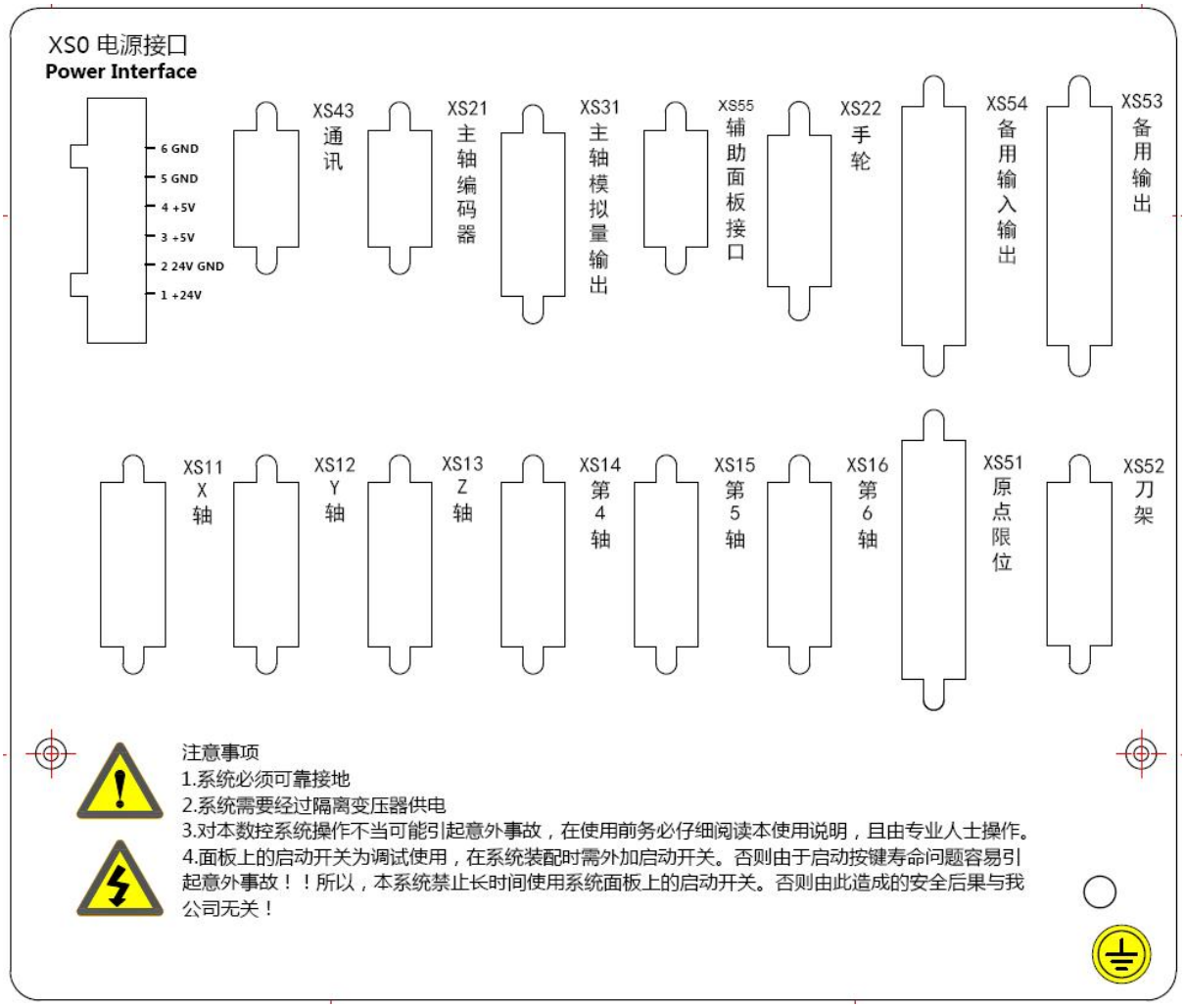


Figure 4-2 Back view of the interface layout

Note: The switch power supply L and N must be connected to AC 220 and the current is 0.5A.

4.1 Feed motor interface

Feed motor interface, DB 15-hole connector, labeled XS 11 ~ XS 16. XS 11 = X-axis, XS 12 = Y-axis, XS 13 = Z, axis, XS 14 = 4th axis, XS 15 = 5th axis, S16= 6th axis.

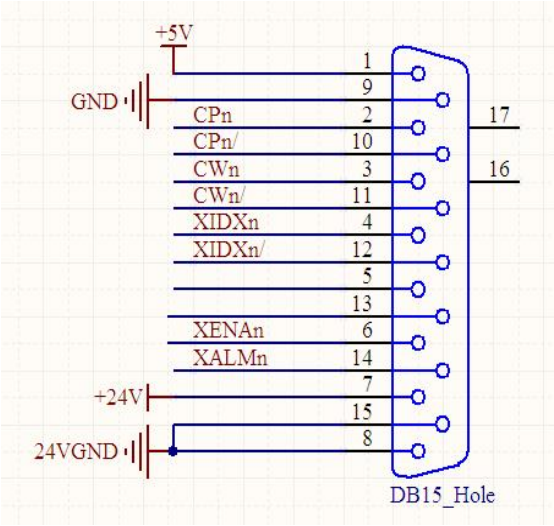


Figure 4-3 Feed motor interface

Table 4-1 Feed motor interface

XS 11-XS 16 feed motor interface DB15 hub				
signal	Pipe	I/O	function	Effective level
+5V	1	OUT	The 5V power supply	5V
GND	9	OUT	5V power supply	0V
CPn	2	OUT	pulsing signal +	5V
CPn/	10	OUT	pulsing signal-	5V
CWn	3	OUT	Direction signal +	5V
CWn/	11	OUT	Direction Signal-	5V
XIDXn	4	IN	Index (zero-bit) signal +	5V
XIDXn/	12	IN	Index (zero-bit) signal-	5V
XENAn	6	OUT	Servo drive enables	0V
XALMn	14	IN	Servo drive alarm	0V
+24V	7	OUT	The 24V power supply	24V
24VGND	8、15	OUT	24V, the power supply	0V

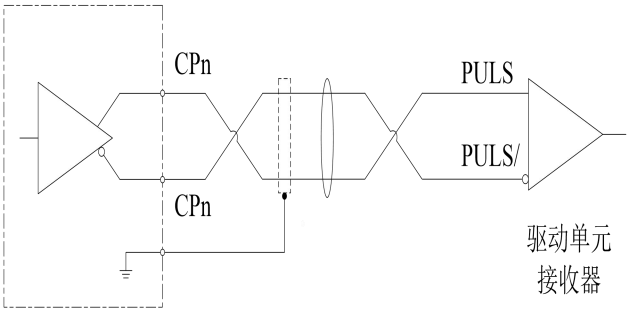
pay attention to:

1. The drive signal line must be twisted-pair shielding cable, and the length shall not exceed 10m.

2. The input mode of alarm signal ALM is determined by "Connection (T) parameter" (1009-1014).

4.1.1 Pulse motion code signal:

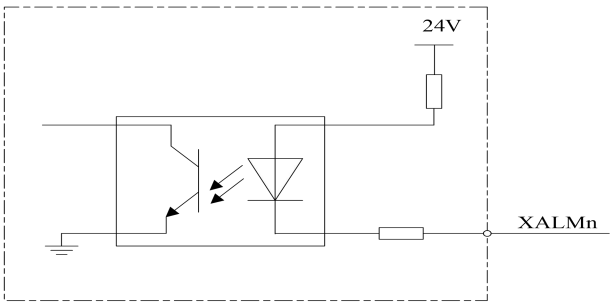
CPn and CPn / are pulse signals, CWn and CWn / are directional signals, XIDXn and XIDXn / are index (zero) signals, and the three signals are differential signals.5V is the power output of the differential signal 5V, and GND is the differential signal ground. The difference signal shall be twisted pair with shield wire and grounded with shield end. The connection circuit diagram is as follows:



graph 4-4

4.1.2 Alarm signal of the driving unit XALMn:

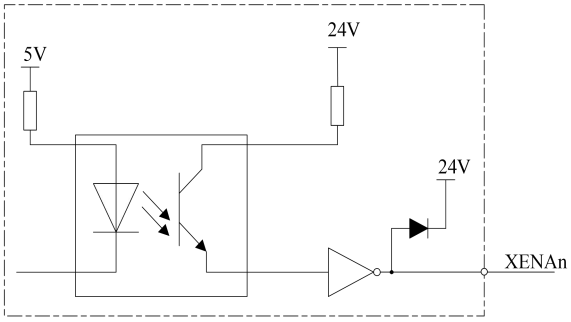
The mode of receiving the signal on the system side is as follows. Low level is effective.



graph 4-5

4.1.3 System servo enable signal XENAn:

The servo enabling signal XENAn is the open circuit output of the collector, and the low level is effective.



graph 4-6

4.1.4 Connecdiagram of common servo drives

Yaskawa Σ -V series servo driver connection diagram 4-7 is as follows:

系统端			安川Σ-V系列-CN1	
信号	管脚		信号	管脚
CPn	2		PULS	7
CPn/	10		PULS/	8
CWn	3		SIGN	11
CWn/	11		SIGN/	12
XIDXn	4		PC0	19
XIDXn/	12		PC0/	20
5V	1			
GND	9			
+24V	7		+24V	47
XENAn	6		/S-ON	40
XALMn	14		ALM+	31
24VGND	15		ALM-	32
24VGND	8			

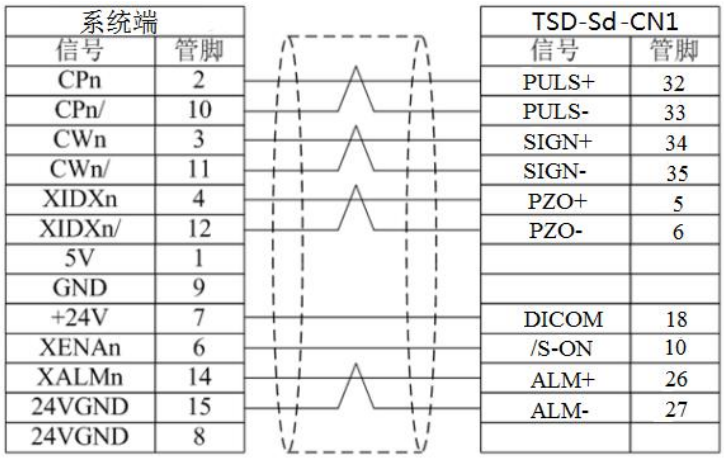
graph 4-7

TSD-H1E series servo driver connection diagram: Figure 4-8:

系统端			TSD-H1E-CN1	
信号	管脚		信号	管脚
CPn	2		PLUS+	20
CPn/	10		PLUS-	7
CWn	3		SIGN+	19
CWn/	11		SIGN-	6
XIDXn	4		OZ+	13
XIDXn/	12		OZ-	25
5V	1			
GND	9			
+24V	7		+24V	1
XENAn	6		SON	14
XALMn	14		ALM	17
24VGND	15		COM	18
24VGND	8			

graph 4-8

TSD-Sd series servo drive connection diagram as 4-9:



graph 4-9

4.2 Origin limit interface

Origin limit interface, D B 25 needle connector, system number XS 51.

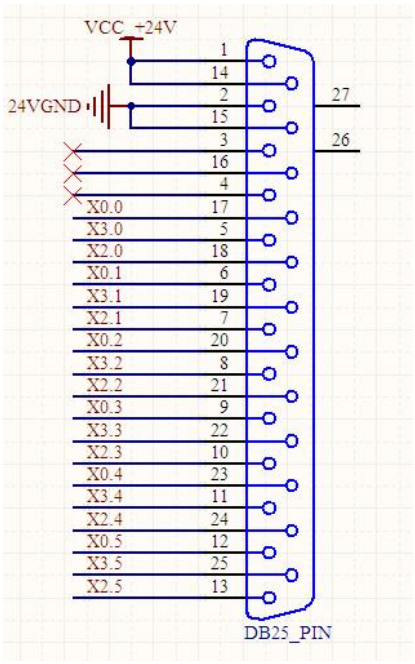


Figure 4-10 Origin limit interface

Table 4-2 Origin limit interface

XS 51 Origin limit interface DB25 pin holder					
signal	Pipe foot	I/O	function	Custom M code	Effective level
VCC_+24V	1、14	OUT	24V power supply		+24V
24VGND	2、15	OUT	24V, the power supply		0V

X0.0	17	IN	X axis origin	M100	0V
X3.0	5	IN	The X-axis positive	M130	0V
X2.0	18	IN	The X-axis negative	M120	0V
X0.1	6	IN	Origin of Y axis	M101	0V
X3.1	19	IN	Positive limit of the	M131	0V
X2.1	7	IN	The Y-axis negative	M121	0V
X0.2	20	IN	Z axis origin	M102	0V
X3.2	8	IN	Z-axis positive limit	M132	0V
X2.2	21	IN	Negative limit of the	M122	0V
X0.3	9	IN	Origin on axis 4	M103	0V
X3.3	22	IN	Positive limit on axis	M133	0V
X2.3	10	IN	Negative limit on axis	M123	0V
X0.4	23	IN	The 5th axis origin	M104	0V
X3.4	11	IN	Positive limit on axis	M134	0V
X2.4	24	IN	Negative limit on axis	M124	0V
X0.5	12	IN	The 6th axis origin	M105	0V
X3.5	25	IN	Positive limit on axis	M135	0V
X2.5	13	IN	Negative limit on axis	M125	0V

4.3 Spindle analog output and output interface

Spindle simulation and output interface, D B15 pin connector, system label are XS 31.

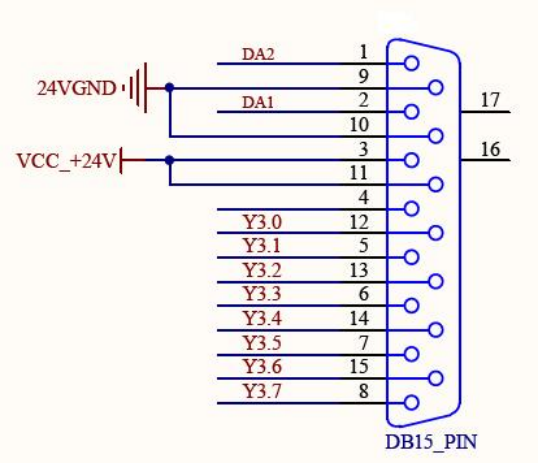


Figure 4-11, the spindle interface

Table 4-3 Spindle interface

XS 31 spindle analog capacity and output interface DB15 pin seat					
signal	Pipe foot number	I/O	function	M custom code	Effective level
DA2	1	OUT	Simog 2nd spindle output		0-10V
DA1	2	OUT	Simog 1st spindle output		0-10V
24VGND	9、10	OUT	24VGND		0V
VCC_24V	11	OUT	24V power supply		+24V
X5.1	3	IN	Quite stop in place	M151	0V
X5.6	4	IN	Spindle alarm	M156	0V
Y3.0	12	OUT	The spindle is turning	M730/M930	0V
Y3.1	5	OUT	Spindle stop	M731/M931	0V
Y3.2	13	OUT	Spindle reversal	M732/M932	0V
Y3.3.	6	OUT	The main shaft quasi stop	M733/M933	0V
Y3.4	14	OUT	C/S pattern	M734/M934	0V
Y3.5	7	OUT	Processing blowing	M735/M935	0V
Y3.6	15	OUT	Hydraulic station output	M736/M936	0V
Y3.7	8	OUT	cooling	M737/M937	0V

4.4 Standby output interface

Standby output interface, DB 25-hole connector, system code XS 53.

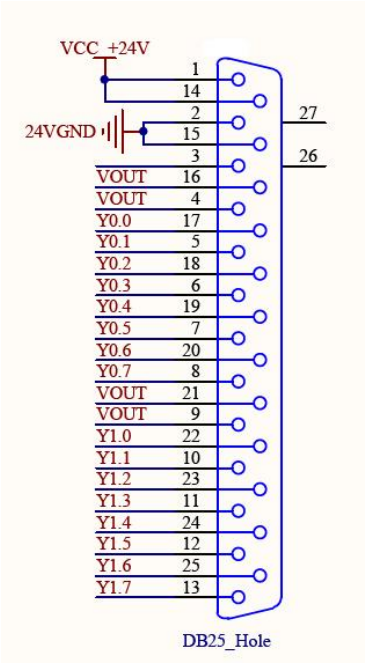


Figure 4-12 The standby output interface

Table 4-4 The standby output interface

XS 53 Standby output interface DB25 hub					
signal	Pipe foot number	I/O	function	Custom M code	Effective level
Y1.7	13	OUT	reserve	M717/M917	0V
Y1.6	25	OUT	reserve	M716/M916	0V
Y1.5	12	OUT	reserve	M715/M915	0V
Y1.4	24	OUT	reserve	M714/M914	0V
Y1.3	11	OUT	reserve	M713/M913	0V
Y1.2	23	OUT	reserve	M712/M912	0V
Y1.1	10	OUT	reserve	M711/M911	0V
Y1.0	22	OUT	Quick clamping	M710/M910	0V
VOUT	4、9、16、21	IN	A 24V power supply input		+24V
Y0.7	8	OUT	work light	M707/M907	0V
Y0.6	20	OUT	Tricolor light-red light	M706/M906	0V
Y0.5	7	OUT	Three-color light-yellow light	M705/M905	0V

Y0.4	19	OUT	Tricolor light-green light	M704/M904	0V
Y0.3	6	OUT	reserve	M703/M903	0V
Y0.2	18	OUT	reserve	M702/M902	0V
Y0.1	5	OUT	reserve	M701/M901	0V
Y0.0	17	OUT	reserve	M700/M900	0V
24VGND	2、15	OUT	24V power supply		0V
VCC_+24V	1、14	OUT	24V power supply		+24V

pay attention to:

1. When the standby output interface is used, the VOUT needs to connect to the + 24V power supply. If the + 24V is the external power supply, the external 24V power supply should be shared with the system power supply ground.

4.5 Standby input and output interface

Standby input / output, DB25-hole connector, system number XS 54.

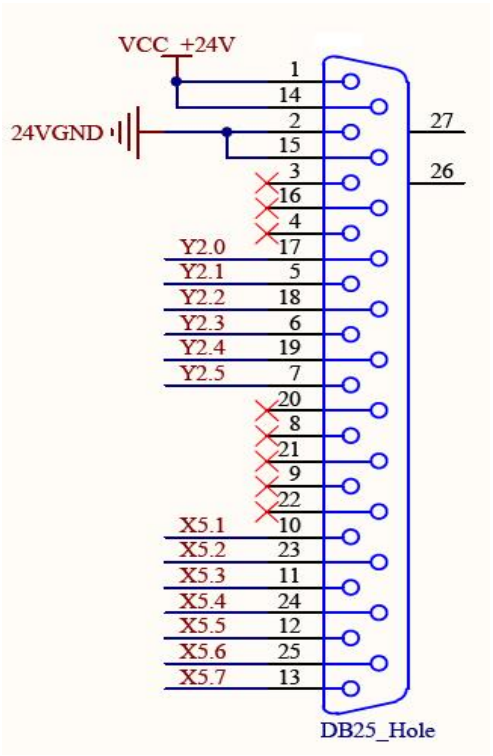


Figure 4-13 Alternate input / output interface

Table 4-5 The standby input and output interfaces

XS 54 standby input / output interface DB25 hub						
signal	Pipe foot number	I/O	Disk knife bank	The bamboo hat knife library	Custom M code	Effective level
VCC_+24V	1、14	OUT	24V power supply			+24V
24VGND	2、15	OUT	24V power supply			0V
Y2.0	17	OUT	On the knife cover	Knife library launched	M720/M920	0V
Y2.1	5	OUT	Under the knife set	Knife library to recover	M721/M921	0V
Y2.2	18	OUT	Spindle pine knife		M722/M922	0V
Y2.3	6	OUT	The robotic arm starts		M723/M923	0V
Y2.4	19	OUT	Plambing machine		M724/M924	0V
Y2.5	7	OUT	exit		M725/M925	0V
X5.2	23	IN	Spindle pine knife button		M152/M162	0V
X5.3	11	IN	The spindle pine knife is in place		M153/M163	0V
X5.4	24	IN	The spindle clip knife is in place		M154/M164	0V
X5.5	12	IN	Air pressure alarm detection		M155/M165	0V
X5.7	13	IN	Urgent stop signal			0V

4.6 knife rack interface

Knife holder connector, DB15-hole Type D connector, system number XS 52.

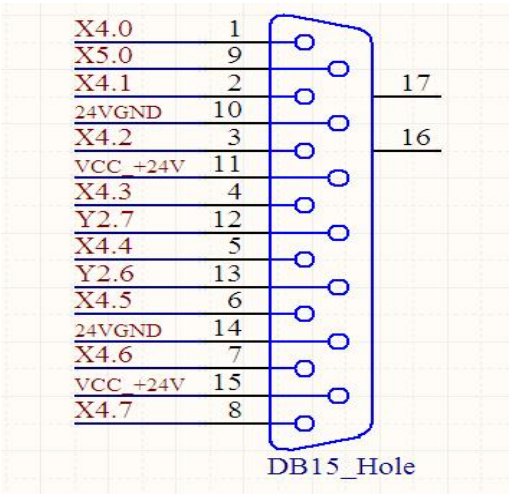


Figure 4-14 Knife rack interface

Table 4-6 knife rack interface

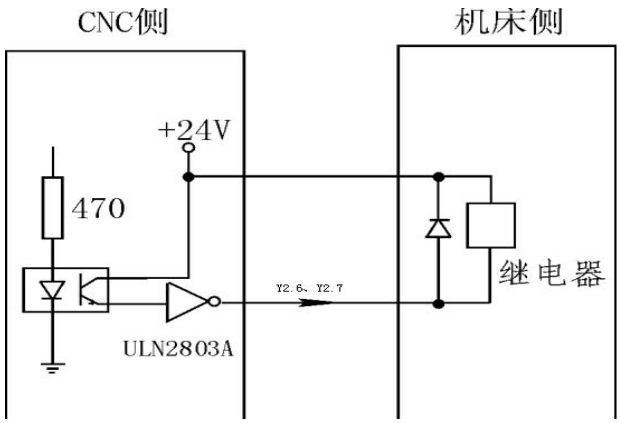
XS 52 electric tool holder interface DB15 hub						
signal	Pipe foot number	I/O	Disk knife bank	The bamboo hat knife library	Custom M code	Effective level
X4.0	1	IN	The buckle knife in place		M140	0V
X4.1	2	IN	Mechanical arm origin		M141	0V
X4.2	3	IN	Mechanical arm brake		M142	0V
X4.3	4	IN	The knife cup is in place		M143	0V
X4.4	5	IN	Under the knife cup in place		M144	0V
X4.5	6	IN	Knife library origin		M145	0V
X4.6	7	IN	Knife library count		M146	0V
X4.7	8	IN	Klibrary overload alarm		M147	0V
X5.0	9	IN			M150	0V
24VGND	10、14	OUT	24VGND			0V
VCC_+24V	11、15	OUT	+24V			+24V
Y2.6	12	OUT	Knife library reversal		M726/M926	0V
Y2.7	13	OUT	The knife bank is turning		M727/M927	0V

pay attention to:

1. All the input IN / output OUT is for the system, and the input IN is input to the system by an external signal, and the output OUT is the output signal of the system to the external signal;

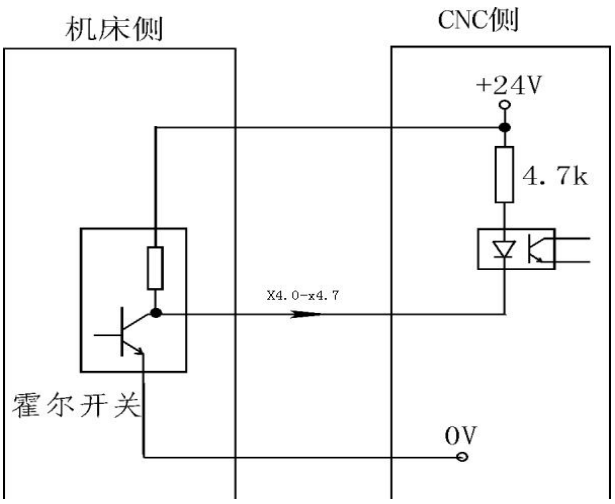
2. The positive turn Y2.6 and the reverse Y2.7 signal control the single contact intermediate relay, and the user should also install two AC contactors;

Schematic diagram of system output signals Y2.6 and Y2.7:



graph 4-15

Schematic of knife holder input signal X4.0~X4.7:



graph 4-16

4.7 Spindle encoder interface

Spindle encoder, DB 9 hole connector, system code is XS 21.

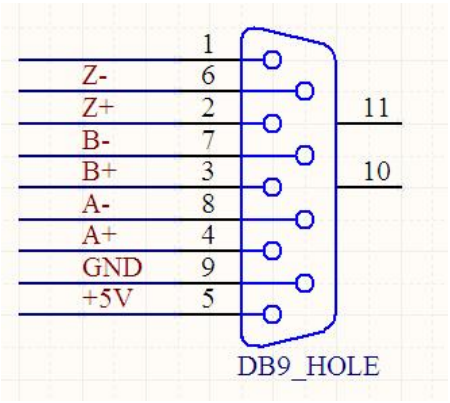


Figure 4-17 Spindle encoder interface

Table 4-7 Spindle encoder interface

XS 21 spindle encoder interface DB9 hub				
signal	Pipe foot	I/O	function	Effective level
Z-	6	IN	Spindle Encoder Z	5V
Z+	2	IN	Spindle encoder Z signal	5V
B-	7	IN	Spindle Encoder B	5V
B+	3	IN	Spindle encoder B	5V

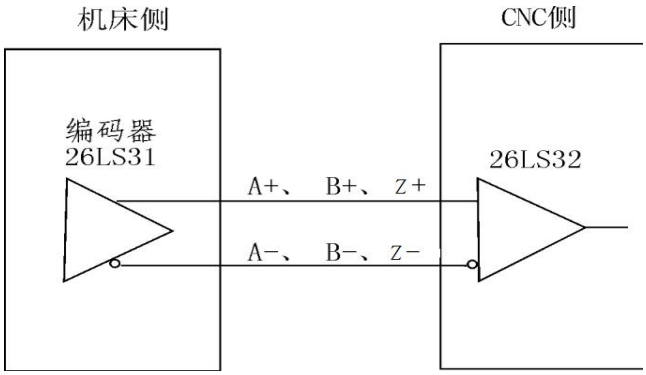
A-	8	IN	Spindle Encoder A	5V
A+	4	IN	Spindle encoder A signal	5V
GND	9	OUT	5V power supply	0V
+5V	5	OUT	5V power supply	+5V

pay attention to:

1. The output signal of encoder adopts long line output, and the power supply is + 5V.

2, the signal line must be twisted pair shielded cable, the length shall not exceed 10m.

Spindle encoder input signal A+, A-, B +, B-, Z +, Z-Schematic diagram:



4.8 Communication interface

Communication interface, DB 9 pin interface, system number XS 43.

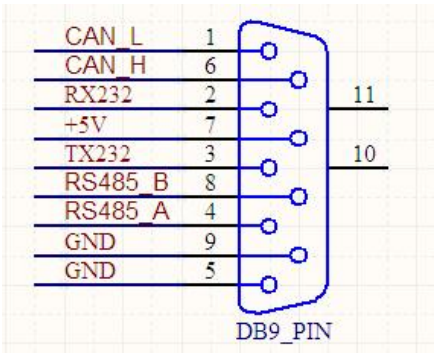


Figure 4-18 Communication interface

Table 4-8 Communication Interface

XS 43 Communication interface DB9 pin holder				
signal	Pipe foot	I/O	function	Effective level
R X 232	2	IN	RX 232 Receive the	

T X 232	3	OUT	The RX 232 sends a	
GND	5	OUT	The RS232 signal	0V
RS485_A	4	OUT	The RS485 signal, A	
RS485_B	8	OUT	The RS485 signal,	
GND	9	OUT	Power to	0V
CAN_L	1	OUT	CAN signal L	
CAN_H	6	OUT	CAN signal H	
+5V	7	OUT	5V power supply	+5V

4.9 Hand-wheel interface

Hand wheel interface, identify XS 22.

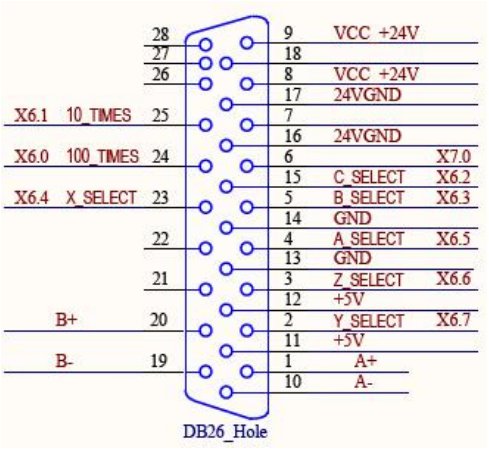


Figure 4-19, hand wheel interface

Table 4-9 Hand wheel interface

XS 22 Hand wheel interface DB26 hub				
signal	Pipe foot	I/O	function	Effective
A+	1	IN	Hand wheel encoder A	5V
A-	10	IN	Hand wheel encoder	5V
B-	19	IN	Hand wheel encoder	5V
B+	20	IN	Hand wheel encoder B	5V
+5V	11、12	OUT	5V power supply	+5V
GND	13、14	OUT	5V power supply	0V
X6.4	23	IN	Axis X	0V
X6.7	2	IN	Axis selection Y	0V

X6.6	3	IN	Axis selection Z	0V
X6.5	4	IN	Axis selection A	0V
X6.3	5	IN	Axis selection B	0V
X6.2	15	IN	Axis C	0V
X6.1	25	IN	Fold x10	0V
X6.0	24	IN	The fication x100	0V
X7.0	6	IN	jerk	0V
VCC_+24V	8、9	OUT	24V power supply	+24V
24VGND	16、17	OUT	24V, the power supply	0V

4.10 Auxiliary panel interface

Auxiliary panel interface is used to connect the additional panel, used for the use of external switch button, etc., marked XS 55.

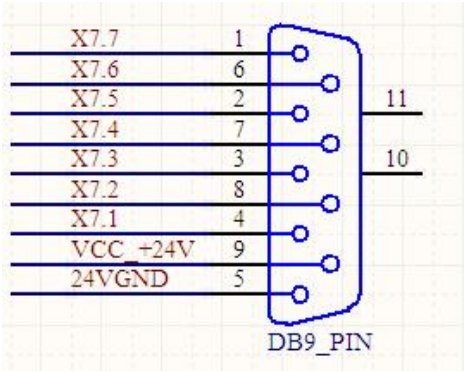


Figure 4-20 Auxiliary panel interface

Table 4-10 Auxiliary panel interface

XS 55 Auxiliary panel interface DB9 pin holder				
signal	Pipe foot	I/O	function	Effective level
X7.7	1	IN	Urgent stop signal	0V
X7.6	6	IN	External start	0V
X7.5	2	IN	External suspension	0V
X7.4	7	IN	Quick clamping	0V
X7.3	3	IN		0V
X7.2	8	IN		0V
X7.1	4	IN		0V

VCC_+24V	9	OUT	24V power supply	+24V
24VGND	5	OUT	24V, the power supply	0V

The fifth Appendix

Appendix I parameter description

velocity N			
Param	**Windo**	**The parameter name**	**span**
0500	7000	Automatic feed, give the speed limit	0-30000
0533	7000	X-axis automatic feed limit	0-999999
0534	7000	The Y-axis automatic feed speed limit	0-999999
0535	7000	Z-axis automatic feed speed limit	0-999999
0536	4000	The A-axis automatic feed speed limit	0-999999
0537	4000	Axis B is based on the automatic feed speed limit	0-999999
0538	4000	The C-axis automatic feed speed limit	0-999999
0504	10000	Empty running speed	0-999999
0502	7000	Automatic fast shift speed at 100% multiplier (G00 linear interpolation)	0-999999
0503	1000	F0 fast velocity (G00 linear interpolation)	0-999999
0542	7000	X-axis automatic fast shift speed limit, G0 non-linear 100% times rate fast shift speed	0-999999
0543	7000	Y-axis automatic fast shift speed limit, G0 non-linear 100% times rate fast shift speed	0-999999
0544	7000	Z-axis automatic fast shift speed limit, G0 nonlinear 100% times rate fast shift speed	0-999999
0545	4000	Axis A automatic fast shift speed limit, G0 nonlinear 100% times rate fast shift speed	0-999999
0546	4000	B axis automatic fast shift speed limit, G0 nonlinear 100% times rate fast shift speed	0-999999
0547	4000	C-axis automatic fast shift speed limit, G0 nonlinear 100% times rate fast shift speed	0-999999
0553	1000	Fast shift speed (G00 linear interpolation) at X-axis F0 x rate	0-999999
0554	1000	Fast shift speed (G00 linear interpolation) at Y axis F0 rate	0-999999
0555	1000	Fast shift speed (G00 linear interpolation) at Z axis F0 fold rate	0-999999
0556	1000	Fast shift speed (G00 linear interpolation) at axis F0	0-999999
0557	1000	Fast shift speed (G00 linear interpolation) at B axis F0 fold rate	0-999999
0558	1000	Fast shift speed (G00 linear interpolation) at C axis F0 rate	0-999999
1072	6000	Manual fast shift speed at motor of 1 100% multiplier	0-999999

1073	6000	Motor 2 100% fold speed when manual fast shift speed	0-999999
1074	6000	Motor 3 100% multiplier	0-999999
1075	6000	Motor 4 100% fold speed when manual fast shift speed	0-999999
7076	6000	Motor 5 100% multiplier	0-999999
1077	6000	Motor 6 100% power rate when the manual fast shift speed	0-999999
1081	1000	Manual fast shift speed at motor of 1 100% multiplier	0-999999
1082	1000	Motor 2 F0 fold-speed when manual fast shift speed	0-999999
1083	1000	Motor 3 F0 times rate when the manual fast shift speed	0-999999
1084	1000	Manual fast shift speed at motor 4 F0 x rate	0-999999
1085	1000	Motor 5 F0 times rate when the manual fast shift speed	0-999999
1086	1000	Manual speed shift at 6 F0 rate	0-999999
1063	100	Motor 1 is at a manual feed speed	0-999999
1064	100	Motor 2 manual feed speed	0-999999
1065	100	Motor 3 manual feed speed	0-999999
1066	100	Motor 4 manual feed speed	0-999999
1067	100	Motor 5 manual feed speed	0-999999
1068	100	Motor 6 manual feed speed	0-999999
0520	500	The feed speed of the longest moving auxiliary axis when the I feed axis displacement is 0	0-999999
0576	1	The forward rate (%) corresponding to 1 pulse / second in the hand wheel trial cut	0-100
0577	1	Hand wheel test cutting fast shift rate (%)	0-100
0578	1	If axis A is the synthetic feed speed, reference axis 0: No 1: Yes	0-100
0579	1	If axis B is the synthetic feed speed, refer to axis 0: No 1: Yes	0-100
0580	1	Whether axis C is the synthetic feed speed, reference axis 0: No 1: Yes	0-100
accelerated speed G			
3841	1	Reduce 0: off 1: linear 2: S	And 0 or 1 or 2
0505	10000	S-type acceleration and deceleration for the maximum acceleration	0-10000
0506	50	S-type acceleration, deceleration and acceleration time	0-10000
0507	3000	Maximum acceleration of linear acceleration and deceleration	0-10000
0564	3000	Linear plus or minus maximum acceleration (Group 1)	0-10000

0567	3000	Linear maximum acceleration (Group 2)	0-10000
1405	5	Automatic rear-acceleration and deceleration time constant of motor 11 (ms)	0-100
1406	5	Motor 2 automatic rear acceleration and deceleration time constant 1 (ms)	0-100
1407	5	Motor 3 automatic rear acceleration and deceleration time constant 1 (ms)	0-100
1408	5	Motor 4 automatic rear acceleration and deceleration time constant 1 (ms)	0-100
1409	5	Automatic rear-acceleration and deceleration time constant of motor 51 (ms)	0-100
1410	5	Motor 6 automatic rear acceleration and deceleration time constant 1 (ms)	0-100
1414	10	Motor 1 automatic rear acceleration and deceleration time constant 2 (ms)	10-1000
1415	10	Motor 2 automatic rear acceleration and deceleration time constant 2 (ms)	10-1000
1416	10	Motor 3 automatic rear acceleration and deceleration time constant 2 (ms)	10-1000
1417	10	Motor 4 automatic rear acceleration and deceleration time constant 2 (ms)	10-1000
1418	10	Automatic rear-acceleration and deceleration time constant of motor 52 (ms)	10-1000
1419	10	Motor 6 automatic rear acceleration and deceleration time constant 2 (ms)	10-1000
1423	5	Reacceleration and deceleration time constant of motor 1 (ms)	0-100
1424	5	Reacceleration and deceleration time constant 1 (ms)	0-100
1425	5	Reacceleration and deceleration time constant of motor 3 hands 1 (ms)	0-100
1426	5	Reacceleration and deceleration time constant of motor 4 hand wheel 1 (ms)	0-100
1427	5	Reacceleration and deceleration time constant of motor 1 (ms)	0-100
1428	5	Reacceleration and deceleration time constant 1 (ms)	0-100
1432	20	Reacceleration and deceleration time constant of motor 1 (ms)	10-1000
1433	20	Reacceleration and deceleration time constant of motor 2 (ms)	10-1000
1434	20	Reacceleration and deceleration time constant 2 (ms)	10-1000
1435	20	Reacceleration and deceleration time constant of motor 2 (ms)	10-1000
1436	20	Reacceleration and deceleration time constant 2 (ms)	10-1000
1437	20	Reacceleration and deceleration time constant 2 (ms)	10-1000
3845	0	Pause time at the end of the fast move (ms)	0-1000
3844	0	Pause time at the end of the feed (ms)	0-1000
0571	1	Stop plug 0: Slow ate 1: Do not slow down	0 Or 1
0509	600	Maximum angle speed for this variable	0-10000
0510	0.001	The circular arc	0.001-1

0518	0.3	Speed smoothing upper bound range (ratio)	0-1000
0519	0.3	Speed Smooth Underbounding Range (ratio)	0-1000
0521	600	The maximum speed stage allowed by the X-axis	0-1000
0522	600	The maximum velocity phase allowed by the Y-axis	0-1000
0523	600	The maximum velocity stage allowed by the Z-axis	0-1000
0524	600	The maximum velocity stage allowed by the A axis	0-1000
0525	600	The maximum velocity phase allowed by the B axis	0-1000
0526	600	The maximum velocity phase allowed by the C-axis	0-1000
programming X			
3804	1	Increment unit of 0: IS-A 1: IS-B 2: IS-C	0-2
3004	1	The system is in 0: clear state 1: reset state	0 Or 1
3801	0	0: G00 1: G01 when the power is on or cleared	0 Or 1
3802	0	0: G17 1: G18 2: G19	0-2
3803	0	03 mode 0: G90 1: G91	0 Or 1
3965	1	03 mode 0: G99 1: G98	0 Or 1
3805	1	06 mode 0: G20 1: G21	0 Or 1
3846	0	G00 insertion method 0: Linear 1: Nonlinear	0 Or 1
3806	0.03	Ararc interpolation error maximum	0.001-10
3808	0.5	Return amount d of G73 (mm)	0-1000
3963	0.5	Return amount d of G83 (mm)	0-1000
3811	0	G74 / G84 Rigid tapping mode 0: high-speed deep hole 1: deep hole	0 Or 1
3812	1000	G74 / G84 rigid tapping after each penetration	0-9999999
3831	1	G74 / G84	1-5
3833	0.5 5	G74 / G84 Return amount during rigid tapping (mm)	0-9999999
3903	1	Q in G74 / G84 tapping 0: Invalid 1: valid	0 Or 1
3959	1	G76 / G87 (-3~3)	-3~3
0572	0	Move the shortest path direction of 0: No 1: Yes	0 Or 1
0595	1	Does G0 use feed multiplier 0: No 1: Yes	0 Or 1
3997	0	Helical gear compensation direction 0: do not reverse 1: reverse	0 Or 1

3904	0	Scale scale is valid 0: Invalid 1: valid	0 Or 1
3960	0	Tool length compensation type 0: A1: B	0 Or 1
3906	0	Select the tip radius compensation type of 0: A 1: B	0 Or 1
3907	0	Is the tip radius compensation type C enabled 0: No 1: Yes	0 Or 1
3908	0	The tip radius compensated connection type 0: line 1: arc	0 Or 1
3909	8	The tip radius compensates for the number of prereads	3-8
3961	0	The tip radius compensation interference check 0: Invalid 1: valid	0 Or 1
3861	100	Call the G code for the macroprogram 09010	0-999999
3862	100	Call the G code for the macroprogram 09011	0-999999
3863	100	Call the G code for the macro program 19012	0-999999
3864	100	Call the G code for the macroprogram 19013	0-999999
3865	100	Call the G code for the macroprogram 19014	0-999999
3866	100	Call the G code for the macro program 19015	0-999999
3867	100	Call the G code for the macroprogram 19016	0-999999
3868	100	Call the G code for the macroprogram 19017	0-999999
3869	100	Call the G code for the macroprogram 19018	0-999999
3870	100	Call the G code for the macroprogram 19019	0-999999
3871	100	The actual value of the macro program G code	101-9999
3872	1	The macroprogram starting program number is called via the G code	1-9999
3873	1	Number of macroprograms called by the G code	1-9999
3874	100	The M code for calling the macro program 09020	0-9999
3875	100	The M code for calling the macro program 09021	0-9999
3876	100	The M code for calling the macro program 09022	0-9999
3877	100	The M code for calling the macro program 09023	0-9999
3878	100	The M code for calling the macro program 09024	0-9999
3879	100	The M code for calling the macro program 09025	0-9999
3880	100	The M code for calling the macro program 09026	0-9999
3881	100	Call the M code of the macro program 09027	0-9999
3882	100	The M code for calling the macro program 09028	0-9999
3883	100	Call the M code of the macro program 09029	0-9999

3884	100	Starting value of calling the macro program M code	101-9999
3885	1	The macroprogram starting program number is called by the M code	1-9999
3886	1	The number of macroprograms called by the M code	1-9999
3887	6	Call the M code for the subprogram 09001	0-9999
3888	100	Call the M code for the subprogram 09002	0-9999
3889	100	Call the M code for the subprogram 09003	0-9999
3890	100	Call the M code for the subprogram 09004	0-9999
3891	100	Call the M code for the subprogram 09005	0-9999
3892	100	Call the M code for the subprogram 09006	0-9999
3893	100	Call the M code for the subprogram 09007	0-9999
3894	100	Call the M code for the subprogram 09008	0-9999
3895	100	Call the M code for the subprogram 09009	0-9999
3896	100	Call to the subprogram M code, the starting value	101-9999
3897	1	Starting program number via M code	1-9999
3898	1	Number of subprograms called by the M code	1-9999
3899	0	Whether to allow T code call subprogram 0: does not allow 1: allowed	0 Or 1
3900	0	Call the character for the subprogram 09004	65-90
3901	0	Call the character of the subprogram 09005	65-90
3211	0	When switching the common British system, the X-axis is at the non-reference point 0: no alarm 1: alarm	0 Or 1
3212	0	When switching the mesystem, the Y axis is at non-reference point 0: no alarm 1: alarm	0 Or 1
3213	0	When switching the common English system, the Z-axis is at the non-reference point 0: no alarm 1: alarm	0 Or 1
3214	0	When switching commons, when the coordinate system offset exists, 0: Alarm 1: clear	0 Or 1
compensate Y			
1099	0	Motor 1 Reverse clearance (mm)	0-999999
1100	0	Motor 2 Reverse clearance (mm)	0-999999
1101	0	Motor 3 Reverse clearance (mm)	0-999999
1102	0	Motor 4 Reverse clearance (mm)	0-999999
1103	0	Motor 5 Reverse clearance (mm)	0-999999
1104	0	Motor 6 Reverse clearance (mm)	0-999999

1495	0	Motor 1 reverse clearance compensation mode: 0: time smoothing 1: position smoothing	0 Or 1
1496	0	Motor 2 reverse clearance compensation mode: 0: time smoothing 1: position smoothing	0 Or 1
1497	0	Motor 3 reverse clearance compensation mode: 0: time smoothing 1: position smoothing	0 Or 1
1498	0	Motor 4 reverse clearance compensation mode: 0: time smoothing 1: position smoothing	0 Or 1
1499	0	Motor 5 reverse clearance compensation mode: 0: time smoothing 1: position smoothing	0 Or 1
1500	0	Motor 6 reverse clearance compensation mode: 0: time smoothing 1: position smoothing	0 Or 1
1117	0.1	Maximum compensation per control cycle (number of pulses)	0-999999
1118	0.1	Maximum compensation amount per control cycle (number of pulses)	0-999999
1119	0.1	Maximum compensation per control cycle (number of pulses)	0-999999
1120	0.1	Motor 4 reverse clearance maximum compensation per control cycle (number of pulses)	0-999999
1121	0.1	Motor 5 maximum clearance clearance per control cycle (number of pulses)	0-999999
1122	0.1	Motor 6 reverse clearance maximum compensation per control cycle (number of pulses)	0-999999
1504	0	Motor 1 reverse, clearance first stage output (number of pulses)	0-9999999
1505	0	Motor 2 reverse, clearance first stage output (pulses)	0-9999999
1506	0	Motor 3 reverse, clearance first stage output (number of pulses)	0-9999999
1507	0	Motor 4 reverse, clearance first stage output (pulses)	0-9999999
1508	0	Motor 5 reverse, clearance first stage output (number of pulses)	0-9999999
1509	0	Motor 6 reverse, clearance first stage output (number of pulses)	0-9999999
1513	0	Start position of motor 1 (pulse number)	0-9999999
1514	0	Start position of the second stage output of the reverse clearance of motor 2 (pulse number)	0-9999999
1515	0	Start position of the second stage output of motor 3 reverse clearance (number of pulses)	0-9999999
1516	0	Start position of motor 4 (pulse number)	0-9999999
1517	0	Start position of second stage output of motor 5 (number of pulses)	0-9999999
1518	0	Start position of second stage output of motor 6 (pulse number)	0-9999999
1522	0	End position of second stage output of motor 1 (pulse number)	0-9999999
1523	0	End position of second stage output of motor 2 (pulse number)	0-9999999
1524	0	End position of second stage output of motor 3 (pulse number)	0-9999999
1525	0	End position of second stage output of motor 4 (pulse number)	0-9999999
1526	0	End position of second stage output of motor 5 (pulse number)	0-9999999
1527	0	End position of second stage output of motor 6 reverse clearance (pulse number)	0-9999999

1126	0.5	Motor 1 pitch compensation point interval (mm)	0-999999
1127	0.5	Motor 2 pitch compensation point interval (mm)	0-999999
1128	0.5	Motor 3 pitch compensation point interval (mm)	0-999999
1129	0.5	Motor 4 pitch compensation point interval (mm)	0-999999
1130	0.5	Motor 5 pitch compensation point interval (mm)	0-999999
1131	0.5	Motor 6 pitch compensation point interval (mm)	0-999999
1135	33	Compensation number corresponding to the origin of motor 1	0-999999
1136	33	Compensation number corresponding to the origin of motor 2	0-999999
1137	33	The compensation number corresponding to the origin of motor 3	0-999999
1138	33	Compensation number corresponding to the origin of motor 4	0-999999
1139	33	The compensation number corresponding to the motor 5 pitch compensation origin	0-999999
1140	33	The compensation number corresponding to the origin of the motor 6	0-999999
1144	30	Motor 1 distance compensation number in the negative direction	0-999999
1145	30	Motor 2 is compensation in the negative direction	0-999999
1146	30	Motor 3 pitch compensation in the negative direction	0-999999
1147	30	Motor 4 compensates for the farthest end of the negative direction	0-999999
1148	30	Motor 5 compensates for the farthest end of the negative direction	0-999999
1149	30	Motor 6 compensates for the farthest end of the negative direction	0-999999
1153	35	Motor 1 compensates for the farthest end of the positive direction	0-999999
1154	35	Motor 2 compensates the farthest end of the positive direction	0-999999
1155	35	Motor 3 pitch compensation in the farthest direction of the compensation number	0-999999
1156	35	Motor 4 compensates for the farthest end of the positive direction	0-999999
1157	35	Motor 5 compensates for the farthest end of the positive direction	0-999999
1158	35	Motor 6 compensates for the farthest end of the positive direction	0-999999
feed shaft Z			
1000	1	Motor 1 is enabled 0: not enabled, 1: enabled	0 Or 1
1001	1	Is motor 2 enabled 0: not enabled, 1: enabled	0 Or 1
1002	1	Is motor 3 enabled 0: not enabled, 1: enabled	0 Or 1
1003	1	Is motor 4 enabled 0: not enabled, 1: enabled	0 Or 1

1004	1	Is motor 5 enabled 0: not enabled, 1: enabled	0 Or 1
1005	1	Is motor 6 enabled 0: not enabled, 1: enabled	0 Or 1
1027	0.0005	Motor 1 pulse equivalent	0-999999
1028	0.0010	Motor 2-pulse equivalent	0-999999
1029	0.0010	Motor 3-pulse equivalent	0-999999
1030	0.0010	Motor 4-pulse equivalent	0-999999
1031	0.0010	Motor 5-pulse equivalent	0-999999
1032	0.0010	Motor 6-pulse equivalent	0-999999
1279	1	Motor shaft 1 name 1: X 2: Y 3: Z 4: A 5: B 6: C 0: None	0-6
1280	2	Motor 2 shaft name 1: X 2: Y 3: Z 4: A 5: B 6: C 0: None	0-6
1281	3	Motor shaft 3 shaft name 1: X 2: Y 3: Z 4: A 5: B 6: C 0: None	0-6
1282	4	Motor 4 shaft name 1: X 2: Y 3: Z 4: A 5: B 6: C 0: None	0-6
1283	5	Motor 5 shaft name 1: X 2: Y 3: Z 4: A 5: B 6: C 0: None	0-6
1284	6	Motor shaft 6 shaft name 1: X 2: Y 3: Z 4: A 5: B 6: C 0: None	0-6
1018	0	Motor 1 direction 0: do not take reverse 1: take reverse	0 Or 1
1019	0	Motor 2 direction 0: do not take reverse 1: take reverse	0 Or 1
1020	0	Motor 3 direction 0: do not take reverse 1: take reverse	0 Or 1
1021	0	Motor 4 direction 0: do not take reverse 1: take reverse	0 Or 1
1022	0	Motor 5 direction 0: do not take reverse 1: take reverse	0 Or 1
1023	0	Motor 6 direction 0: do not take reverse 1: take reverse	0 Or 1
3356	0	Axft attribute 0 of motor 1: straight axis 1: angular axis	0 Or 1
3357	0	Axft attribute 0 of motor 2: straight axis 1: angular axis	0 Or 1
3358	0	Axft attribute 0 of motor 3: straight axis 1: angle axis	0 Or 1
3359	0	Axft attribute 0 of motor 4: straight axis 1: angular axis	0 Or 1
3360	0	Axft attribute 0 of motor 5: linear axis 1: angle axis	0 Or 1
3361	0	Axft attribute 0 of motor 6: linear axis 1: angle axis	0 Or 1
1477	0	Is motor 1 a rotating shaft 0: No 1: Yes	0 Or 1
1478	0	Is the motor 2 a rotating shaft 0: No 1: Yes	0 Or 1
1479	0	Is the motor 3 a rotating shaft 0: No 1: Yes	0 Or 1
1480	0	Is the motor 4 a rotating shaft 0: No 1: Yes	0 Or 1

1481	0	Is motor 5 a rotating shaft 0: No 1: Yes	0 Or 1
1482	0	Is motor 6 a rotating shaft 0: No 1: Yes	0 Or 1
1486	0	Number of pulses of motor 1	0-999999
1487	0	Number of pulses of motor 2	0-999999
1488	0	Number of pulses of motor 3	0-999999
1489	0	Number of pulses of motor 4	0-999999
1490	0	Number of pulses of motor 5	0-999999
1491	0	Number of pulses for a week of the motor 6 rotation	0-999999
1387	1	After motor 1, the machine coordinate 0: zero 1: no zero	0 Or 1
1388	1	After motor 2, machine coordinate 0: zero 1: no zero	0 Or 1
1389	1	After motor 3 power supply, machine coordinate 0: zero 1: no zero	0 Or 1
1390	1	After motor 4, the machine coordinate 0: zero 1: no zero	0 Or 1
1391	1	After motor 5, machine coordinate 0: zero 1: no zero	0 Or 1
1392	1	After motor 6 power, machine coordinate 0: zero 1: no zero	0 Or 1
1261	100	Motor 1 electronic gear ratio molecule	1-9999999
1262	100	Motor 2 electronic gear ratio molecule	1-9999999
1263	100	Motor 3 electronic gear ratio molecule	1-9999999
1264	100	Motor 4 electronic gear ratio molecule	1-9999999
1265	100	Motor 5 electronic gear ratio molecule	1-9999999
1266	100	Motor 6 electronic gear ratio molecule	1-9999999
1270	100	Motor 1 electronic gear score parent	1-9999999
1271	100	Motor 2 electronic gear score parent	1-9999999
1272	100	Motor 3 electronic gear score mother	1-9999999
1273	100	Motor 4 electronic gear score mother	1-9999999
1274	100	Motor 5 electronic gear score mother	1-9999999
1275	100	Motor 6 electronic gear score mother	1-9999999
1090	0.0005	Motor 1 handwheel or incremental minimum movement value (mm)	
1091	0.0010	Motor 2 handwheel or incremental minimum movement value (mm)	
1092	0.0010	Motor 3 handwheel or incremental minimum movement value (mm)	

1093	0.0010	Motor 4 handwheel or incremental minimum movement value (mm)	
1094	0.0010	Motor 5 handwheel or incremental minimum movement value (mm)	
1095	0.0010	Motor 6 handwheel or incremental minimum movement value (mm)	
1171	6000	Motor 1 hand wheel movement speed upper limit	0-999999
1172	6000	Upper movement speed of motor 2	0-999999
1173	6000	Upper limit of motor 3 handwheel movement speed	0-999999
1174	6000	Upper limit of movement speed of motor 4	0-999999
1175	6000	Upper limit of movement speed of motor 5	0-999999
1176	6000	Upper limit of handwheel movement speed of motor 6	0-999999
1180	0	Position cumulative limit for motor 1	0-999999
1181	0	Cumulative limit for motor 2	0-999999
1182	0	Cumulative limit value for the motor 3 handwheel position	0-999999
1183	0	Cumulative limit value for motor 4 handwheel position	0-999999
1184	0	Cumulative limit value for motor 5 handwheel position	0-999999
1185	0	Cumulative limits for motor 6 handwheel position	0-999999
1639	131072	Motor 1 absolute encoder reading per turn pulse	0-9999999
1640	131072	Motor 2 absolute encoder reading per turn pulse	0-9999999
1641	131072	Motor 3 absolute encoder reading per turn pulse	0-9999999
1642	131072	Motor 4 absolute encoder reading per turn pulse	0-9999999
1643	131072	Motor 5 absolute encoder reading per turn pulse	0-9999999
1644	131072	Motor 6 Absolute encoder reading per turn pulse	0-9999999
1648	0	Motor 1 absolute encoder reading to the command pulse conversion multiplier molecule	0-9999999
1649	0	Motor 2 absolute encoder reading to the command pulse conversion multiplier molecule	0-9999999
1650	0	Motor 3 absolute encoder reading to the command pulse conversion multiplier molecule	0-9999999
1651	0	Motor 4 absolute encoder reading to the command pulse conversion multiplier molecule	0-9999999
1652	0	Motor 5 absolute encoder reading to the command pulse conversion multiplier molecules	0-9999999
1653	0	Motor 6 absolute encoder reading to the command pulse conversion multiplier molecule	0-9999999
1657	131072	Motor 1 absolute encoder reading to the command impulse conversion multiplier denominator	0-9999999
1658	131072	Motor 2 absolute encoder reading to the command impulse conversion multiplier denominator	0-9999999
1659	131072	Motor 3 absolute encoder reading to the command impulse conversion multiplier denominator	0-9999999

1660	131072	Motor 4 absolute encoder reading to the command impulse conversion multiplier denominator	0-9999999
1661	131072	Motor 5 absolute encoder reading to the command impulse conversion multiplier denominator	0-9999999
1662	131072	Motor 6 absolute encoder reading to the command impulse conversion multiplier denominator	0-9999999
1666	0	Motor 1 absolute encoder reading	0 Or 1
1667	0	Motor 2 absolute encoder reading	0 Or 1
1668	0	Motor 3 absolute encoder reading is reversed	0 Or 1
1669	0	Motor 4 absolute encoder reading	0 Or 1
1670	0	Motor 5 absolute encoder reading	0 Or 1
1671	0	Motor 6 absolute encoder reading is reversed	0 Or 1
1675	0	The absolute value encoder position corresponding to the origin of the motor 1	-999999-999999
1676	0	The absolute value encoder position corresponding to the origin of the machine tool for the motor 2	-999999-999999
1677	0	Absolute encoder position corresponding to the origin of	-999999-999999
1678	0	Absol4 encoder position corresponding to the origin of the	-999999-999999
1679	0	Motor 5 The absolute value encoder position corresponding	-999999-999999
1680	0	Motor 6 The absolute value encoder position corresponding	-999999-999999
1684	0	Absolute position acquisition method for motor 1 (0: no 1: N protocol 2: Z protocol)	And 0 or 1 or 2
1685	0	Absolute position acquisition method of motor 2 (0: no 1: N protocol 2: Z protocol)	And 0 or 1 or 2
1686	0	Absolute position acquisition method of motor 3 (0: no 1: N protocol 2: Z protocol)	And 0 or 1 or 2
1687	0	Absolute position acquisition method of motor 4 (0: no 1: N protocol 2: Z protocol)	And 0 or 1 or 2
1688	0	Absolute position acquisition method of motor 5 (0: no 1: N protocol 2: Z protocol)	And 0 or 1 or 2
1689	0	Absolute position acquisition method for motor 6 (0: no 1: N protocol 2: Z protocol)	And 0 or 1 or 2
1837	0	Motor 1 absolute multiple ring displacement address	0-9999999
1838	0	Multi-loop displacement address of absolute motor 2	0-9999999
1839	0	Motor 3 absolute multiple loop displacement address	0-9999999
1840	0	Motor 4 absolute multiple loop displacement address	0-9999999
1841	0	Motor 5 absolute multiple loop displacement address	0-9999999
1842	0	Motor 6 absolute value multi-loop displacement address	0-9999999
1846	0	The absolute value of motor 1	0-9999999
1847	0	The absolute value of motor 2	0-9999999

1848	0	Single loop displacement address of absolute motor 3	0-9999999
1849	0	Single loop displacement address of absolute motor 4	0-9999999
1850	0	Single loop displacement address of absolute motor 5	0-9999999
1851	0	Single loop displacement address of absolute motor 6	0-9999999
0584	0	When X-axis PLC control, DI / D0 signal group, 0 is not enabled, range 0-4	0-4
0585	0	When Y-axis PLC control, DI / D0 signal group, 0 is not enabled, range 0-4	0-4
0586	0	For Z-axis PLC control, DI / D0 signal group, 0 is not enabled, range 0-4	0-4
0587	0	For A-axis PLC control, DI / D0 signal group, 0 is not enabled, range 0-4	0-4
0588	0	For B-axis PLC control, DI / D0 signal group, 0 is not enabled, range 0-4	0-4
0589	0	For C-axis PLC control, DI / D0 signal group, 0 is not enabled, range 0-4	0-4
1108	0	Motor 1 PLC control shaft name 0: Not enabled	0-6
1109	0	Motor 2 PLC control shaft name 0: Not enabled	0-6
1110	0	Motor 3 PLC control shaft name 0: Not enabled	0-6
1111	0	Motor 4 PLC control shaft name 0: Not enabled	0-6
1112	0	Motor 5 PLC control shaft name 0: Not enabled	0-6
1113	0	Motor 6 PLC control shaft name 0: Not enabled	0-6
1765	0	Motor 1 read parameter 0: Unsupported 1: support	0 Or 1
1766	0	Motor 2 read parameter 0: not supported 1: supported	0 Or 1
1767	0	Motor 3 read parameter 0: Unsupported 1: supported	0 Or 1
1768	0	Motor 4 read parameter 0: Unsupported 1: supported	0 Or 1
1769	0	Motor 5 read parameter 0: not supported 1: support	0 Or 1
1770	0	Motor 6 read parameter 0: Unsupported 1: supported	0 Or 1
1774	0	Motor 1 read information 0: Not supported 1: support	0 Or 1
1775	0	Motor 2 read information 0: Not supported 1: support	0 Or 1
1776	0	Motor 3 read information 0: Not supported 1: support	0 Or 1
1777	0	Motor 4 read information 0: Not supported 1: support	0 Or 1
1778	0	Motor 5 read information 0: Not supported 1: support	0 Or 1
1779	0	Motor 6 read information 0: Not supported 1: support	0 Or 1
1783	0	Motor 1 reads the high torque value	0-99
1784	0	Motor 2 reads the value of high torque	0-99

1785	0	Motor 3 reads the value of high torque	0-99
1786	0	Motor 4 reads the value of high torque	0-99
1787	0	Motor 5 reads the high torque value	0-99
1788	0	Motor 6 reads the value of high torque	0-99
1792	0	Motor 1 reads the alarm value of excessive torque	0-99
1793	0	Motor 2 reads the alarm value of excessive torque	0-99
1794	0	Motor 3 reads the alarm value of excessive torque	0-99
1795	0	Motor 4 reads the alarm value of excessive torque	0-99
1796	0	Motor 5 reads the alarm value of excessive torque	0-99
1797	0	Motor 6 reads the alarm value of excessive torque	0-99
0062	1	Whether to execute the coordinate reset 0 when power up: do not execute 1: execute	0 Or 1
principal axis M			
0551	0	The logical spindle 0 corresponds to the physical channel number	-1-9999999
0012	0	Spindle 0 DAC channel No	0-9999999
0011	4095	Spindle 0 DAC max	0-9999999
0513	400	Default spindle speed	0-9999999
0014	3000	Spindle 0 maximum speed limit	0-999999
0013	1500	Spindle speed at spindle 0 analog output (first gear)	0-999999
0043	2500	Spindle speed at maximum analog output of spindle 0 (second gear)	0-9999999
0044	3500	Spindle speed at spindle 0 analog output (third gear)	0-9999999
0045	4500	Spindle speed at maximum spindle 0 analog output (fourth gear)	0-9999999
0017	1	Spindle 0 Encoder 0: Unconnected 1: connected	0 Or 1
0018	0	Spindle 0 encoder channel number	0-9999999
0019	4096	Spindle 0 encoder resolution	0-9999999
0053	1	Spindle 0 encoder to spindle gear ratio (integer multiple)	0-9999999
0511	0	Reference feed motor number for spindle 0 G96	0-9999999
0512	50	Spindle 0 G96 constant line speed Speed of the spindle minimum speed	0-9999999
0552	-1	Logical spindle 1 corresponds to the physical channel No	-1-9999999
0023	1	Spindle 1 DAC channel no	0-9999999

0022	4095	Spindle 1 DAC max	0-9999999
0532	400	Default rotation speed of the spindle 1	0-9999999
0025	5000	Maximum speed limit of spindle 1	0-9999999
0024	1500	Spindle speed at spindle 1 analog output (first gear)	0-9999999
0026	0	Spindle 1 Encoder 0: Unconnected 1: connected	0 Or 1
0027	0	Spindle 1 encoder channel No	0-9999999
0028	1000	Spindle-1 encoder resolution	0-9999999
0064	0	The DAC value at the output voltage of the spindle 0 is 0V	0-9999999
0065	0	The DAC value when the output voltage of spindle 1 is 0V	0-9999999
3990	0	Whether the spindle 1 rotation speed is displayed	0 Or 1
reference point S			
0514	1	Return to reference point 0 before automatically running the program: required 1: Not required	0 Or 1
0063	0	Remove the return signal light during the emergency stop 0: No 1: Yes	0 Or 1
1216	0	Motor 1 return reference mode 0: with index 1: no 2: positioning 3: floating	0-3
1217	0	Motor 2 return reference mode 0: index 1: no 2: positioning 3: floating	0-3
1218	0	Motor 3 return reference mode 0: with index 1: no 2: positioning 3: floating	0-3
1219	0	Motor 4 return reference mode 0: with index 1: no 2: positioning 3: floating	0-3
1220	0	Motor 5 return reference mode 0: with index 1: no 2: positioning 3: floating	0-3
1221	0	Motor 6 return reference mode 0: with index 1: no 2: positioning 3: floating	0-3
1810	0	Motor 1 origin in switch Type 0: normally closed 1: normally open	0 Or 1
1811	0	Motor 2 Origin switch Type 0: normally closed 1: normally open	0 Or 1
1812	0	Motor 3 Origin switch Type 0: normally closed 1: normally open	0 Or 1
1813	0	Motor 4 Origin switch Type 0: normally closed 1: normally open	0 Or 1
1814	0	Motor 5 origin switch type 0: normally closed 1: normally open	0 Or 1
1815	0	Motor 6 Origin switch Type 0: normally closed 1: normally open	0 Or 1
1819	6000	Motor 1 fast movement speed limit, 100% rate fast movement	0-999999
1820	6000	Motor 2 return fast movement speed limit, 100% rate fast movement	0-999999
1821	6000	Motor 3 fast movement speed limit, 100% rate fast movement	0-999999
1822	6000	Motor 4 fast movement speed limit, 100% rate fast movement	0-999999
1823	6000	Motor 5 fast movement speed limit, 100% rate fast movement	0-999999

1824	6000	Motor 6 return fast shift speed limit, 100% rate fast movement	0-999999
1828	1000	Motor 1 return reference F0 rate block shift speed	0-999999
1829	1000	Motor 2 return reference F0 rate block shift speed	0-999999
1830	1000	Motor 3 return reference F0 rate block shift speed	0-999999
1831	1000	Motor 4 return reference F0 rate block shift speed	0-999999
1832	1000	Motor 5 return reference F0 rate block shift speed	0-999999
1833	1000	Motor 6 return reference F0 rate block shift speed	0-999999
1207	500	Motor 1 returns to the lowest speed of the reference point	0-999999
1208	500	Motor 2 returns to the lowest speed of the reference point	0-999999
1209	500	Motor 3 returns to the lowest speed of the reference point	0-999999
1210	500	Motor 4 returns to the lowest speed of the reference point	0-999999
1211	500	Motor 5 returns to the lowest speed of the reference point	0-999999
1212	500	Motor 6 returns to the lowest speed of the reference point	0-999999
1801	0	Return reference offset value of motor 1 (mm)	0-999999
1802	0	Return reference offset value of motor 2 (mm)	0-999999
1803	0	Motor 3 return offset value (mm)	0-999999
1804	0	Motor 4 return of offset value (mm)	0-999999
1805	0	Motor 5 return of offset value (mm)	0-999999
1806	0	Motor 6 return offset value (mm)	0-999999
1252	0	Coordinate value of machine tool after motor 1 (mm)	0-999999
1253	0	Machine coordinate value after motor 2 (mm)	0-999999
1254	0	Coordinate value of machine tool after motor 3 (mm)	0-999999
1255	0	Coordinate value of machine tool after motor 4 (mm)	0-999999
1256	0	Machine coordinate value after motor 5 (mm)	0-999999
1257	0	Coordinate value of machine tool after motor 6 (mm)	0-999999
1225	999999	Motor 1 forward direction software limit (mm)	-999999-999999
1226	999999	Motor 2 forward direction software limit (mm)	-999999-999999
1227	999999	Motor 3 forward direction software limit (mm)	-999999-999999
1228	999999	Motor 4 Positive direction software limit (mm)	-999999-999999

1229	999999	Motor 5 Positive direction software limit (mm)	-999999-999999
1230	999999	Motor 6 forward direction software limit (mm)	-999999-999999
1234	-999999	Motor 1 negative direction software limit (mm)	-999999-999999
1235	-999999	Motor 2 negative direction software limit (mm)	-999999-999999
1236	-999999	Motor 3 negative direction software limit (mm)	-999999-999999
1237	-999999	Motor 4 negative direction software limit (mm)	-999999-999999
1238	-999999	Motor 5 negative direction software limit (mm)	-999999-999999
1239	-999999	Motor 6 negative direction software limit (mm)	-999999-999999
1243	1	Motor 1 soft limit 0: effective after return reference 1: always valid	0 Or 1
1244	1	Motor 2 soft limit 0: effective after return reference 1: always valid	0 Or 1
1245	1	Motor 3 soft limit 0: effective after return reference 1: always valid	0 Or 1
1246	1	Motor 4 soft limit 0: effective after return reference 1: always valid	0 Or 1
1247	1	Motor 5 soft limit 0: effective after return reference 1: always valid	0 Or 1
1248	1	Motor 6 soft limit 0: effective after return reference 1: always valid	0 Or 1
3970	0	Second reference point coordinates on the X-axis	-999999-999999
3971	0	Second reference point coordinates of the Y-axis	-999999-999999
3972	0	Second reference point coordinates on the Z-axis	-999999-999999
3973	0	The second reference point coordinates of the A-axis	-999999-999999
3974	0	Second reference point coordinates on the B-axis	-999999-999999
3975	0	Second reference point coordinates of the C-axis	-999999-999999
3976	0	Third reference point coordinates on the X-axis	-999999-999999
3977	0	Third reference point coordinates on the Y-axis	-999999-999999
3978	0	Third reference point coordinates on the Z-axis	-999999-999999
3979	0	Third reference point coordinates on the A-axis	-999999-999999
3980	0	Third reference point coordinates in axis B	-999999-999999
3981	0	Third reference point coordinates in the C-axis	-999999-999999
3982	0	Fourth reference point coordinates of the X-axis	-999999-999999
3983	0	Fourth reference point coordinates of the Y-axis	-999999-999999
3984	0	Fourth reference point coordinates on the Z-axis	-999999-999999
3985	0	Fourth reference point coordinates on the A-axis	-999999-999999

3986	0	Fourth reference point coordinates on the B-axis	-999999-999999
3987	0	Fourth reference point coordinates in the C-axis	-999999-999999
linkage T			
0020	0	Emergency stop signal 0: normally closed 1: normally open	0 Or 1
1009	0	1 Motor servo alarm signal 0: normally closed 1: normally open	0 Or 1
1010	0	2 Motor servo alarm signal 0: normally closed 1: normally on	0 Or 1
1011	0	3 Motor servo alarm signal 0: normally closed 1: normally open	0 Or 1
1012	0	4 Motor servo alarm signal 0: normally closed 1: normally on	0 Or 1
1013	0	5 Motor servo alarm signal 0: normally closed 1: normally open	0 Or 1
1014	0	6 Motor servo alarm signal 0: normally closed 1: normally open	0 Or 1
0000	1	Encoder 1 interface 4 x enable 0: Invalid 1: valid	0 Or 1
0001	0	Encoder 2 interface 4 x enable 0: Invalid 1: valid	0 Or 1
0046	0	Encoder 1 direction 0: no reverse 1: reverse	0 Or 1
0047	0	2 encoder direction 0: do not take reverse 1: take reverse	0 Or 1
0066	0	The direction of encoder 3 is reversed	0 Or 1
0006	1	0: Increment 1: hand wheel	0 Or 1
0009	2	Encoder channel number entered by the hand wheel	0-999999
0515	150	Feed rate input maximum value	0-999999
3962	4	Spindle servo motor number 0: invalid	0-999999
3998	0	Electronic gearbox driven shaft motor number 0: Invalid	0-6
0054	1	Whether to enable modbus or not	0 Or 1
0055	19200	MODBUS Serial port port rate	0-9999999
0056	0	MODBUS Verification method (0= no check, 1= check, 2= even check)	And 0 or 1 or 2
0057	20	MODBUS Polling cycle	0-9999999
0058	500	Wait delay before sending the F-MA signal (CNC ready)	0-9999999
0059	200	MODBUS, Task execution cycle	1-9999999
0060	20	MODBUS, Conversion timeout time	1-9999999
0061	15	MODBUS, Response timeout time	1-9999999
0029	-1	The PLC-input X address for the IO-Link module 0 mapping	-999999-999999

0030	-1	The PLC input X address for the IO-Link Module 1 mapping	-999999-999999
0031	-1	The PLC input X address for the IO-Link Module 2 mapping	-999999-999999
0032	-1	The PLC input X address for the IO-Link Module 3 mapping	-999999-999999
0033	-1	The PLC-output Y address for the IO-Link module 0 mapping	-999999-999999
0034	-1	The PLC-output Y address of the IO-Link module 1 mapping	-999999-999999
0035	-1	The PLC output Y address of the IO-Link module 2 mapping	-999999-999999
0036	-1	The PLC-output Y address for the IO-Link module 3 mapping	-999999-999999
0037	5	IO-LINK allowed module NO response responses	0-999999
0038	20000	IO-LINK communication timeout time setting (ms)	0-999999
0039	1	IO-LINK is empty after timeout output 0: not void 1: empty	0 Or 1
0052	10	The IO-LINK execution cycle	0-999999
3005	1	Position Capture trigger signal (1-16: Universal input 0-15)	0-15
3006	0	Position capture trigger signal 0: Down edge trigger 1: Up edge trigger	0 Or 1
0068	2	Number of M2 bus slave stations	1-30
0069	0	Servo bus type 0: No 1: M2	0 Or 1
0070	0	Servo spindle command source 0: Pulse signal 1: bus	0 Or 1
0071	0	Whether to synchronize axis 0: Not enable 1: enable	0 Or 1
0072	0	Maximum allowable synchronization error (number of pulses)	0-999999
0073	0	Motor 4 is the spindle motor number from the control shaft	0-6
4004	1	Synchronous axis coordinates are hidden 0: not hidden 1: hidden	0 Or 1
1756	0	Motor 1 slave station address, 0: not connected 31: pulse	0-31
1757	0	Motor 2 slave station address, 0: not connected 31: pulse	0-31
1758	0	Motor 3 slave station address, 0: not connected 31: pulse	0-31
1759	0	Motor 4 from the station address, 0: not connected 31: pulse	0-31
1760	0	Motor 5 from the station address, 0: not connected 31: pulse	0-31
1761	0	Motor 6 from the station address, 0: not connected 31: pulse	0-31
other F			
3002	0	Number of counts after the system 0: no zero 1: zero	0 Or 1
0050	100	The servo enables to prohibit the delay time	0-999999
3215	0	08000-08999 on program protection 0: not on 1: on	0 Or 1

3216	0	09000-09999 on program protection 0: not on 1: on	0 Or 1
in common use O			
3002	0	Number of counts after the system 0: no zero 1: zero	0 Or 1